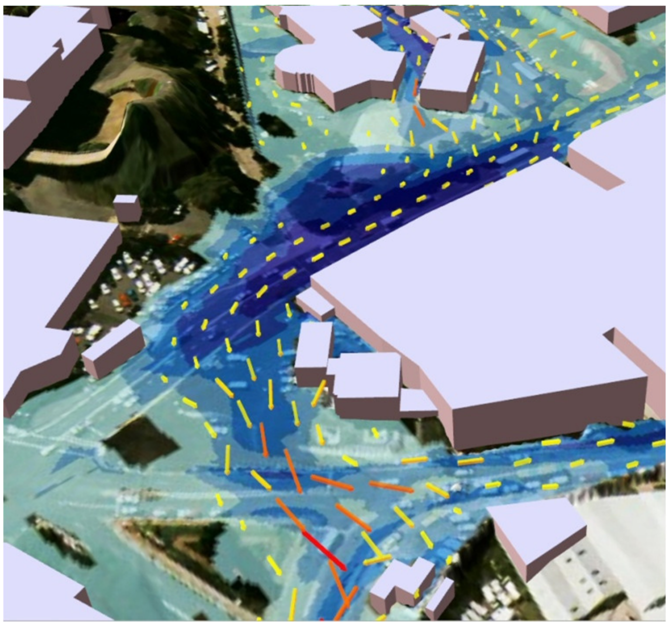


Australian Rainfall & Runoff

Revision Projects

PROJECT 15

Two Dimensional Modelling in Urban and Rural floodplains

STAGE 1&2 REPORT

P15/S1/009

NOVEMBER 2012

ENGINEERS AUSTRALIA
Water Engineering

Engineers Australia
Engineering House
11 National Circuit
Barton ACT 2600

Tel: (02) 6270 6528
Fax: (02) 6273 2358
Email:arr_admin@arr.org.au
Web: www.engineersaustralia.org.au

AUSTRALIAN RAINFALL AND RUNOFF
REVISION PROJECT 15: TWO DIMENSIONAL MODELLING IN URBAN AND RURAL FLOODPLAINS

STAGE 1 AND 2 DRAFT REPORT

NOVEMBER, 2012

Project Project 15: Two Dimensional Modelling in Urban and Rural Floodplains	**AR&R Report Number** P15/S1/009
Date 18 November 2012	**ISBN** 978-085825-9850
Editors Mark Babister (Editor) Cathie Barton (Editor)	**Verified by**

ACKNOWLEDGEMENTS

This project was made possible by funding from the Federal Government through the Department of Climate Change and Energy Efficiency. This report and the associated project are the result of a significant amount of in kind hours provided by Engineers Australia Members.

ENGINEERS
AUSTRALIA
Water Engineering

FOREWORD

AR&R Revision Process

Since its first publication in 1958, Australian Rainfall and Runoff (ARR) has remained one of the most influential and widely used guidelines published by Engineers Australia (EA). The current edition, published in 1987, retained the same level of national and international acclaim as its predecessors.

With nationwide applicability, balancing the varied climates of Australia, the information and the approaches presented in Australian Rainfall and Runoff are essential for policy decisions and projects involving:

- infrastructure such as roads, rail, airports, bridges, dams, stormwater and sewer systems;
- town planning;
- mining;
- developing flood management plans for urban and rural communities;
- flood warnings and flood emergency management;
- operation of regulated river systems; and
- prediction of extreme flood levels.

However, many of the practices recommended in the 1987 edition of AR&R now are becoming outdated, and no longer represent the accepted views of professionals, both in terms of technique and approach to water management. This fact, coupled with greater understanding of climate and climatic influences makes the securing of current and complete rainfall and streamflow data and expansion of focus from flood events to the full spectrum of flows and rainfall events, crucial to maintaining an adequate knowledge of the processes that govern Australian rainfall and streamflow in the broadest sense, allowing better management, policy and planning decisions to be made.

One of the major responsibilities of the National Committee on Water Engineering of Engineers Australia is the periodic revision of ARR. A recent and significant development has been that the revision of ARR has been identified as a priority in the Council of Australian Governments endorsed National Adaptation Framework for Climate Change.

The update will be completed in three stages. Twenty one revision projects have been identified and will be undertaken with the aim of filling knowledge gaps. Of these 21 projects, ten projects commenced in Stage 1 and an additional 9 projects commenced in Stage 2. The remaining two projects will commence in Stage 3. The outcomes of the projects will assist the ARR Editorial Team with the compiling and writing of chapters in the revised ARR.

Steering and Technical Committees have been established to assist the ARR Editorial Team in guiding the projects to achieve desired outcomes. Funding for Stages 1 and 2 of the ARR revision projects has been provided by the Federal Department of Climate Change and Energy Efficiency. Funding for Stages 2 and 3 of Project 1 (Development of Intensity-Frequency-Duration information across Australia) has been provided by the Bureau of Meteorology.

Project 15 Two Dimensional Modelling

At the time that the 1987 Edition of Australian Rainfall and Runoff was prepared, the use one-dimensional hydrodynamic models for assessment of flooding in riverine and urban systems was an emerging area with two dimensional models being computationally impractical for real world problems. Since that time the situation has changed considerably with technological advances enabling 2D models to become the tool of choice for most hydraulic flood assessments.

For this reason it is necessary to provide guidance on the use of 2D models as part of the updating of ARR.

This project aims to provide guidance to not just modellers, but to those who commission studies and use model results. This document has been prepared in a collaborative approach by a team of Australian industry experts in the field of two dimensional hydrodynamic modelling. While the document has been independently peer reviewed it is recognized by the project team that many aspects could be covered in more detail. It is also recognized in two dimensional hydrodynamic modelling that practice and technology can advance very quickly. For this reason the document will be reviewed after a 3 month period of industry comment. It is also recognized that this document will need periodic updating.

Mark Babister
Chair Technical Committee for
ARR Research Projects

Assoc Prof James Ball
ARR Editor

AR&R REVISION PROJECTS

The 21 AR&R revision projects are listed below:

ARR Project No.	Project Title	Starting Stage
1	Development of intensity-frequency-duration information across Australia	1
2	Spatial patterns of rainfall	2
3	Temporal pattern of rainfall	2
4	Continuous rainfall sequences at a point	1
5	Regional flood methods	1
6	Loss models for catchment simulation	2
7	Baseflow for catchment simulation	1
8	Use of continuous simulation for design flow determination	2
9	Urban drainage system hydraulics	1
10	Appropriate safety criteria for people	1
11	Blockage of hydraulic structures	1
12	Selection of an approach	2
13	Rational Method developments	1
14	Large to extreme floods in urban areas	3
15	**Two-dimensional (2D) modelling**	**1**
16	Storm patterns for use in design events	2
17	Channel loss models	2
18	Interaction of coastal processes and severe weather events	1
19	Selection of climate change boundary conditions	3
20	Risk assessment and design life	2
21	IT Delivery and Communication Strategies	2

AR&R Technical Committee:

Chair: Mark Babister, WMAwater

Members: Associate Professor James Ball, Editor AR&R, UTS
Professor George Kuczera, University of Newcastle
Professor Martin Lambert, Chair NCWE, University of Adelaide
Dr Rory Nathan, SKM
Dr Bill Weeks, Department of Transport and Main Roads, Qld
Associate Professor Ashish Sharma, UNSW
Dr Bryson Bates, CSIRO
Steve Finlay, Engineers Australia

Related Appointments:
ARR Project Engineer: Monique Retallick, WMAwater
Assisting TC on Technical Matters: Dr Michael Leonard, University of Adelaide

PROJECT TEAM

Main contributors:

- Mark Babister , Editor and AR&R TC manager, WMAwater
- James Ball, UTS
- Cathie Barton, Editor, BMT WBM,
- Warwick Bishop, Water Technology
- Steve Gray, WMAwater
- Rhys Hardwick Jones, WMAwater
- Andrew McCowan, Water Technology
- John Murtagh, NSW OEH
- Bill Peirson, UNSW, WRL
- Brett Phillips, Cardno
- Ted Rigby, Rienco
- Monique Retallick, WMAwater
- Grantley Smith, UNSW WRL
- Bill Syme, BMT WBM
- Stefan Szylkarski, DHI
- Rhys Thompson, Cardno
- Bill Weeks, QLD Department of Transport and Main Roads

and thank you to:

- Roger Austin, Wallingford Software
- Martin Lambert, University of Adelaide
- John Macintosh, Water Solutions
- Steve Muncaster, DSE
- Habib Rehman, Hydrostorm
- Chris Thomas, Worley Parsons
- Connie Kotze and Amelia Grace for the Graphic artwork, Cardno

Note: All project team members provided their time on an in kind basis

ABBREVIATIONS

TERM	DEFINITION
1D	One-dimensional
2D	Two-dimensional
3D	Three-dimensional
CFL	Courant–Friedrichs–Lewy
CFD	Computational Fluid Dynamics
AEP	Annual Exceedance Probability
ARI	Annual Recurrence Interval
ARR	Australian Rainfall & Runoff
DEM	Digital Elevation Model
DTM	Digital Terrain Model
LIDAR	Light Detection And Ranging

GLOSSARY

TERM	DESCRIPTION
1D domain	Those parts of the model that are 1D.
1D element	A specific part of the 1D domain.
1D/2D models	A model incorporating both 1D and 2D domains.
2D domain	Those parts of the model that are 2D.
2D element	A specific part of the 2D domain.
Accuracy	The degree of closeness of measurement results to the true value.
Advection & Dispersion	Dispersion is the net mass flux along a concentration gradient and advection is the flux due to the fluids bulk motion.
Annual Exceedance Probability (AEP)	The chance of a flood of a given or larger size occurring in any one year, usually expressed as a percentage. For example, if a peak flood discharge of 500 m^3/s has an AEP of 5%, it means that there is a 5% chance (that is one-in-20 chance) of a 500 m^3/s or larger event occurring in any one year (see ARI).
Average Recurrence Interval (ARI)	The long term average number of years between the occurrence of a flood as big as, or larger than, the selected event. For example, floods with a discharge as great as, or greater than, the 20 year ARI flood event will occur on average once every 20 years. ARI is another way of expressing the likelihood of occurrence of a flood event. Not a preferred term because it misleads the public about risk.
Arithmetic Equations	Discrete equations used within a numerical model.
Boundary	A condition that is required to be satisfied at all or part of the boundary of a region in which a set of differential equations is to be solved.
Branched-1D	A 1D model that has branched 1D channels. Sometimes called Quasi-2D.
Breaklines	Breaklines are survey strings used to define continuous features. In relation to 2D modelling, they are used to define floodplain features such as levees, embankment and channels, that need to be specifically included in the DTM and the hydraulic model due to their critical hydraulic importance.
Computational Fluid Dynamics (CFD)	A branch of fluid mechanics that uses numerical methods and algorithms to solve nonlinear partial differential equations governing viscous fluid flows.
Compound Channel	Also known as a two-stage channels are channels that are flanked on either or both sides by floodplains.
Computational Model	Computer program, run on a single computer, or a network of computers, that attempts to simulate physical processes through mathematical representations.
Cross-section	A survey string that is taken perpendicular to the main flow direction in

	a river.
DEM	Digital Elevation Model: A simplification of a DTM, being a structured raster or regular grid of 3D coordinate values x, y, and z.
Direct Rainfall	The application of rainfall directly onto the 2D domain. This is usually used in place of a hydrologic model.
DTM	Digital Terrain Model: An unstructured network of elements of variable size and sometimes shape that define the elevation of the terrain. (refer Chapter 5).
Empirical Equations	Equations derived from observation and experimentation rather than mathematical representation of a physical system.
Evaporation	Water is transferred from the surface to the atmosphere through evaporation, the process by which water changes from a liquid to a gas.
Evapotranspiration	The sum of evaporation and plant transpiration from the Earth's land surface to the atmosphere.
Event Time	Length of an event run within a model (not the time it takes to run the model).
Explicit/Implicit	Explicit and implicit methods are approaches used in numerical analysis for obtaining numerical solutions of time-dependent differential equations. Explicit methods calculate the state of a system at a later time from the state of the system at the current time, while implicit methods find a solution by solving an equation involving both the current state of the system and the later one.
Fixed Grid	A uniform, structured grid used to define the 2D domain of a numerical model.
Flexible Mesh	A non-uniform, unstructured mesh used to define the 2D domain of a numerical model.
Grid	See fixed grid.
Grid/Mesh	This document refers to the uniform square or rectangular elements of a finite difference model as a grid and the quadrilateral, triangular or curvilinear elements of the flexible mesh models (such as finite element and finite volume models) as a mesh. Statements that are applicable to both types use the grid/mesh format of reference.
Hydraulic Hazard	A source of potential harm or a situation with a potential to cause loss. In relation to this manual the hazard is flooding which has the potential to cause damage to the community.
Hydraulics	Term given to the study of water flow in waterways; in particular, the evaluation of flow parameters such as water level and velocity.
Hydraulic Model	A computational model that is able to describe or represent the motion of water.
Hydrodynamic Model	This document uses the terms "hydrodynamic model" and "hydraulic model" interchangeably. Refer to hydraulic model for definition.
Implicit	See explicit.
In-bank	When the event is contained within the banks of the main channel.
Infiltration	The penetration of water through the ground surface into the sub-surface soil.
Interception	The part of the rainfall that is intercepted by the earth's surface and which subsequently evaporates. In this definition the earth's surface includes everything that becomes wet after a rainfall event and that dries out soon after. It includes: vegetation, soil surface, litter, build-up surface, etc.
LiDAR	An optical remote sensing technology that can measure the terrain surfaces for hydraulic modelling studies. High-resolution digital elevation models are generated by airborne LIDAR.
Long-Section	Terrain and water surface information that is taken from a line drawn along the main flow direction, usually the channel centreline.
Magnitude	Size or extent.
Manning's 'n'	Manning's 'n' is an estimate of channel roughness, used in the calculation of flow velocity and discharge. The value of n varies with channel slope, bed material composition, in-stream vegetation and channel sinuosity.

Mathematical Equations	Continuous equations used to represent a physical system.
Mathematical Model	The mathematical representation of the physical processes involved in runoff generation and stream flow. These models are often run on computers due to the complexity of the mathematical relationships between runoff, stream flow and the distribution of flows across the floodplain.
Mesh	See flexible mesh.
Modeller	The person using the model.
Numerical Model	A computer model using arithmetic equations, also called a computational model.
Numerical Transients	A sudden shock or change in the internal boundary conditions that is generated by the numerical scheme or the schematisation that would not be present in the real world. A typical example is as the element changes from wet to dry or the reverse, the solution changes while in the real world this is often a gradual process and the location of the element transitions though a stage of being partially wet but the numerical scheme only allows for an element to be fully wet or fully dry.
Photogrammetry	The process of making maps or scale drawings with elevations from stereo photographs, especially aerial photographs.
Physical Model	A physical model is a scaled representation of a hydraulic flow situation.
Practitioner	The person working in the field of hydraulic engineering or floodplain management.
Precision	The degree of closeness of measurement results to each other. Precision is the repeatability or reproducibility of a measurement.
Quasi-2D	A 1D model that has branched 1D channels. More correctly called a Branched 1D model.
Rating Ratio	The ratio between the highest observed flow and highest gauge flow.
Resolution	In relation to a fixed grid or a flexible mesh: used to describe the size of the elements or cells of the grid or mesh.
Run Time	Time taken to run a model.
Severity	A dimension for classifying seriousness.
St Venant Equations	Also called the shallow water equations, they are derived from depth-integrating the Navier–Stokes equations.
Transpiration	Process by which water that is absorbed by plants, usually through the roots, is evaporated into the atmosphere from the plant surface, such as leaf pores.
Verification	The process of evaluating the model set-up usually by testing against an event that was not used during the calibration phase.
Wetting and Drying	The process whereby elements of the 2D domain become numerically wet and then eventually dry again during an event simulation.

TABLE OF CONTENTS

TABLE OF FIGURES

TABLE OF TABLES

1. CHAPTER 1 - INTRODUCTION

1.1. Introduction

Over the past 5 years, two-dimensional (2D) hydrodynamic models have increasingly become the standard approach for predicting design flood behaviour. In a 2D model the flow solution is based on the numerical solution of the full 2D depth-averaged equations of motion computed at each active water grid point. The ARR revision process identified that 2D modelling was often being carried out in isolation from the rest of the design flood estimation process and that there was a need for more detailed guidance on the use of two-dimensional (2D) hydrodynamic models for the prediction of design flood parameters. The aim of ARR Research Project 15 Stage 1 is to provide a review of current practice in 2D hydrodynamic simulation including identification of those areas where current practice is not supported by either theoretical or empirical research. This document contains the outcomes of the review and, in detailing current practice and supporting background, it provides guidance on appropriate development and usage of 2D hydrodynamic models.

1.2. Scope

The objectives of the first Stage of Project 15 are to consolidate existing knowledge and practice, to highlight deficiencies in the existing knowledge and practice, and to highlight deficiencies in the available information supporting current practice. The clear deliverables defined by the funding agreement with Department of Climate Change and Energy Efficiency (DCC, 2008) are:

> *"15.1 Assessment of two dimensional models and approaches for simulation of events in urban areas.*
>
> *15.2 Development of a methodology for assessing the performance of these models.*
>
> *15.3 A report detailing the above work including a detailed scoping of future stages."*

In considering these deliverables, the Project 15 team formed the view that:

1. There was an absence of documentation detailing "state-of-the-art" approaches related to the practice of 2D hydrodynamic modelling, not just in urban areas;

2. The industry needed a document outlining accepted practice where that practice was supported by appropriate research and development; and

3. The deliverables required as part of the funding agreement with Department of Climate Change would be met best through provision of a document filling this gap.

The aim of this report is to consolidate industry practice and provide clear guidance on the establishment and use of hydrodynamic models in rural and urban environments. The focus of Stage 2 of Project 15 is to fill the knowledge gaps identified in Stage 1 and to provide more detailed information on some emerging areas of practice.

Consistent with the approach adopted elsewhere in ARR Research Projects and the ARR Revision Project itself, references to specific software packages are minimised to avoid any implied endorsement of a specific software package, which may inhibit future research and development. Strength and weaknesses of various conceptual approaches are documented allowing a user to select an approach that best suits their situation. Further research by the informed user on the conceptual approach employed by the alternative software packages, will then allow the user to select the appropriate package. As software packages are constantly evolving, this method allows the user to access the latest specifications and capabilities of each software package at the time of need.

Approaches to 2D hydrodynamic modelling presented herein are intentionally not prescriptive but rather provide a basis for informed choice by a modeller This is consistent with the philosophy of ARR whereby a description of approaches, their reliability and suitability for different situations, are presented together to allow the user to gain a better insight into each approach. This lack of prescription enables the modeller to use engineering judgement to select the model approach that is most appropriate for their specific project.

1.3. 2D Hydraulic Modelling – Past & Present

Hydrodynamic modelling is a field of engineering hydrology that has advanced rapidly in recent decades, largely as a result of dramatic increases in computational resources. Early efforts in field in the 1950s occurred as part of research projects undertaken primarily by academics and other researchers, rather than practitioners. A primary requirement for this research was access to “state-of-the-art” computer systems. By the 1970s and 1980s, one-dimensional (1D) models were being applied to standard engineering hydraulic problems by both academics and practitioners and two-dimensional (2D) hydrodynamic models were being applied to estuarine and port circulation problems. The field of computational hydraulics was firmly established. Currently, (at 2012), there are several mature 1D and 2D hydraulic modelling software packages commercially available. With rapid parallel advancement in techniques for computer resources, data acquisition, and the representation and analysis of spatial information, 2D computational hydraulic modelling has become a standard approach to a wide and increasing variety of engineering problems.

Applications where 2D hydrodynamic models are currently commonly employed include:

- Estimating flood depths, velocities, extents and flood hazards resulting from elevated rainfall and/or stream flows in a catchment;
- Coastal inundation resulting from tides and/or storm surge;
- Design of drainage systems and structures;
- Estimating the consequences of dam failure;
- Analysis of flood impacts due to development within a catchment/floodplain;
- Evaluation and design of flood mitigation strategies (such as levees or sea-walls); and
- Emergency response planning including evacuation strategies.

Additionally, there are a number of other problems, such as water resource management, typically characterised by lower flow volumes and increased importance of loss processes (such as seepage and evapotranspiration) relative to flood assessments.

1.4. 2D Hydraulic Modelling – The Future

With continual advances in computational hydraulics, exponential increases in computing power, ongoing-research activities and the additional challenges presented by climate change, aspects of this document are likely to be out of date at the time of publication. Nonetheless, there is a need to predict potential developments and future problems related to 2D hydrodynamic modelling systems. At the time of publication, a considerable amount of research is being undertaken in many of the areas identified in Chapters 9 and 10. It is expected that further areas and issues requiring research will evolve with time.

While finite-difference and finite-element techniques are currently the most widely used in flood modelling, significant research into finite-volume techniques and alternative computational frameworks (for example, curvilinear grids and Lagrangian grids) is being undertaken both by research institutions and software suppliers. Outcomes from these research activities are likely to change current practice in hydrodynamic modelling of flood behaviour. It is expected that the progression toward finite volume approaches will be very rapid in the coming years. Furthermore, computational speeds may reach a point where 3D Computational Fluid Dynamics (CFD) approaches become a viable tool for hydraulic modelling.

Irrespective of future changes to some of the more specific details presented herein, the broad tenets governing the major steps of data acquisition, model creation, calibration and interpretation of model predictions are expected to remain valid into the foreseeable future.

1.5. Report Overview

This document is structured to provide a useful resource for the modelling practitioner, with information and guidance relating to major phases of the modelling process being provided in separate chapters of this report.

Chapters 2 to 4 of the report identify the steps taken in the modelling process:

- Chapter 2 guides the user through the selection of hydrodynamic model types with 1D, quasi 2D, 2D, combined 1D/2D or 3D considered. The advantages and disadvantages of each type are outlined enabling the modeller to make an informed choice.
- Chapter 3 contains discussion on 2D modelling approaches and detail on the governing equations. A description of initial conditions, boundary conditions, model assumptions, implicit and explicit schemes is provided.
- Chapter 4 outlines the modelling process, including the definition of the problem, methods for selecting the most appropriate approach, data collection, model development, model calibration, the interpretation of model results, 1D/2D modelling, current issues and direct rainfall. Chapter 4 is intended to provide the modeller with a series of considerations to be addressed prior to undertaking a 2D modelling assessment, and to assist in the selection of a suitable approach and tools. This chapter may be useful for those preparing project briefs and proposals where hydraulic modelling may be required, or during the early planning

stages on an analysis.

Chapters 5 to 9 provide further detail on the steps involved in the modelling process. In these chapters the practitioner can find a summary of best practice approaches and potential pitfalls for mainstream hydraulic modelling applications:

- Chapter 5 outlines the data required when developing a 2D model. This includes coordinate systems, data management and filtering data checking, Digital Terrain Model (DTM) specifications, accuracies of DTM data (LiDAR, hydrosurvey, and ground survey) and systematic bias, and issues commonly encountered with data.
- Chapter 6 covers the model development phase including selecting and appropriate model type, mesh/grid type, boundary conditions, study extent, run times and computer space requirements, and the identification of key hydraulic features.
- Chapter 7 deals with model calibration and uncertainty. It includes the danger of calibrating to small events and extrapolating to large events, checking design storms conform to observed behaviour, using Manning's 'n' values out of the standard ranges to compensate for other model development issues and calibration versus accuracy. Several detailed examples of model calibrations on real catchments are included in the report.
- Chapter 8 covers the interpretation of model results. When reviewing the modelling results, the modeller should consider the model limitations based on an understanding of the assumptions made in model development and the limitations of the model approach. Care should be taken when using a catchment wide model for localised analysis. Guidance is given on result types and the associated uncertainty, model limitations, what constitutes a real impact when comparing different scenarios, sensitivity testing guidelines, and hydraulic hazard.
- Chapter 9 includes modelling of combined 1D/2D systems, how 1D networks connect with 2D grids, how structures are modelled, duplication of energy losses through structures, and momentum transfer (or lack of transfer) between 1D and 2D components of a model.
- Chapter 10 contains discussion on a number of emerging applications or current known issues in 2D modelling. The applications covered in this Chapter are those in which the modeller is advised to proceed with caution, as they are either areas where there are significant limitations with current approaches, or where there are multiple techniques that may be considered for a given application. Whilst advice is offered in the sub-sections of Chapter 10, such advice should be considered no substitute for acceptable, project specific, model calibration/validation, which is always the final word on the validity of any given modelling approach. Some of the issues dealt with are recommended areas for further research. These topics are evolving fields and, as such, current best practice may have evolved since the publication of this document (and thus past the practice described herein). Topics include: Supercritical flow, adaptive timesteps/run times, wetting/drying, valid parameter ranges/envelopes, eddy viscosity, very shallow and very deep flow, numerical precision and accuracy issues, sub-grid/mesh features, structures, and buildings.
- Chapter 11 deals with the emerging practice of using direct rainfall, describing its mechanisms and concepts. Direct rainfall when used in 2D hydrodynamic models replaces the rainfall-runoff processes that have traditionally been modelled by separate hydrological models.

A summary of commercial and academic distributors of 2D hydrodynamic modelling software at the time of writing is provided in Appendix A. The list is not exhaustive. Furthermore, inclusion in the list definitely does not indicate any endorsement or otherwise of the software packages.

1.6. Fundamental Advice

Within the details of this report, there are some broad fundamental themes that all modellers should be aware of and should regularly remind themselves of:

- All models are coarse simplifications of very complex processes. No model can therefore be perfect, and no model can represent all of the important processes accurately;
- Model accuracy and reliability will always be limited by the accuracy of the terrain and other input data;
- Model accuracy and reliability will always be limited by the reliability/uncertainty of the inflow data;
- A poorly constructed model can usually be calibrated to the observed data but will perform poorly in events both larger and smaller than the calibration data set,
- No model is “correct” therefore the results require interpretation,
- A model developed for a specific purpose is probably unsuitable for another purpose without modification, adjustment, and recalibration. The responsibility must always remain with the modeller to determine whether the model is suitable for a given problem, and
- New software packages should first be validated on simple problems based on fundamental hydraulic principles with known solutions, before they are used on complex real world problems.

1.7. References

DCC, 2008, Funding Agreement in Relation to funding for *Stage 1 of the Project to Revise the Australian Rainfall & Runoff Handbook*

2. CHAPTER 2 - 2D CONCEPTUALISATION

2.1. Introduction

The aim of a numerical hydraulic modelling exercise is typically to provide a realistic representation of flow behaviour in a given environment. The hydraulic model provides not only the opportunity to replicate historical flow events but also to predict flow behaviour under different hydrologic conditions or an altered physical environment (that is, impact analysis).

Prior to the advent of computers, hydraulic modelling of river and floodplain flows, and of the behaviour of hydraulic structures, could only be carried out using scaled physical hydraulic models. These models would typically be set-up in large laboratory facilities, of which only a few existed at Universities, Research Institutes or Public Utilities. Due to the time and costs involved, physical models could only be justified for major projects, over relatively small reaches of river or floodplain.

Although the basic equations governing free-surface flow in rivers and floodplains were derived in the 19^{th} Century, it was not until the development of computers in the 1960's and 70's that numerical modelling of channelised flows became practical. With the rapid on-going development of computers and computing power, there has been a continual evolution of numerical modelling tools and modelling techniques. This has resulted in the availability of a wide range of numerical models with increasing capability and complexity.

A brief history of the development of hydraulic models, as well as commonly applied hydraulic models in Australia at the time of writing is given in Appendix A.

In general, it can be said that the more realistic the modelling approach, the greater the probability of achieving a successful outcome. That is, numerical models that are capable of representing complex and variable surface flow processes are more likely to produce reliable results than simplified approaches. However, use of the most sophisticated modelling approach available will not, in itself, guarantee success. This is because the skill of the modeller adapting a generic modelling system to a specific application, and the quality of the data used as model input can be equally important in determining model success. Indeed, there are many applications in which simplified approaches, suitably applied, can be more appropriate (and more efficient and cost effective) than the use of more sophisticated models.

It is the broad understanding of a particular area of hydraulic interest and the translation of this into an appropriate model framework in terms of model type, extent and resolution that defines the essence of Model Conceptualisation.

2.2. An Overview of the Modelling Approach

2.2.1. Development Stages for Numerical Hydraulic Models

The aim of a numerical hydraulic model is to provide a discretised representation of the river and floodplain system in order to mimic flow behaviour. In this respect, the development of a site-specific numerical model comprises a sequence of four main steps (also shown schematically in Figure 2-1):

1. Review and define the physical system (the river and/or floodplain system to be modelled)

2. Select an appropriate mathematical model (the set of equations used to describe the physical system)

3. Select a generic numerical model (the modelling software used to solve the equations)

4. Develop the site-specific numerical model (the generic modelling software combined with site-specific inputs, including topographic data, bed-friction coefficients, flow boundary conditions and other parameters such as pipe or culvert information as appropriate)

At each step, different types of assumptions, approximations and/or simplifications must be made. The steps are discussed briefly below for numerical hydraulic models with further supporting information provided in the following sections. The conceptualisation process is shown schematically in **Error! Reference source not found.**.

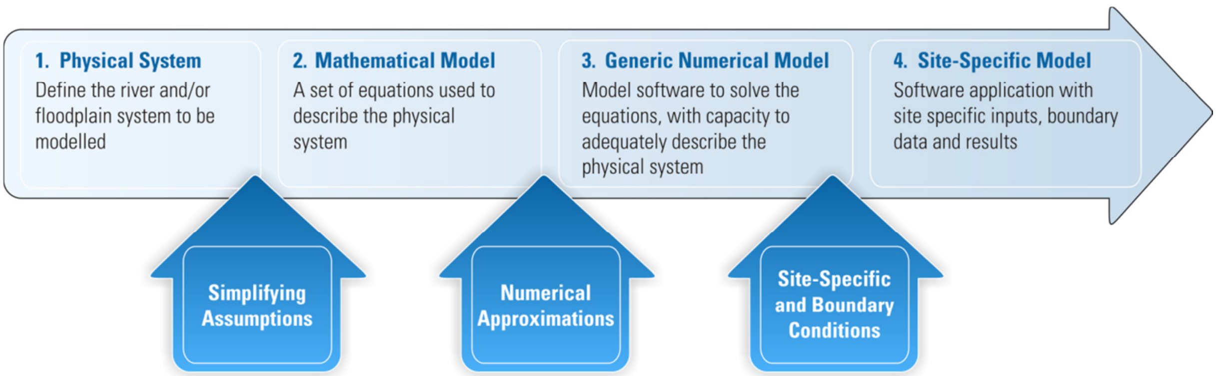


Figure 2-1 Stages in Numerical Hydraulic Model Conceptualisation and Development

2.2.1.1. Review of the Physical System

As a minimum, a broad understanding of the hydraulic behaviour of the physical system in question is essential in order to be able to make an informed choice on the most appropriate mathematical model. Whilst it is common that detailed hydraulic behaviour of the system is unknown, a good working knowledge of the study area and catchment is needed. Aspects such as study area shape, elevation (is the subject land low-lying or close to the sea?) and slope is important. The number and size of hydraulic structures and channel or drainage dimensions should be understood. Land-use is also an extremely important element of the physical system to consider. The distribution of roads, number and type of buildings if present, vegetation extent and composition may each affect the conceptualisation and selections made in later steps.

2.2.1.2. **Selection of the Mathematical Model**

Selection of the appropriate type of mathematical model is the most critical decision in the four step process. In this step the physical system flow behaviour, which can commonly involve complex highly turbulent flows, must be reduced to an equation, or set of equations, describing the main characteristics of the flow. Here assumptions have to be made as to whether the flow can be considered as being one-dimensional (1D), two-dimensional (2D), or a combination of both, and whether the flow can be described as being steady (i.e. constant with time), or unsteady (time-varying). In virtually all rural or urban floodplain modelling, vertical accelerations in the flow field are considered to be negligible and a hydrostatic pressure distribution is assumed, with computations and results based around a depth-averaged velocity. Further details are provided in Chapter 3, which outlines the governing equations utilised in hydraulic models.

2.2.1.3. **Selection of the Numerical Model**

The numerical hydraulic model approximates the continuous mathematical equation(s) of the mathematical model with discrete arithmetic equations. This process is generally carried out within a generic numerical model or modelling software package. The errors introduced by the discretisation process are called “truncation” errors and tend to reduce with the cross-section spacing (1D models) or grid/mesh resolution (2D models) and/or timestep. That is, the smaller the spacing/grid/mesh resolution and timestep, the smaller the truncation errors. In some models, low-order dissipative numerical truncation errors are deliberately introduced, or can be introduced as an option, to help stabilise the numerical computation. Further details on this are provided in Chapter 3.

It is important to understand the distinction between models that are typically referred to as “full 2D” numerical models and those that are of a lesser standard or capability. A full 2D scheme for the purposes of urban and rural flood modelling is accepted to be any model that numerically represents the complete depth-averaged St Venant or shallow water free-surface wave equations (as described in Chapter 3). Simplified numerical representations of the 2D free-surface wave equations can be appropriate in many situations, however the modeller should be aware of the limitations of such schemes. Further, the modeller should also be confident that any limiting assumptions will remain valid over the entire scope of the modelling project of interest, in terms of the range of potential geometries and flows that will be investigated.

2.2.1.4. **Development of the Site-Specific Model**

The site-specific model is developed using the generic numerical hydraulic model (software package) through the selection of:

- a modelling domain;
- cross-section spacing (1D) or grid/mesh resolution (2D) and timestep;

- the input of site-specific data including topographic data (cross-sections and/or topography) and bed-friction data; and
- the application of flow and/or water level boundary conditions.

The site-specific model must then be calibrated to ensure it is capable of reproducing flow behaviour at the subject site. The detailed model development process is termed Model Schematisation, which is described further in Chapter 6, followed by model calibration in Chapter 7.

2.3. Types of Hydraulic Models Available

Hydraulic modelling, in whatever environment it may be applied, essentially aims to represent realistic flow behaviour. In various situations, different types of flow may dominate hydraulic behaviour in a floodplain, and, for computational efficiency, some flow characteristics can be ignored or assumed to be simplified. Most hydraulic models apply a form of the St Venant equations (see Chapter 3 for details on hydrodynamic model schemes in 1D and 2D). During the period up to the early 1990's, when computing power was limited relative to today, abbreviated forms of the St Venant equations were often used. This included models based on the kinematic and diffusive wave approximations (in both 1D and 2D) and it was necessary for the modeller to be sure that the form of simplified equations used matched the flow conditions being modelled. For example, in a well-confined valley with reasonable slope and limited floodplain storage, the kinematic wave approximation is considered a reasonable approach. In some cases, for example where tidal influence is important, only the full dynamic approximation is appropriate. With the significant improvements in desktop computing power (and reduced costs) available from the mid-1990's onwards, most hydraulic models (1D and 2D) now utilise the full dynamic form of the St Venant equations. It is important to also recognise that the St Venant equations are subject to important assumptions and are not applicable in all situations. For instance, a hydrostatic pressure distribution is assumed to exist through the water column, which does not allow for vertical accelerations. Hence, detailed flow behaviour over a weir crest is not able to be explicitly modelled with these equations; however this limitation has been overcome in most hydraulic models through the use of empirical relationships to represent these discrete flow situations.

Due to different assumptions and applied numerical schemes, a significant difference still exists between the input data requirements, computational effort and resulting outputs of the various modelling approaches currently available. As such, it is important to select the approach that is best suited to the application. Further, it is imperative that the flow situation being modelled does not contravene the assumptions/limitations of the modelling system being used to reproduce it.

Given the efficiency of current numerical schemes and the computing power available, it is recommended that models incorporating the full 1D and/or 2D description are utilised for typical flooding applications.

To provide guidance on when a 2D approach is appropriate or necessary, information is presented in Table 2-1 on the range of hydraulic modelling options available and the general applications to which each is suited.

Table 2-1 Hydraulic model approaches

Model	Structure	Typical Application
1D Models	With these models the main channel and floodplain of a waterway is schematised as a single 1D channel, comprising a series of spaced cross-sections.	The use of 1D models is generally restricted to modelling single waterway branches, or simply connected (dendritic) channel systems, where flow in the floodplain is well connected to the main channel. Due to their inherent limitations, 1D models have generally been replaced by more flexible 1D branched, full 2D or combined 1D/2D models.
Branched 1D Models	These models allow arbitrary connections of multiple channel systems, and are an evolutionary development of simpler 1D models. Floodplains can be represented as separate flowpaths and there can be multiple flowpaths within a single floodplain. This provides a more realistic description of flows through a street network for example. It is noted that within each branch or flowpath the flow is represented by the one-dimensional cross-sectionally averaged equations of motion.	These models are sometimes referred to as quasi-2D models, but should not be confused with genuine 2D models. These models can be applied where flowpaths are well defined and clear controls exist between flowpaths.
2D Models	With these models survey information for the study area is projected onto a 2D model grid or mesh. Grids may be a square or rectangular, as typically the case in finite difference models. A mesh may be variable-sized quadrilaterals, triangles or of curvilinear nature, as is typically the case for finite element or finite volume models. The flow solution is based on the numerical solution of the full 2D depth-averaged equations of motion computed at each active computational element.	Full 2D models are capable of providing a detailed description of the flow in urban or rural floodplains and overbank areas. Full 2D models are more computationally demanding than 1D models. This may be a factor when considering long simulations or real-time forecasting applications. In addition, fixed grid models may have problems in providing adequate resolution of in-bank flows.
Integrated 1D/2D Models	With these models the main channel(s) and/or structures (such as culverts, bridges or pipe networks) are described by the 1D domain that is connected	These integrated models aim to provide a more comprehensive, efficient and accurate representation of a hydraulic system by making the most of both branched 1D and full 2D model

Model	Structure	Typical Application
	dynamically to the 2D domain of the overbank area. There can be a number of independent 1D domains within the overall integrated model.	capabilities.
3D Models	These models are similar to 2D models; however there is the opportunity for non-uniform vertical velocity profiles. This allows for the computation of 3D phenomena such as wind circulation in shallow areas or density-stratification within the water column.	These models are typically not used in urban or rural flood situations, as flow depth is too shallow and/or velocities too great to develop stratified conditions. Other 3D cases such as weir flow occur at a scale too small to resolve in a typical flood model

In general, 1D models are applicable when flowpaths are well defined and the length scale of the flow process in question is much greater than the width. This is typically the case for long lengths of in-bank flow in channels and rivers. For example, a 10 km length of river that is say 100 m wide will have a length/width ratio of 100:1.

2D model domains are applicable when flowpaths are poorly defined (typically in areas with flat terrain), the length scale and width scale are of the same order or hydraulic details of flow patterns such as eddies and separation zones are of interest. A typical application where 2D modelling is appropriate could comprise a section of urban floodplain that is 5 km wide over a length of 15 km. In this case, the length to width ratio is only 3:1 and hence a two-dimensional hydraulic modelling approach is justified.

Another way of considering the distinction between the two main types of overland flow models is that for the long river example mentioned above, the flow can be considered to be constrained in the lateral direction and flow can only freely move in one direction (upstream/downstream). Hence, this case is well suited to a 1D modelling approach. For the case of a flat, wide floodplain, flood flows can be visualised as being spread-out and able to freely move both down the floodplain but also laterally, across the floodplain. In this case, the longitudinal and lateral movement of flood waters would be best described as flow in two-dimensions (horizontally) and a 2D model would be most appropriate.

Apart from general overland flow characteristics, the other main distinction between 1D and 2D models is the definition of hydraulic structures such as weirs and culverts. These are more readily implemented in a one-dimensional form, as the complex local hydraulic properties at these locations often contravene basic assumptions of the numerical hydraulic scheme (such as the hydrostatic pressure distribution, which does not hold for flow over weirs). For most hydraulic structures (such as weirs, culverts and bridges), the discretised St Venant equations are replaced by empirical relations developed over decades of hydraulic observation and experimentation. Over the past decade, there have been significant developments in the integration of 1D and 2D models to provide better representation of floodplain features (the detailed attributes of 1D/2D models are discussed in Chapter 9). Subsequently, 1D hydraulic structure descriptions can generally be nested within either a 1D or 2D model. Hence, the treatment of hydraulic structures should not greatly affect the selection of the approach most appropriate for a given application.

2.3.1. Advantages/Disadvantages of the 2D Approach

The main advantage of a full 2D model is that it can provide a realistic description of the flows throughout a floodplain system. When compared with 1D or branched-1D models discussed above, it can be said that a full 2D model has the following **advantages**:

- Floodplain flowpaths do not need to be pre-determined by the modeller, as they are computed directly as a function of the model terrain and the applied flows.
- Flowpaths can change with changes in water level in much the same way as they do in reality.
- Within an urban context, cross-momentum of flow splits at road intersections is accounted for, thus providing a far more realistic representation of the dynamics of flow shedding across a pavement at a road junction. This can have a significant impact on flow splits in the road reserve downstream of an intersection.
- Losses due to two-dimensional effects such as bends and flow separations are automatically included within the computation, and do not need to be accounted for by increasing the roughness parameter (as such, the bed-friction coefficients can be specified more directly as a function of bed-roughness), or energy loss factors.
- Model results can provide details of the flow distribution within individual flowpaths.
- Model results can be used directly for mapping flood extents and inundation depths, velocities and safety hazard (flux).

Conversely, a full 2D model has the following **disadvantages,** when compared with 1D or branched-1D models:

- Significantly more survey data are required. The magnitude of this disadvantage has declined over recent time as aerial survey techniques have improved and become more cost-effective, allowing survey data to be collected over large areas. Aerial Laser Survey (ALS or LiDAR) is an example of such a technique (refer to Chapter 5 for more details on these techniques).
- Significantly more computation time is required. Even with the power of modern desktop computers, 2D model simulations can take many hours and even days to complete, depending on the model extent, grid/mesh[1] resolution and event time. This is simply due to the increase in the number of model elements (1D compared to 2D) requiring a longer computation time to resolve.
- The 2D approach usually requires a trade-off between the total number of grid/mesh elements (determined by the grid/mesh-resolution and model extent) and run time, particularly for fixed grid models. As the model domain increases in area, grid and *average*

---

[1] This document refers to the uniform square or rectangular elements of a finite difference model as a **grid** and the quadrilateral, triangular or curvilinear elements of the flexible mesh models (such as finite element and finite volume models) as a **mesh**. Statements that are applicable to both types use the “grid/mesh” format.

mesh resolution correspondingly needs to decrease in order to maintain the total number of grid/mesh elements and avoid excessive run times. Thus, the *average* topographic resolution in 2D models can diminish due to run time concerns. In general, flexible mesh models are less impacted by these resolution issues, as they are able to support greater mesh resolution in areas that require a more detailed topographic description, while compensating for this by expanding to a coarser mesh resolution in areas that do not (due to their ability to incorporate variable mesh resolution). Fixed grid models are not able to compensate in such a way and the main creek and/or river channel(s) can only be resolved at the same scale as the model grid. As a result, in-bank flows in a fixed grid model may not be described well compared to a branched-1D model with detailed cross-sections, depending on the grid resolution. However, many software packages are developing an ability to accommodate multiple fixed or even temporally varying resolution grid elements (as in nested grid and adaptive tessellation models). These new developments will make fixed grid models less constrained by resolution versus computation time issues. Issues related to the selection of an appropriate model grid/mesh resolution are described further in Chapter 6, Model Schematisation.

Advantages and disadvantages of a 2D modelling approach are summarised in Table 2-2. In general, for higher accuracy and greater spatial extents, a full 2D model will be appropriate. This will in turn increase run times, data processing time and data storage requirements. Conversely, if less accuracy is needed and faster run times are important, a 1D approach may be desirable (such as for rapid assessments or real-time forecasting).

Table 2-2 Advantages and Disadvantages of 1D or 2D modelling approach

Features	Advantages	Disadvantages
1D models		
•Series of linked channels with discrete cross-sections at regular intervals (say every 100 - 1000 m) •Output at each cross-section can include water level, depth and velocity (averages)	•Relatively fast to run (run time typically < 1 hour) •Can be time consuming to build, but relatively quick to modify •Result files are relatively small	•Requires cross-sections to be input to model, extracted either from field survey or DEM •Can be time consuming to build, but relatively quick to modify •Requires more interpolation and interpretation of results
2D models		
•Detailed grid or mesh-based topography with element resolutions for an urban environment typically ranging from 1m to 10m. For more extensive floodplain environments, element resolution can typically range from 10m to 100m. •Output at each grid/mesh element can include water level, depth and velocity	•Less interpolation of results required and more readily linked to GIS •Modeller is not required to identify flowpaths in advance •Can model complex flowpaths •Floodplain storage is implicitly defined •Inputs and outputs defined spatially in GIS type environments, results in better data continuity and more readily accessible/understandable results for community/client	•Requires detailed grid/mesh to be interpolated from aerial and/or field survey based DEM (plus roughness mapping over catchment) •Can be time consuming to build, modifications often not as easy as for 1D •Relatively slow to run (run times typically range from hours to days) •Result files are relatively large (up to GB per simulation) •Can in some cases instil over-confidence in the result that may not be justified if the underlying data are inadequate

Figure 2-2 and Figure 2-3 provide examples of a typical 2D model topography and simulation results respectively.

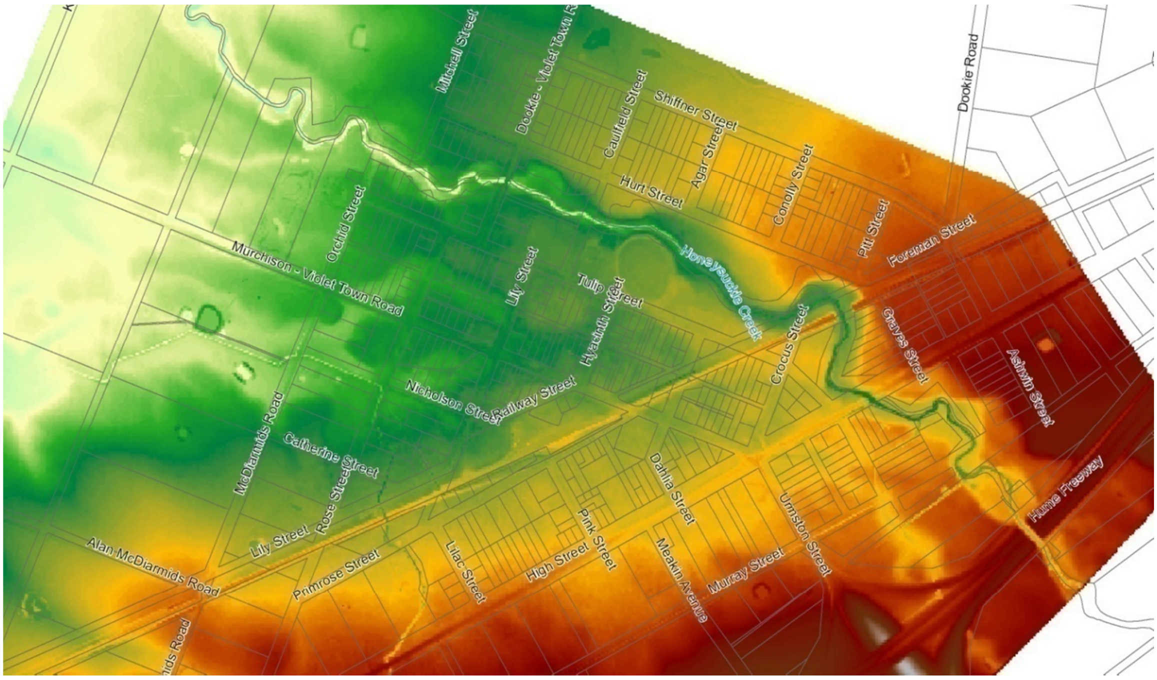


Figure 2-2 Violet Town 2D Model Topography

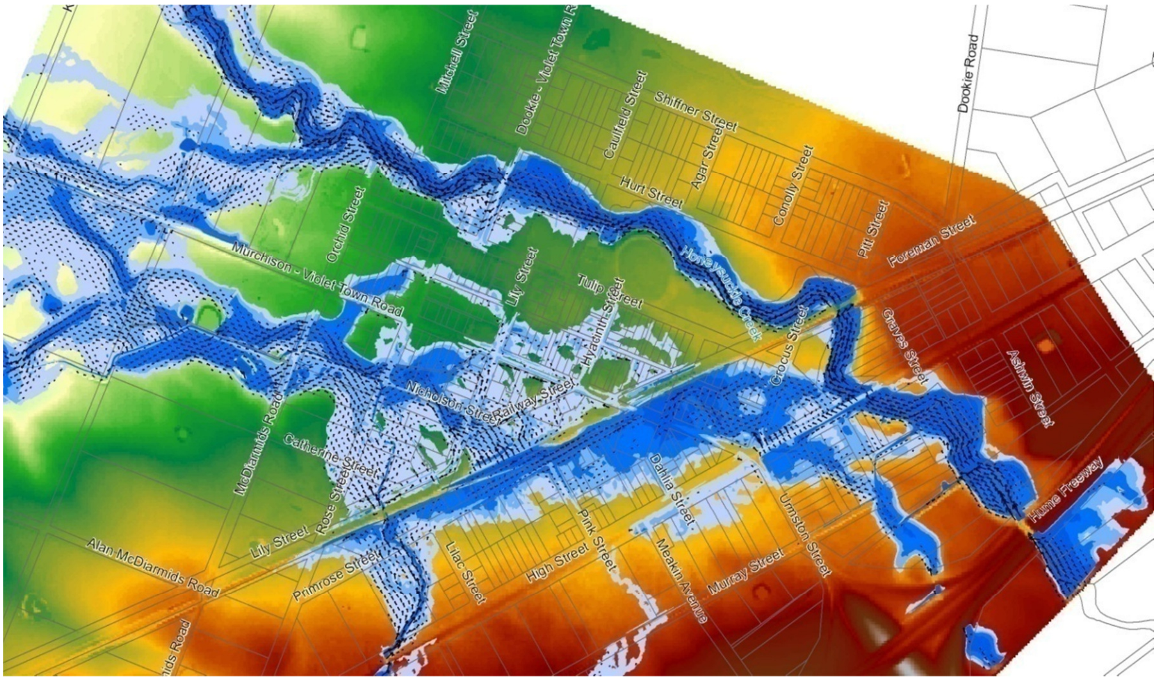


Figure 2-3 **Violet Town Flood Depths and Velocity Vectors**

2.4. 2D Application Areas

Currently, 2D models are applied to many different situations at a variety of resolutions and over areas ranging from small (say 100 square metres) to extremely large (1000's square kilometres). A non-exhaustive list of applications is provided below:

- Floodplain studies – inclusive of flood studies all the way through to mitigation impact assessment;
- Flood Emergency Response – Model results can be used to enable emergency services to better prepare and respond to flood events by identifying potential flood hazard and planning evacuation routes. Model outputs can also enhance mapping outputs and improve flood intelligence for both responsible agencies and the community, leading to a reduction in flood impacts. Whilst not commonly used at present, it is possible that 2D models may be utilised more commonly for real-time flood warning in the future.
- Urban drainage studies – in such applications the hydraulic model may also perform the routing functionality typically carried out by a hydrologic model. The 2D model provides the "major" drainage layer and interfaces also with the "minor" drainage system (i.e. pits and pipes) dynamically;
- Dam Break assessments - Often a 2D model is used to route dam break hydrographs. 2D models are well-suited to this application as the flowpaths resulting from a dam break are often unexpected or different to typical flowpaths. It is important to clarify that 2D models are generally not able to effectively replicate dambreak outflow from the collapsing face but can effectively model flood behaviour on the floodplain downstream of the break;
- In any environment in order to assess the flood impact due to development;
- In-bank river flow modelling where grid/mesh resolution is sufficiently fine. This may be

carried out in order to provide flow velocity that varies over the cross-section as opposed to 1D models in which velocity will be averaged over the cross-section. This approach is often used in ecosystem/habitat assessment;

- Wetland modelling - where routing paths are ill-defined and filling and draining processes are complex.
- Lake or estuary studies – often at the lower end of river systems the floodplain interacts with a lake or estuary and subsequently ocean or lake dynamics become important (tide, storm surge, or seiching).
- Water quality and sediment transport studies – these applications build on the two-dimensional hydrodynamics to provide information on water-dependent processes such and pollutant transport and river morphology.

2.5. 2D Floodplain Model Developments

Early 2D floodplain models (from around the beginning of the 1990's) were typically adaptations of existing ocean or coastal models and as such were not well adapted to computing shallow flows over land. Compared to ocean conditions, floodplain flow is shallower with higher velocities and a much greater proportion of the model area that wets and dries. Further, floodplain models tend to be dominated by friction forces (i.e., gravity flow down a slope) whereas most ocean models are dominated by inertial forces (i.e., tidal exchange with an almost flat water surface). Added to this, is the extra complexity of physical obstructions on the floodplain such as banks and hydraulic structures. Hence over time, 2D models have evolved to incorporate more robust wetting and drying schemes, capacity to explicitly model channels and hydraulic structures through 1D linking, and dynamic coupling with pipe networks to allow full major/minor urban drainage system analysis.

A more recent development in 2D and integrated 1D/2D models has been the use of direct rainfall to estimate flows, typically in small catchments. This technique is generally applied in situations where the study/model area forms a significant proportion (or sometimes all) of the contributing catchment. Under these circumstances traditional hydrologic approaches; such as the Rational Method, or Rainfall-Runoff models require the (often arbitrary) definition of sub-catchment inflow points within the hydraulic model area. In flat areas, this may be difficult to define and the location selected to apply discrete catchment inflows can significantly influence the hydraulic model results.

By applying a "rainfall excess" directly to the hydraulic model grid/mesh (or by incorporating a simple loss accounting scheme within the model) it is possible to simulate the rainfall/runoff process, as well as hydraulic routing of the resulting overland flows throughout the model area. This potentially provides a more realistic representation of catchment storage effects and distribution of surface runoff. However, such an outcome is not guaranteed and it is essential to have good topographic data, and appropriate roughness coefficients to the successfully use this approach (Muncaster et al, 2006).

The use of a 2D hydraulic model in this way integrates the hydrologic and hydraulic aspects of rainfall-runoff processes into a single model. There are however a number of uncertainties in this process, particularly as these techniques have not been well researched or validated at the current time. For example, there are unresolved questions with respect to the relationship between grid/mesh resolution and the influence of sub-grid/mesh scale processes both in terms of bed friction and catchment storage. Further research and development is required to provide a better description of the hydrologic losses involved, and to assess the role of bed roughness in determining surface runoff rates. Section 11 provides a more detailed description on the processes and issues related to direct rainfall methods.

2.6. CFD and Physical Modelling

In some complex flow situations where the 2D free-surface modelling assumptions are not valid and the generally available 2D free-surface flow models will not provide sufficiently accurate answers. Such situations may include:

- Highly localised flow effects in the immediate vicinity of features that are sub-grid/mesh in scale. For example, piers in a floodway or estuary;

- Highly localised flow effects downstream of an overflow structure/weir where the hydrostatic pressure assumption is violated; and
- A series of structures where transitions from sub- to super- to sub-critical flow are likely over an extended flow distance (St Venant equations will approximate super-critical flow only and in all likelihood the modelled location of the hydraulic jump will be in error).

In such instances, it may be that an alternative approach is required. The options available are Computational Fluid Dynamics (CFD) modelling or physical scale modelling. These are described in the following sections.

2.6.1.1. Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics (CFD) models are appropriate to use under conditions when the boundaries are regular and well-formed, which in general does not include most civil and environmental open channel flows. Note also that wave breaking and surface rupture is common in open channel flows and CFD investigations of such processes remain in the research domain. CFD models may be utilised to investigate particular isolated hydraulic structure problems such as dam spillways and flows through gross pollutant traps. These models are not practical to apply at the floodplain scale.

2.6.1.2. Physical Models

Physical models are able to provide reliable flow/discharge relationships for situations in which numerical model results cannot be verified. This includes situations in which the flow assumptions for open channel flow are no longer valid, such as flow through a customised hydraulic structure for which no reliable empirical data are available. A scaled prototype of the real-world structure/topography is typically built within a controlled laboratory environment comprising a hydraulic flume and water delivery system. The scale model is subject to flows and results are measured. The real-scale hydraulic performance is calculated through similitude in which results can be scaled up from the model to full scale. Physical models are generally considered expensive relative to numerical models; however, they do have a place in certain specialist applications.

2.7. References

Muncaster, S. H. et al, 2006, ‘Design flood estimation in small catchments using two-dimensional hydraulic modelling – A case study’, *Proceedings of 30th Hydrology and Water Resources Symposium*.

3. CHAPTER 3 – FUNDAMENTALS

3.1. Introduction

This Chapter aims to provide the modeller with an ability to look beyond comparisons of proprietary models and instead focus on the model fundamentals. Various 2D models available to the modeller are based on variations (and in some cases, abbreviations) of the St Venant Equations, utilise different numerical methods to solve the equations and/or utilise a different discretisation or solving approach (e.g. finite difference or finite element and explicit or implicit. This Chapter will provide insight into how a model's overall "numerical scheme" may impact on its range of applicability and on the strengths and weaknesses of the various schemes.

Governing equations for both the 2D and 1D unsteady flow models are provided throughout the text. Where possible, comments on strengths and weaknesses of the various schemes are referred to in terms of these equations. This will allow the modeller to gain an appreciation of the fundamental basis of the various schemes, and to gain insight into how a specific scheme may perform in a specific situation.

3.2. Background

The hydraulics of flood modelling is complex. The significant transitions in open channel flow that can occur will determine the patterns of flood behaviour. Further, the locations of the important rapidly varied flow features can shift during a flood, further complicating the flood modelling process.

Appropriately-scaled physical models deal directly with transitions in open channel flow. However, numerical models use various forms of the fundamental governing equations of fluid flow, subject to previously-determined assumptions and parameterisations. It is crucial to the flood modelling process that the fundamental equations, and the inherent assumptions associated with their application, be adequately understood.

It is impossible in a few pages to address all the mathematical, numerical and fluid mechanical aspects of the flood modelling process, as this requires many years of advanced training. The Chapter presents a brief introduction to:

- the fundamental equations used in conventional flood models,
- reconciliation of the different forms of the governing equations,
- numerical representation of the governing equations (including explicit and implicit time representation),
- key assumptions often used during the flood modelling process,
- hybrid approaches to flood modelling,
- incorporation of boundary conditions,
- the application of Computational Fluid Dynamics approaches, and
- future developments in numerical flood modelling approaches.

3.3. 2D Equations of Motion

Fully 2D hydrodynamic models are based on the numerical solution of depth-averaged equations describing the conservation of mass and momentum in two horizontal dimensions *x* and *y*.

In a form used by many of the commonly used 2D models, these equations can expressed in terms of three main dependent variables; ς*, u and v*, as shown in Figure 3.1.

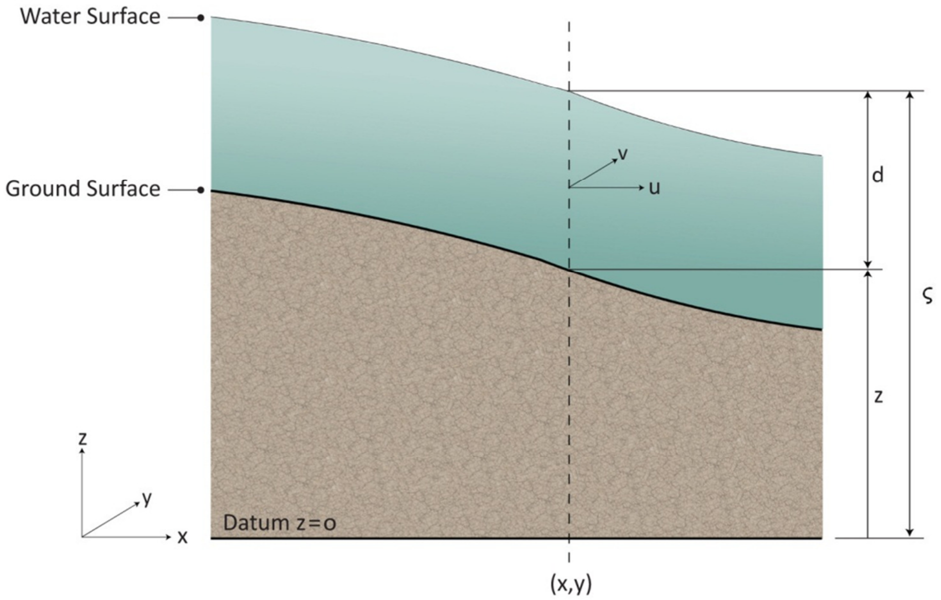


Figure 3-1 Definition of Symbols

Where:

- ς: is the water surface elevation relative to a fixed datum *(m).*
- *u*: is the depth-averaged velocity in the *x* direction *(m/s)*
- *v*: is the depth-averaged velocity in the *y* direction *(m/s)*

These are described as a function of the three main independent variables:

- *x:* the horizontal distance in the x direction (*m*)
- *y:* the horizontal distance in the y direction (*m*)
- *t:* the time (*s*)

Additionally, the time varying water depth at any location *d(x,y),* can be expressed as:

$$d = \varsigma - z$$

where:

- *z*: is the bed surface elevation relative to a fixed datum *(m).*

3.3.1. The Mass Equation

For flooding applications, water can be considered to be incompressible. As such, water volume can be used to represent the water mass. In terms of the variables described above, the depth-averaged equation describing the conservation of volume (and therefore mass) in two horizontal directions can be expressed as:

$$\frac{\partial \varsigma}{\partial t} + \frac{\partial (d.u)}{\partial x} + \frac{\partial (d.v)}{\partial \mathrm{y}} = 0$$

Where:

$\frac{\partial \varsigma}{\partial t}$*:* is the rate of increase (or decrease) in water level, which for a fixed cell size is representative of the rate of change of volume of water contained in the cell, and

$\frac{\partial (d.u)}{\partial x} + \frac{\partial (d.v)}{\partial \mathrm{y}}$ *:* is the spatial variation in inflow (or outflow) across the cell in the *x* and *y* directions.

Simply put, any increase (or decrease) in volume, must be balanced by a net inflow (or outflow) of water.

3.3.2. The Momentum Equations

In a similar form, the equations for describing the conservation of momentum in the *x* and *y* directions can be expressed as:

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + g\frac{\partial \varsigma}{\partial x} = 0$$

$$\frac{\partial v}{\partial t} + v\frac{\partial v}{\partial y} + u\frac{\partial v}{\partial x} + g\frac{\partial \varsigma}{\partial y} = 0$$

where:

g*:* is the acceleration due to gravity (m/s^2)

The equations presented above are in the primitive, Eulerian form. The same equations can exist in other forms; e.g. the conservation law form (Abbott, 1979) and the conservative-integral form (Leveque, 2002).

Due to the symmetry between the two *x* and *y* momentum equations, further discussion will be focussed on the x-momentum equation only.

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + g\frac{\partial \varsigma}{\partial x} = 0$$

where:

$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y}$*:* is the partial differential form of the flow acceleration *du/dt*

$g\dfrac{\partial\varsigma}{\partial x}$: is the hydrostatic pressure gradient

It can be shown that the momentum equation is effectively an impulse/momentum equation, where the flow acceleration, that is, the rate of increase (or decrease) in momentum is balanced by the impulse of the hydrostatic pressure gradient.

3.3.3. Assumptions

In the derivation of the these equations, it has been assumed that:

- The flow is incompressible
- The pressure is hydrostatic (i.e. vertical accelerations can be neglected and the local pressure is dependent only on the local depth).
- The flow can be described by continuous (differentiable) functions of ς, *u* and *v* (that is, it does not include step changes in ς, *u* and *v)*..
- The flow is two-dimensional (that is, the effects of vertical variations in the flow velocity can be neglected).
- The flow is nearly horizontal (that is, the average channel bed slope is small).
- The effects of bed friction can be included through resistance laws (e.g., Manning's equation) that have been derived for steady flow conditions.

3.4. Properties of the Equations

The main properties of the equations of motion have an important bearing on the requirements of the numerical solution procedures that are used to solve them. In this respect, it is noted that the basic 2D equations described above are sometimes called the "long wave" or "shallow water wave" equations. They describe the propagation of waves where the wave length is very much longer than the water depth.

Additionally, it is noted that the momentum equation includes terms consistent with a transport (or advection) equation. These are discussed briefly below.

3.4.1. Propagation Properties of the Equations

To discuss the wave propagation properties of the equations of motion it is simpler to consider the 1D uniform-depth versions of the equations. In 1D, the mass equation presented in Section 3.3.1, becomes:

$$\frac{\partial\varsigma}{\partial t} + \frac{\partial(d.u)}{\partial x} = 0$$

Which for uniform depth, simplifies to:

$$\frac{\partial\varsigma}{\partial t} + d\frac{\partial .u}{\partial x} = 0$$

The corresponding 1D version of the momentum equation presented in Section 3.3.2, is:

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + g\frac{\partial \varsigma}{\partial x} = 0$$

For low velocity flows, the $u\frac{\partial u}{\partial x}$ convective momentum term can be neglected to provide the 1D "linearised" momentum equation:

$$\frac{\partial u}{\partial t} + g\frac{\partial \varsigma}{\partial x} = 0$$

Differentiating the uniform depth mass equation with respect to time *t* and differentiating the linearised momentum equation with respect to distance *x*, multiplying it by the depth *d*, and subtracting, results in:

$$\frac{\partial^2 \varsigma}{\partial t^2} - gd\frac{\partial^2 \varsigma}{\partial x^2} = 0$$

This is the classic 1D linear long-wave equation. It describes waves propagating in the positive and negative *x* directions with characteristic shallow water wave celerities of $\pm\sqrt{gd}$. This has important implications in determining the solution procedure to be used, the boundary conditions required, and the rate at which information can propagate through a model domain.

3.4.2. Transport Properties of Momentum Equation

The first three terms of the momentum equation represent the acceleration of the flow, *du/dt*, which when expanded into partial differential form becomes:

$$\frac{du}{dt} = \frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y}$$

where:

$\frac{du}{dt}$*:* represents the local acceleration of the flow at a fixed point. That is, the local rate of increase (or decrease) in velocity, and

$u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y}$*:* represents the advected acceleration of the flow. That is, the acceleration that is caused by the transport (or advection) of the flow across spatial gradients in velocity (momentum). These terms are commonly called the "convective momentum" terms

In isolation, the three acceleration terms in the momentum equation would form the main part of a 2D transport (or advection) equation describing the transport of velocity (or momentum), where:

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = 0$$

That is, when the velocities and/or the spatial variation of the velocities are high, the acceleration component of the momentum equation displays similar properties to those of a 2D transport equation. This can have important implications in determining the numerical solution procedure to be used as numerical schemes that are good at solving the main wave propagation components of the equations are not necessarily good at solving the transport components.

Problems associated with accurate modelling of transport equations have been highlighted by Leonard (1979a). Simple first order schemes are inaccurate and diffusive, while second order schemes (that are good for solving wave propagation) tend to be oscillatory and unstable. This has lead to the use of more innovative approaches to modelling the convective momentum terms (e.g., Abbott and Rasmussen, 1977), and the use of higher third order solution schemes (e.g., Leonard, 1979b, Stelling, 1984).

3.5. Extension of the Equations for Modelling Applications

The 2D mass and momentum equations described in Sections 3.3.1 and 3.3.2 are sometimes referred to as the 2D “long wave” equations. These equations can be used to describe the behaviour of waves, including flood waves, which are long relative to the water depth.

For practical modelling applications, these equations need to be expanded to include the additional effects of other phenomena of interest. The most important of these is probably the inclusion of the dissipative effects of bed-friction in the momentum equation. The inclusion of additional terms to form extended modelling equations is considered below.

3.5.1. Extension of the Mass Equation

For modelling applications, the mass equation can be expanded to include additional source and/or sink terms to allow for localised and/or distributed inflows and outflows, as follows:

$$\frac{\partial \varsigma}{\partial t} + \frac{\partial (d.u)}{\partial x} + \frac{\partial (d.v)}{\partial \mathrm{y}} = Sources - Sinks$$

Where the *Source* terms can represent localised inflows such as may occur at stormwater or pump outlets, or distributed inflows associated with rainfall, and the *Sink* terms can represent localised outflows at drainage pits or pump intakes or distributed losses due to infiltration or, in long-term simulations, evaporation.

3.5.2. Extension of the Momentum Equations

For modelling applications, the extension of the momentum equations to include the effects of bed-friction, eddy viscosity and other source and sink terms is discussed below.

Bed Friction

For flood modelling applications, the momentum equation must be coupled with a suitable friction formulation. This is typically achieved by adding a Chezy-type friction term to the momentum equation, which then becomes:

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + g\frac{\partial \varsigma}{\partial x} = -\frac{gu\sqrt{u^2 + v^2}}{C^2 d}$$

where:

C: is a Chezy roughness coefficient ($m^{1/2}s^{-1}$)

For practical modelling applications, the Chezy coefficient can be related to the more usual (for Australian applications) Manning's 'n' by the Strickler relation, where:

$$n = \frac{d^{1/6}}{C}$$

In some European models, the friction coefficient is sometimes specified in terms of Manning's 'M', where:

$$n = \frac{1}{M}$$

Eddy Viscosity

Most commercially available 2D models also include an "eddy viscosity" type term to allow for the effects of sub-grid scale mixing processes. This can be important when modelling flow separations and eddies, or in situations where it is necessary to model channel/overbank interactions.

Introducing a typical eddy viscosity formulation, the *x* momentum equation becomes:

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + g\frac{\partial \varsigma}{\partial x} = -\frac{gu\sqrt{u^2+v^2}}{C^2 d} + E\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right)$$

where:

E: is an "eddy viscosity" coefficient (m^2s^{-1})

If, for illustration purposes only, the hydrostatic pressure and friction terms are neglected, the *x* momentum equation can be rearranged to the form:

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} - E\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) = 0$$

This is analogous to a two-dimensional advection-diffusion equation describing the transport and diffusion of *u*, the *x* velocity component. Continuing the analogy, the eddy viscosity coefficient *E* becomes equivalent to the diffusion coefficient used in advection-diffusion modelling. Thus, as well as having wave propagation and transport properties, discussed in Section 3.3, the momentum equation can also have diffusion properties.

Eddy viscosity and its application to 2D flood models is discussed in more detail in Chapter 10. It is noted, however, that for eddy viscosity calculations to be meaningful, the $u\frac{\partial u}{\partial x}$ and $v\frac{\partial u}{\partial y}$ convective momentum terms must be modelled with sufficient accuracy. This is discussed in more detail in Section 3.9.4.

Other Terms

The early 2D flood models were originally derived from 2D coastal and estuarine models. These models typically included additional terms to represent wind shear and Coriolis effects. When these terms are included, along with additional source/sink terms to allow for the addition or loss of momentum associated with any sources or sinks of mass, discussed above, the *x* momentum equation becomes:

$$\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+g\frac{\partial \varsigma}{\partial x}=-\frac{gu\sqrt{u^2+v^2}}{C^2 d}+E\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right)+fVV_x-\Omega u+Source/Sink$$

where:

f: is a wind shear stress coefficient

V: is the wind speed (m/s)

V_x: is the component of the wind speed in the *x* direction (m/s)

Ω: is a latitude dependent Coriolis parameter

The wind and Coriolis terms are only likely to become important in wide open floodplains or in lake or estuarine systems, and are not considered further in the present discussion.

3.6. Final Forms of the Equations

As developed above, the final forms of the mass and momentum equations used in many 2D flood models can be expressed as:

Mass

$$\frac{\partial \varsigma}{\partial t}+\frac{\partial (d.u)}{\partial x}+\frac{\partial (d.v)}{\partial \mathrm{y}}=Sources-Sinks$$

x-Momentum

$$\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+g\frac{\partial \varsigma}{\partial x}=-\frac{gu\sqrt{u^2+v^2}}{C^2 d}+E\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right)+Source/Sink$$

y-Momentum

$$\frac{\partial v}{\partial t}+v\frac{\partial v}{\partial y}+u\frac{\partial v}{\partial x}+g\frac{\partial \varsigma}{\partial y}=-\frac{gv\sqrt{u^2+v^2}}{C^2 d}+E\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right)+Source/Sink$$

This coupled system of equations provides the three equations necessary to solve for the three dependent variables; ς the free surface elevation, *u* the velocity in the *x* direction and *v* the velocity in the *y* direction.

3.7. Modelling Requirements and Simplifications

The previous sections show that the combination of the mass and momentum equations can describe the wave propagation properties associated with a flood, and how the momentum equations include terms for describing the effects that advection and dispersion of momentum can have on the flow. The relative importance of these properties can vary significantly depending on the flow conditions. This has little impact on how the mass equation is treated, but in some cases can allow simplifying assumptions to be made in the treatment of the momentum equations.

3.7.1. The Mass Equation

For flood modelling applications, it is important that the solution procedure used in the model does not generate or destroy mass numerically. It is therefore essential that all the terms in the mass equation are described accurately in the numerical solution procedure.

With the staggered grids used by most finite difference models, there can be issues with achieving time and space centring of the non-linear spatial derivative terms. In this respect, it is noted that Stelling *et al* (1998) presented a numerical scheme that conserves mass and maintains non-negative water levels. Nevertheless, modellers should be aware that any errors in the mass equation, however small, can accumulate with time as the computation progresses. If the mass equation is not modelled correctly, the error accumulation can continue to the extent that the final solution may be compromised.

3.7.2. The Momentum Equation

For the momentum equations, the relative importance of the different terms can vary quite significantly depending on the flow conditions. In some conditions it may be possible for simplifying assumptions to be made either to the equations themselves, or to the way in which individual terms are treated numerically. The types of simplifications used tend to be made for numerical expediency, or to avoid numerical problems with particular types of flow (e.g., super-critical flow). The extent to which they can be used is dependent upon the level of detail and/or accuracy required.

Ignoring wind, Coriolis and source/sink terms, the x-momentum equation developed above can be expressed as:

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + g\frac{\partial \varsigma}{\partial x} = -\frac{gu\sqrt{u^2+v^2}}{C^2 d} + E\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right)$$

Using this as a base, some of the more commonly used approximations to the momentum equation are discussed below.

The Linearised Momentum Equation

With this approximation, the convective momentum (momentum transport) terms are neglected. When these terms are neglected, the eddy viscosity (momentum dispersion) terms have little physical meaning and can also be neglected. With this approach, the *x*-momentum equation reduces to:

$$\frac{\partial u}{\partial t} + g\frac{\partial \varsigma}{\partial x} = -\frac{gu\sqrt{u^2+v^2}}{C^2 d}$$

This approach should only be used in areas where the velocities are small, and the wave propagation properties of the flow are dominant. This rarely happens in most practical flood flow simulations. However, it is noted that the linearised momentum equation is sometimes used for numerical expediency in order to maintain stability in high velocity flow areas, including regions of super-critical flow. Although this approximation maintains the wave propagation properties of the full momentum equation, it cannot model momentum dominated effects, including flow separations and eddies, and main channel/overbank momentum transfers.

The Steady State Momentum Equation

With this approximation, the local acceleration term *δu/δt* is neglected and the *x*-momentum equation reduces to:

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+g\frac{\partial \varsigma}{\partial x}=-\frac{gu\sqrt{u^2+v^2}}{C^2 d}+E\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right)$$

This approximation neglects the wave propagation properties of the momentum equation. It can be used in reaches with moderate to steep slopes, where the flow is dominated by friction. However, it should not be used for rapidly varying flows, such as in dam-breaks, or in reaches with flat slopes and/or deep water where the local acceleration term (and wave propagation properties of the equation) becomes more important.

The Diffusive Wave Approximation

With this approximation, the convective momentum and eddy viscosity terms are also neglected and the *x*-momentum equation reduces to:

$$\frac{\partial \varsigma}{\partial x}=-\frac{u\sqrt{u^2+v^2}}{C^2 d}$$

That is, the water surface slope is balanced by the friction slope.

As for the steady state momentum equation, this approximation can be used to describe gradually varying flows in reaches with moderate to steep slopes. It includes backwater effects, but has the added limitation that it cannot be used to simulate flow separations and eddies, or main channel/overbank momentum transfers.

The Kinematic Wave Approximation

With this approximation, the surface slope of the water is assumed to be the same as the bed slope the *x*-momentum equation further reduces to:

$$\frac{\partial z}{\partial x}=-\frac{u\sqrt{u^2+v^2}}{C^2 d}$$

That is, the friction slope is equal to the bed slope.

This approximation is effectively the same as solving for the flow properties using a steady state friction law (such as Manning's equation). Backwater effects are not included, and water can only flow downstream. As such, the kinematic wave approximation can only be used to describe gradually varying flows in reaches with moderate to steep slopes where backwater effects can be neglected.

3.7.3. Model Applications

Most of the 2D flood models in common use are based on the numerical solution of the full final forms of the mass and momentum equations, as described above. This works well in straight-forward flow situations. However, most of the commonly used solution procedures can have stability problems when modelling the transport properties (convective momentum terms) of the momentum equations in high velocity flows, and can become ill-conditioned when modelling the wave propagation properties of the equations in super-critical flows. This is generally approached by either deliberately introducing numerical stabilising terms (e.g., Stelling *et al,* 1998, McCowan *et al*, 2000), or by using simplified forms of the momentum equation to describe the flow.

With respect to the latter, it is noted that simplified forms of the momentum equation can be used to describe special flow conditions. For example, BMT WBM (2008) uses the kinematic wave approximation to describe flow conditions in regions with super-critical flow. This approach is reasonable for most flooding applications as super-critical flows are upstream controlled and are normally friction dominated. However, whenever simplified forms of the equations are used, it is important for the modeller to understand their limitations, and care may be required in interpreting the results, particularly in transition areas.

The full set of the equations should always be used for describing flows in relatively flat reaches, and regions of relatively flat, deep water such as in estuaries and lakes.

3.8. *Alternative Forms of the Equations*

The equations discussed above have been described in terms of ς, the free surface elevation, and *u* and *v*, the depth-averaged velocities in the *x* and *y* directions. An alternative approach is to represent the velocity variables in terms of the depth-averaged volume fluxes, *p* and *q*, where:

$p = u.d$, the depth-averaged discharge per unit width in the x direction (m^2s^{-1}), and

$q = v.d$, the depth-averaged discharge per unit width in the y direction (m^2s^{-1})

Mass

In terms of ζ, *p* and *q*, the equation describing the depth-averaged conservation of mass can be expressed as:

$$\frac{\partial \varsigma}{\partial t} + \frac{\partial p}{\partial x} + \frac{\partial q}{\partial y} = 0$$

X-Momentum

The corresponding equation for describing the conservation of momentum in the *x* direction can be expressed as:

$$\frac{\partial p}{\partial \mathrm{t}}+\frac{\partial}{\partial \mathrm{x}}\left(\frac{\mathrm{p}^2}{d}\right)+\frac{\partial}{\partial \mathrm{y}}\left(\frac{\mathrm{pq}}{\mathrm{d}}\right)+\mathrm{gd}\frac{\partial \varsigma}{\partial \mathrm{x}}=-\frac{gp}{C^2 d}\sqrt{\frac{p^2}{d^2}+\frac{q^2}{d^2}}+dE\left(\frac{\partial^2}{\partial x^2}\frac{p}{d}+\frac{\partial^2}{\partial y^2}\frac{q}{d}\right)$$

These equations are an expanded version of the "conservation law" form of the long wave equations as described, for example, by Abbott (1979).

Advantages

The main advantage of this form of the equations is that there is only a single variable in each of the derivatives in the mass equation. That is, the mass equation is linear. This has the significant advantage that, barring coding errors and errors in source/sink terms, the numerical solution of this equation will remain mass conservative.

Disadvantages

The disadvantages of this form of the equations are that the non-linear convective momentum terms become more complex, making numerical discretisation more difficult, and that the hydrostatic pressure term now has a non-linear coefficient. However, these disadvantages do not normally cause problems in most flooding applications.

Proponents of this form of the equations consider that the advantage of having mass conservation virtually guaranteed outweighs the potential disadvantages in discretisation of the momentum equation.

Eddy Viscosity Issues

Currently, one commercially available package that uses conservation law based equations provides options for "flux based", or "velocity based" eddy viscosity formulations.

In the case of the flux based formulation, the eddy viscosity term becomes:

$$E\left(\frac{\partial^2 p}{\partial x^2}+\frac{\partial^2 p}{\partial y^2}\right)$$

Although much simpler numerically, this term results in the dispersion of the depth-averaged discharge $p=u.d$, rather than the velocity u. In areas of rapidly changing depths this can lead to an artificial transfer of flow velocity from deep water to shallow water. As a result, the velocity based eddy viscosity formulation is considered to be the more appropriate for all flooding applications.

3.9. Numerical Solution Procedures

3.9.1. Types of Numerical Solutions

Development of numerical representation techniques for the governing equations of fluid flow is an active area of engineering research and entire volumes have been devoted to documenting the development of such techniques (e.g., Abbott, 1979; Stelling, 1984; Abbott and Basco, 1989; Zienkiewicz and Taylor, 2000; LeVeque, 2002).

There are three primary generic approaches:

- Finite Difference
- Finite Element
- Finite Volume

The main features of each of these approaches are outlined below.

Finite Difference: Nodal locations within the solution domain must be defined on a regular fixed grid (in cartesian or curvilinear space). The key distinctive mathematical element of finite difference techniques is that single point values of the key variables are used to estimate the gradient terms in governing equations. Finite difference methods are customarily taught at undergraduate engineering level.

Finite Element: The solution domain is subdivided into an assembly of elemental areas or volumes. Finite element techniques yield solutions that are smooth and continuous over each defined element. Solutions are obtained by integrating particular forms of the governing equations over each element whilst ensuring matching values at the interfacial nodes connecting each element.

Finite Volume: In the finite volume technique, the solution domain is subdivided in a manner similar to finite element techniques. However, each discretised volume is treated as a unique control volume (cell) represented by volume-averaged values of the conserved variables. The finite volume methods are most intuitively thought of as control-volume methods, due to their basis in the conservative-integral form of the shallow water equations. The rate of change of conserved variables is derived by integrating the cell-interface fluxes. A key step in these methods involves calculating the numerical fluxes at the cell interfaces, otherwise known as a Riemann problem. Various implicit and explicit integration methods can be used to advance the solution in time. Being based on the conservative integral form of the shallow water wave equations, finite volume schemes are generally better able to handle shocks (hydraulic jumps and bores) and may therefore potentially perform better in mixed regime flow situations.

There are also hybrid approaches that combine the Finite Element and Finite Volume schemes, known as Discontinuous Galerkin schemes. Also it is interesting to note that generally the finite difference schemes can be derived using the Finite Volume framework on a rectilinear/curvilinear grid.

Finite difference techniques have fixed (generally square) grids and are relatively simple to implement and to use. They have been used extensively in the historical development of 2D flood models, and are still used extensively in practice (e.g., Stelling *et al*, 1998; McCowan *et al*, 2000 and Syme, 2001; DHI, 2005).

With their flexible meshes, finite element and finite volume techniques can make it possible to concentrate the computation onto particular areas of interest. Early development was mostly with finite element techniques (e.g., King and Roig, 1988), however due to potential mass conservation issues, more recent flexible mesh model development has focussed more on finite volume techniques (e.g., Van Drie *et al*, 2008).

3.9.2. The Discretisation Process

In all numerical solution procedures the continuous equations of motion developed above have to be approximated by discrete (discontinuous) arithmetic expressions that can then be solved by a computer. The differences between the main approaches discussed above are in the way in which the approximate arithmetic expressions are formed, and the methods used to solve them.

In the following discussion, simple finite difference approximations have been used to illustrate the way in which the equations of motion can be transformed into discrete arithmetic equations that can then be solved numerically. For this, a simplification of the final forms of the equations presented in Section 3.6 has been used.

Neglecting sources and sinks, and assuming that the depth of flow is uniform (for illustration only), the mass equation can be written as:

Mass $$\frac{\partial \varsigma}{\partial t} + \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

Similarly, neglecting sources and sinks, and neglecting the convective momentum and eddy viscosity terms, the *x* and *y* momentum equations can be written as:

X-Momentum $$\frac{\partial u}{\partial \mathrm{t}} + \mathrm{g}\frac{\partial \varsigma}{\partial \mathrm{x}} = -\frac{gu}{C^2 d}\sqrt{u^2 + v^2}$$

Y-Momentum $$\frac{\partial v}{\partial \mathrm{t}} + \mathrm{g}\frac{\partial \varsigma}{\partial y} = -\frac{gv}{C^2 d}\sqrt{u^2 + v^2}$$

An Example of a Computational Fixed Grid

One of the simplest ways of discretising these equations is to use a square “staggered” fixed grid similar to that shown in Figure 3.2. Here the water levels ς, the water depths *d* and the bed levels *z* can all be specified at the intersections of each x and y grid line, and the *x* velocities *u*, and *y* velocities *v* can be specified at the mid-points along each x and y grid line, as shown. The locations of the different variables in space and time can then be given in terms of discrete numbers of grid sizes $j\Delta x$ and $k\Delta y$, and time steps $n\Delta t$.

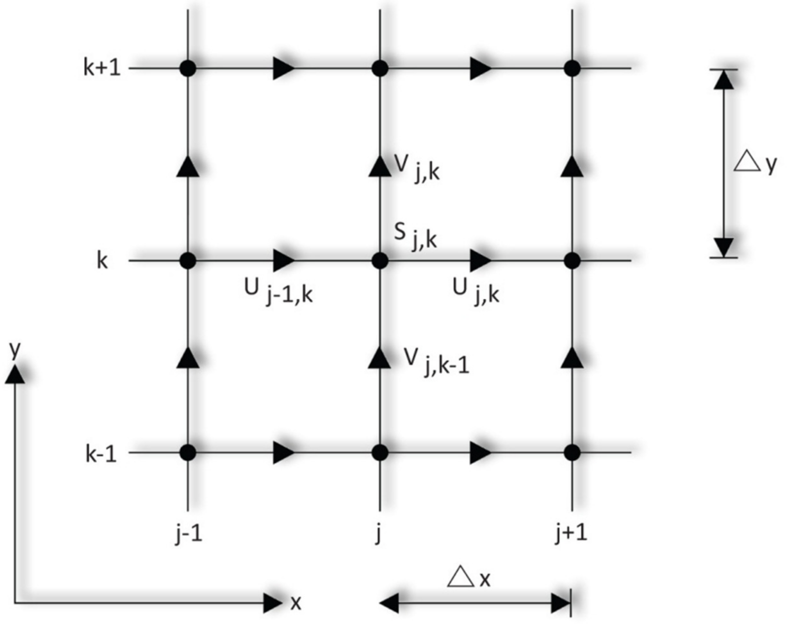


Figure 3-2 Example of a Computational Grid

Discretisation of the Mass Equation

It can be seen that the mass equation contains a derivate of the water surface elevation ς with respect to time *t*. This equation is used to compute the water surface elevation at each successive new timestep.

As a first step, the continuous differential form of the simplified mass equation, above, is approximated by the following discrete difference equation:

$$\frac{\Delta\varsigma}{\Delta t} + \frac{\Delta u}{\Delta x} + \frac{\Delta v}{\Delta y} = 0$$

Finite difference approximations to this equation can be formed in terms of the water surface elevation values $\varsigma_{j\Delta x,\, k\Delta y}$, located at the centre of each *(jΔx, kΔy)* grid cell, and the adjacent *x* and *y* velocity values $u_{(j+1/2)\Delta x, k\Delta y}$, $u_{(j-1/2)\Delta x, k\Delta y}$ and $v_{j\Delta x,(k+1/2)\Delta y}$, $v_{j\Delta x,(k-1/2)\Delta y}$. Using the superscript *n* to represent the location of the variables in time in terms of time steps *nΔt*, the discrete difference form of the mass equation can, for illustration purposes, be represented by the following finite difference equation:

$$\frac{(\varsigma^{(n+1)\Delta t} - \varsigma^{n\Delta t})_{j\Delta x,k\Delta y}}{\Delta t} + \frac{(u_{(j+1/2)\Delta x} - u_{(j-1/2)\Delta x})^{n\Delta t}_{k\Delta y}}{\Delta x} + \frac{(v_{(k+1/2)\Delta y} - v_{(k-1/2)\Delta y})^{n\Delta t}_{j\Delta x}}{\Delta y} = 0$$

If the water surface elevations ς and adjacent *x* and *y* velocity values *u* and *v* are known at timestep *nΔt*, then the above difference equation can be rearranged to compute the water surface elevation ς at the new timestep *(n+1)Δt*, as follows:

$$\varsigma^{(n+1)\Delta t}_{j\Delta x,k\Delta y} = \varsigma^{n\Delta t}_{j\Delta x,k\Delta y} - \frac{\Delta t}{1}\left[\frac{(u_{(j+1/2)\Delta x} - u_{(j-1/2)\Delta x})^{n\Delta t}_{k\Delta y}}{\Delta x} + \frac{(v_{(k+1/2)\Delta y} - v_{(k-1/2)\Delta y})^{n\Delta t}_{j\Delta x}}{\Delta y}\right]$$

Discretisation of the Momentum Equations

Similarly, it can be seen that the *x* (and *y*) momentum equations contain time derivatives of the *x* (and *y*) velocities *u* (and *v*). These equations are used to compute the *u* (and *v*) velocities at successive time steps.

Using only the *x* momentum equation as an example, the continuous differential form of the simplified *x* momentum equation can be approximated by the following discrete difference equation:

$$\frac{\Delta u}{\Delta t} + g\,\frac{\Delta \varsigma}{\Delta x} = F_x$$

where: F_x is the component of the friction term in the *x* direction

Using the same approach as for the mass equation, the discrete form of the *x* momentum equation can, for illustration purposes, be approximated by the following finite difference equation centred on *[(j+1/2)Δx,kΔy]*, the grid location of the *u* velocity:

$$\frac{(u^{(n+1)\Delta t} - u^{n\Delta t})_{(j+1/2)\Delta x,k\Delta y}}{\Delta t} + g\,\frac{(\varsigma_{(j+1)\Delta x} - \varsigma_{j\Delta x})^{n\Delta t}_{k\Delta y}}{\Delta x} = -F^{n\Delta t}_{x(j+1/2)\Delta x,k\Delta y}$$

If the water surface elevations ς and adjacent *x* and *y* velocity values *u* and *v* are known at timestep *nΔt*, then the above difference equation can be rearranged to compute the *x* velocity *u* at the new timestep *(n+1)Δt*, as follows:

$$u^{(n+1)\Delta t}_{(j+1/2)\Delta x,k\Delta y} = u^{n\Delta t}_{(j+1/2)\Delta x,k\Delta y} - \frac{\Delta t}{1}\left[\frac{(\varsigma_{(j+1)\Delta x} - \varsigma_{j\Delta x})^{n\Delta t}_{k\Delta y}}{\Delta x} + F^{n\Delta t}_{x(j+1/2)\Delta x,k\Delta y}\right]$$

A similar expression representing the *y* momentum equation can be formed to compute the *y* velocity *v* at the new timestep *(n+1)Δt*, as follows:

$$v^{(n+1)\Delta t}_{j\Delta x,(k+1/2)\Delta y} = v^{n\Delta t}_{j\Delta x,(k+1/2)\Delta y} - \frac{\Delta t}{1}\left[\frac{(\varsigma_{(k+1)\Delta y} - \varsigma_{k\Delta y})^{n\Delta t}_{j\Delta x}}{\Delta y} + F^{n\Delta t}_{y.j\Delta x,(k+1/2)\Delta y}\right]$$

Although the finite difference expressions developed above would not be used in practice, they do serve to illustrate how the discretisation process can work.

3.9.3. Discretisation Errors

Modellers should be aware that the discretisation process introduces numerical errors, irrespective of the numerical solution procedure being used. These errors are called “truncation” errors. They are different to computer “round-off” errors, and can have significant implications on the accuracy of a model’s results.

To illustrate the effect, the discretisation of the hydrostatic pressure term in the *x* momentum equation developed in the preceding section has been examined more closely. For a given timestep *nΔt*, and *x* grid line located at *kΔy*, this term can be approximated as follows:

$$g\,\frac{\partial \varsigma}{\partial x} \approx g\,\frac{\Delta \varsigma}{\Delta x} = g\,\frac{\varsigma_{(j+1)\Delta x} - \varsigma_{j\Delta x}}{\Delta x}$$

With this approach, the continuous derivative of the water surface elevation with respect to *x* is approximated by the linear gradient of the water surface elevation between adjacent grid points.

To determine the errors introduced by this approximation, Taylor's Series can be used to expand the water surface elevations in right hand side of the above equation in terms of the water surface elevation at the centre-point, $(j+1/2)\Delta x$, as follows:

$$\varsigma_{(j+1)\Delta x} = \varsigma_{(j+1/2)\Delta x} + \frac{(\Delta x/2)}{1!}\frac{\partial \varsigma}{\partial x} + \frac{(\Delta x/2)^2}{2!}\frac{\partial^2 \varsigma}{\partial x^2} + \frac{(\Delta x/2)^3}{3!}\frac{\partial^3 \varsigma}{\partial x^3} + \frac{(\Delta x/2)^4}{4!}\frac{\partial^4 \varsigma}{\partial x^4} + \ldots\ldots\ldots.$$

$$\varsigma_{j\Delta x} = \varsigma_{(j+1/2)\Delta x} - \frac{(\Delta x/2)}{1!}\frac{\partial \varsigma}{\partial x} + \frac{(\Delta x/2)^2}{2!}\frac{\partial^2 \varsigma}{\partial x^2} - \frac{(\Delta x/2)^3}{3!}\frac{\partial^3 \varsigma}{\partial x^3} + \frac{(\Delta x/2)^4}{4!}\frac{\partial^4 \varsigma}{\partial x^4} - \ldots\ldots\ldots.$$

Substituting these expressions into the original difference equation, results in:

$$g\,\frac{\varsigma_{(j+1)\Delta x} - \varsigma_{j\Delta x}}{\Delta x} = g\,\frac{\partial \varsigma}{\partial x} + \frac{\Delta x^2}{24}\frac{\partial^3 \varsigma}{\partial x^3} + \frac{\Delta x^4}{1920}\frac{\partial^5 \varsigma}{\partial x^5} + \ldots\ldots\ldots.$$

That is, the discrete finite difference approximation is equal to the original continuous partial differential term, plus some additional higher order "truncation error" terms.

Similar truncation error terms can be developed for the finite difference approximations to the other terms in the mass and momentum equations. The properties of these truncation errors can have a significant effect on the accuracy and even the stability of the numerical procedures being used.

For the time and space centred ("second-order") schemes used in most flood models, the lowest order error terms are third-order derivatives with coefficients involving factors of Δt^2 and Δx^2. If the timestep Δt and grid size Δx are treated as fractions of representative time and length scales of the flood under consideration, these truncation errors can be shown to reduce quadratically with decreasing grid size and timestep. That is, the higher the fixed grid resolution and timestep, the smaller the numerical truncation errors.

Second-order finite difference schemes are well suited to modelling the wave propagation properties of a flood flow. However, they are not as well suited to modelling situations where the transport properties of the flow (i.e., the convective momentum terms) become important. Under these conditions, the use of second-order difference schemes can result in artificial "zig-zagging" of the flow and instability (Abbott and Rasmussen, 1977). As a result, Leonard (1979) advocates the use of higher third-order schemes for modelling transport dominated flows.

3.9.4. Numerical Eddy Viscosity

In some models, low first-order difference schemes are deliberately used to discretise the convective momentum terms. This is done to help stabilise the overall numerical computation (e.g., Stelling *et al*, 1998), or to maintain local stability in regions of high velocity and super-critical flow (e.g., McCowan *et al*, 2000). To illustrate how this works, the $u\delta u/\delta x$ convective momentum term can be discretised in a manner similar to the pressure term above, as follows:

$$u\frac{\partial u}{\partial x} \approx u\frac{\Delta u}{\Delta x} = u\frac{u_{j\Delta x} - u_{(j-1)\Delta x}}{\Delta x}$$

The main difference with the current approach is that the continuous derivative of the velocity *u* with respect to *x* is approximated by the linear gradient of the velocity between the grid point at which the derivative is to be taken, and the grid point immediately upstream. As a result, this approach is often termed "upwind" differencing.

As before, Taylor's Series can be used to expand the *u* velocities in the right hand side of the above equation in terms of the *u* velocity at the centre-point, $j\Delta x$, as follows:

$$u_{j\Delta x} = u_{j\Delta x}$$

$$u_{(j-1)\Delta x} = u_{j\Delta x} - \frac{\Delta x}{1!}\frac{\partial u}{\partial x} + \frac{\Delta x^2}{2!}\frac{\partial^2 u}{\partial x^2} - \frac{\Delta x^3}{3!}\frac{\partial^3 u}{\partial x^3} + \frac{\Delta x^4}{4!}\frac{\partial^4 u}{\partial x^4} - \ldots\ldots\ldots\ldots$$

Substituting these expressions into the original difference equation, results in:

$$u\frac{u_{j\Delta x} - u_{(j-1)\Delta x}}{\Delta x} = u\frac{\partial u}{\partial x} - u\frac{\Delta x}{2}\frac{\partial^2 u}{\partial x^2} + u\frac{\Delta x^2}{6}\frac{\partial^3 u}{\partial x^3} - u\frac{\Delta x^3}{24}\frac{\partial^4 u}{\partial x^4} + \ldots\ldots\ldots\ldots$$

This shows that, as previously, the discrete finite difference approximation is equal to the original continuous partial differential term. This time, however, the truncation error now includes a second derivative term which is of the same form as one of the eddy viscosity terms in the *x*-momentum equation. That is, upwinding of the convective momentum terms can be seen to be equivalent to introducing a numerical eddy viscosity term with, in this case, an eddy viscosity coefficient of:

$$E_N = u\frac{\Delta x}{2}$$

For a typical flow velocity of say u=1 m/s, and grid size of Δx=5m, the corresponding numerical eddy viscosity coefficient of E_N = 2.5 m^2/s is likely to be an order of magnitude (or more) greater than the physically realistic value appropriate to those flow conditions. This numerical eddy viscosity is grid size and flow direction dependent. It smooths out irregularities in the flow and helps make the model calculations stable. It also suppresses flow separations and eddy formation and makes it impractical to compute channel/overbank interactions, where the specification of appropriate eddy viscosity coefficients becomes important (refer to Chapter 10). Problems associated with upwind differencing are discussed in detail by Leonard (1979).

In models where upwind differencing of the convective momentum terms is used throughout the entire model domain, numerical eddy viscosity will dominate the solution. In these models, there is little point in including eddy viscosity explicitly in the model equations (e.g., Stelling et al, 1998). The local use of upwind differencing to model super-critical flow (e.g., McCowan et al, 2000) is a separate issue, and is discussed in more detail in Chapter 10.

3.10. **Solution Procedures**

Solution procedures can be either "explicit" or "implicit", irrespective of whether finite difference, finite element or finite volume techniques are being used. Additionally, most of the more commonly used finite difference models use what is known as an "alternating direction implicit" or ADI algorithm. The modeller should aware of the type of solution procedure being used, and of any constraints that it may impose on the on the way in which the computation is carried out.

The "Courant" number C_r is one of the key parameters used to define the differences between explicit and implicit solution procedures. It expresses the number of fixed grid or flexible mesh elements that flow information can travel in one timestep. The Courant number C_r can be defined as:

$$C_r = \left[\frac{(u + \sqrt{gd}).\Delta t}{\Delta x} \right]$$

3.10.1. Explicit Solution Procedures

With an explicit solution procedure, the solutions to the water surface elevations and flow velocities at the new timestep are computed directly (explicitly) as a function of the known values at the old timestep. This is achieved in much the same way as the finite difference example presented in Section 3.9.2.

Explicit schemes tend to be computationally efficient, but have a stability constraint that information can only travel a maximum of one grid/mesh element in a single timestep. That is, that the Courant number must always be less than one (i.e., $Cr \leq 1$). This provides a stability constraint on the timestep Δt, that:

$$\Delta t_{\max} \leq \left[\frac{\Delta x}{(u + \sqrt{gd}} \right]_{\min}$$

This is commonly called the "Courant" stability criterion.

3.10.2. Implicit Solution Procedures

With an implicit solution procedure, the water surface elevations and flow velocities at the new timestep are expressed as a combination of both the known values at the old timestep and adjacent unknown values at the new timestep. As a result, the solutions at one grid/mesh element are linked to those in the neighbouring cells. These solutions are, in turn, linked to those in their neighbouring cells, and so on. In this way, the solutions to the discrete numerical approximations to the mass and *x* and *y* momentum equations for each grid/mesh element are linked "implicitly" to those in every other cell over the entire model domain.

The implicit approach provides much greater flexibility in the way in which the discrete numerical approximations can be formed. It also allows flow information to travel much further than one grid point per timestep. As a result, the Courant stability criterion does not apply to implicit solution procedures. Model time steps are determined more by accuracy requirements rather than stability constraints.

The downside is that, for each timestep, the numerical approximations to the mass and *x* and *y* momentum equations have all to be solved simultaneously at every grid/mesh element throughout the model domain. This makes implicit schemes very "heavy" computationally, and fully implicit schemes are not used extensively in practice.

3.10.3. ADI Solution Procedures

The alternating direction implicit (ADI) solution procedure was originally developed by Leendertse (1967) to provide some of the advantages of the implicit approach in a simpler easier to implement framework. It can only be used with a regular grid structure and, as a result, is generally restricted to finite difference procedures.

With the original ADI approach, the computation of the water surface elevations ς and the *x* and *y* velocities *u* and *v* at each new timestep is broken down into two stages. In the first stage, the mass equations and *x* momentum equations are solved in a series of implicit one-dimensional "sweeps" along each *x* grid line. This provides the water surface elevation ς and *x* velocity *u* values at the new timestep. In the second stage, the *y* momentum equations are solved in a series of explicit one-dimensional sweeps along each *y* grid line. This provides the corresponding *y* velocity *v* values at the new timestep. For the next timestep, the sequence is reversed to balance the scheme in time. That is, the mass equations and *y* momentum equations are solved in the series of implicit one-dimensional "sweeps", this time along each *y* grid line, to provide the ς and *v* values at the new timestep. The *x* momentum equations are then solved in the series of explicit one-dimensional sweeps, this time along each *x* grid line, to provide the corresponding *u* values.

The overall approach can be shown to be independent of the Courant stability criterion and the sequence of alternating one-dimensional *x* and *y* sweeps is much simpler to implement than a fully two-dimensional implicit solution procedure. As a result ADI solution procedures have been used extensively in two-dimensional flood modelling (e.g., Stelling *et al*, 1998; McCowan *et al*, 2000 and Syme, 2001).

It should be noted, however, that the ADI approach is not directly equivalent to a fully two-dimensional implicit scheme. Although the timestep is not subject to the Courant stability criterion, it is subject to accuracy constraints, particularly when the solution involves flow in relatively narrow channels at an angle to the grid. This is because numerical flow information can only travel alternatively along *x* and *y* grid lines, and not directly along a channel that is not aligned with the model grid. The differences are discussed in detail by Benque *et al* (1982). In practice, the timestep should be selected such that the Courant number is significantly less than the minimum number of grid cells used to describe the width of the channel.

3.11. **Numerical Solution Requirements**

The equations of motion and numerical solution procedure are generally a function of the generic two dimensional modelling system or package to be used. With these defined, the development of a site-specific model requires the development of a computational fixed grid, or flexible mesh covering the study area, the allocation of topographic data to individual grid/mesh elements, and the specification of other inputs such as roughness coefficients, eddy viscosity coefficients, etc. The actual mechanism of developing and calibrating a two-dimensional model is described in later chapters of this document. There are, however, two additional sets of information that are necessary from a numerical point of view to enable the solution procedure

to move forward in time.
These are:

- Initial conditions
- Boundary Conditions

These requirements are discussed briefly below.

3.11.1. Initial Conditions

For any model computation to move forward in time there must be a known starting point. For numerical hydrodynamic models this starting point is known as the model "initial conditions". The initial conditions necessary for two-dimensional hydrodynamic models consist of water surface elevation ς and u and v velocity values at every grid/mesh element that is "active" at the start of the computation.

These values can be specified by:

- A "cold start", where initial estimates of the water surface elevation ς values are made, and the u and v velocity values are set to zero (i.e., no flow).
- A "hot start", where the initial water surface elevation ς, and u and v velocity values are specified from the results of a previous model simulation.

3.11.1.1.Cold Starts

In many flood modelling applications, the precise values of the initial conditions are not that critical. This is because the flow conditions at the flood peak will generally be determined by the inflow of water associated with the inflow hydrographs applied at the model boundaries. Provided there is sufficient "warm-up" time at the start of the inflow hydrographs, any errors caused by a poor estimate of the initial conditions will have been washed out of the downstream boundary by the time the flood peak arrives. In these cases, the main role of the initial conditions is to ensure that the model computation starts in a reasonably realistic manner; that is, relatively smoothly and with no initial instabilities

In flood modelling applications where there are lakes, wetlands, retarding basins, or other depressions that may provide initial flood storage, it is important that the initial water surface elevations provide the correct amount of initial storage in these areas. If the initial amount of water in these storage areas is underestimated, this may cause the model to artificially attenuate the flood peak. Conversely, if it is overestimated, the flood peak may be artificially enhanced.

3.11.1.2.Hot Starts

Hot starts are not used extensively in practice. Their main uses tend to be limited to:

- Providing initial conditions for model simulations where significant computing time may be required for the model "warm-up" period required. In these cases, the results of a single prolonged "warm-up" simulation can be used to provide initial conditions for a subsequent series of model simulations.

- Breaking-up very long model simulations into more manageable sub-sections.

3.11.2. Urban Applications

In many urban applications, the area to be modelled may be initially dry. In these cases, the initial water surface elevation ς values will typically be set to the corresponding ground surface elevation z values, and the initial u and v velocity values set to zero. This approach could be considered as a special case of the cold start. It is applicable in model simulations where there is no initial overland flow, including direct rainfall on grid applications.

For models with initially dry overland flowpaths, small areas of active water cells may be used to initiate the computation along open inflow and outflow boundaries. At each boundary location, the initial water surface elevation ς values are generally set to a constant value (i.e., the water level is assumed to be horizontal), and the initial u and v velocity values are set to zero.

3.12. **Boundary Conditions**

With the initial conditions specified, the model boundary conditions are the remaining pieces of necessary information required for the computation to proceed. In this respect it is noted that boundary conditions are required at every grid/mesh element along the model boundaries. These boundaries include both external boundaries and internal boundaries, where:

- External boundaries are located along the external edges or boundaries of the model, where water can flow into or out of the model domain.
- Internal boundaries are located within the model domain at the interface between wet (water) grid/mesh cells, where the computation is to be carried out, and dry (land) cells where there is no computation.

The subject of boundary conditions for two-dimensional flow models is quite complex and the following discussion is, of necessity, relatively superficial.

3.12.1. External Boundaries

The model requires boundary conditions in terms of either water levels or velocities along both the upstream and downstream boundaries.

The upstream boundary conditions are generally provided by a discharge hydrograph. This has to be converted to velocities at each individual grid/mesh element along the boundary. For this to be done, some assumptions need to be made with respect to the distribution of the velocities along the boundary, and to the direction of the flow. As a first approximation it can be assumed that that the flow velocity is uniform across the boundary, and that the flow is perpendicular to the boundary. The appropriate velocity can then be computed as function of the cross-sectional area at the boundary and the discharge. This approach can work well when there is little variation in water depth along the boundary. In cases where there are significant variations in water depth along the boundary, Manning's friction law can be used to provide an improved estimate of the distribution of velocities along the boundary. Depending on the modelling package being used, the flow distribution along the boundary may be computed internally with options for providing user specified flow distributions and directions.

The downstream boundary conditions are generally specified in terms of water surface elevations. These may be specified as a constant, a times series, or computed internally using a rating curve. As a first approximation, it can be assumed that the water surface elevations along the boundary are horizontal. As such, the water surface elevation specified at each individual grid point or mesh element will be the same. Depending on the modelling package being used, the flow directions may be specified as being normal to the boundary, or there may be options for them to be computed as a function of the upstream flow conditions, or specified externally by the user.

Whatever forms of boundary conditions that are being used, it is important to recognise that the model has no information regarding the flow conditions upstream or downstream of the model boundaries. As such, it is important that, wherever possible, the model boundaries should be located in areas where the flow is expected to be relatively uniform. The model boundaries should also be placed far enough upstream and downstream of the area of interest to ensure that any errors in flow distribution and/or direction do not have a significant effect on the model results.

3.12.2. Internal Boundaries

Internal boundaries can be considered as a special case of a velocity boundary. That is, as velocity boundaries where the flow velocity between adjacent pairs of wet and dry cells is set to zero. The locations of the internal boundaries are dynamic as grid/mesh cells are brought into the computation as the flood rises, and taken out again as the flood recedes. This wetting and drying process is discussed in more detail in Chapter 10.

3.13. **References**

Abbott, M.B. (1979), Computational Hydraulics – `s of the Theory of Free Surface Flows, Pitman, London.

Abbott, M.B. and Basco, D.R. (1989), Computatinal Fluid Dynamics – An Introducton for Engineers, Longman Scientific & Technical, Singapore

Abbott, M.B. and Rasmussen C.H. (1977), On the Numerical Modelling of Rapid Contraction and Expansion in Models that are Two-dimensional in Plan, Proc. XV11 Congress, IAHR, Baden-Baden.

Benque, J.P., Cunge, J.A., Feuillet, J., Hauguel, A. and Holly, F.M. (1982), New Method for Tidal Computation, ASCE, J. Waterway, Port, Coastal and Ocean Division, Vol. 108, No. 3.

Bishop, W.A., McCowan, A.D., Sutherland, R.J. and Watkinson, R.J. (1995), Application of Two-Dimensional Numerical Models to Urban Flood Studies, 2nd International Symposium on Stormwater Management, Melbourne.

Cunge, J.A., Holly, F.M. and Verwey, A. (1980), Practical Aspects of Computational River Hydraulics, Pitman, London (Reprinted by University of Iowa).

DHI Water and Environment (2005), MIKE Flood – Integrated 1D - 2D Flood Modelling, Reference Manual, DHI Software, Hørsholm

King, I.P. and Roig, L.C. (1988), Two-Dimensional Finite Element Models for Flood Plains and Tidal Flats, Int'l Conf. on Computational Methods in Flow Analysis, Okayama.

Leendertse, J.J. (1967), Aspects of a Computational Model for Long-Period Water-Wave Propagation, RM-5294-PR, Rand Corporation, Santa Monica.

Leonard, B.P. (1979a), A Survey of Finite Differences of Opinion on Numerical Muddling of the Incomprehensible Defective Confusion Equation, In: T.J.R. Hughes (ed.) Finite Element Methods for Convection Dominated Flows, AMSE, New York.

Leonard, B.P. (1979b), A Stable and Accurate Convective Modelling Procedure Based on Quadratic Upstream Interpolation, Computer Methods in Applied Mech and Eng., North-Holland.

LeVeque, R.J (2002), Finite Volume Methods for Hyperbolic Problems, Cambridge University Press, Cambridge.

McCowan, A D, Rasmussen, E B and Berg, P. (2000), Improving the Performance of a Two-dimensional Hydraulic Model for Floodplain Applications, Proceedings of the 6th Conference on Hydraulics in Civil Engineering, Hobart, 2001

Stelling, G.S. (1984), On the Construction of Computational Methods for Shallow Water Flow Problems, Rijkswaterstaat Communications, No. 35/1984, The Hague.

Stelling, G.S., Kernkamp, H.W.J and Laguzzi, M.M. (1998), Delft Flooding System: A Powerful Tool for Inundation Assessment based upon a Positive Flow Simulation, in V. Babovic and L.C. Larsen (eds) Hydroinformatics '98, A.A. Balkema, Brookfield.

Syme W.J. (2001), TUFLOW – Two & One-dimensional Unsteady FLOW Software for Rivers, Estuaries and Coastal Waters, IEAust Water Panel Seminar and Workshop on 2D Flood Modelling, Sydney.

Van Drie, R., Simon, M. and Schymitzek, I. (2008), 2D Hydraulic Modelling over a Wide Range of Applications, 9th National Conf. on Hydraulic in Water Engineering, Darwin.

4. CHAPTER 4 - MODELLING PROCESS

4.1. Introduction

The guidance in this chapter is designed to identify critical considerations during the planning stage of the numerical modelling process, and to outline guiding principles that can assist the modeller in the selection of appropriate modelling tool, and the preparation of an effective modelling strategy for the problem at hand. With the enormous variety of hydraulic engineering problems likely to be encountered and modelled by water engineering professionals, and with the rapid development of new and existing modelling packages, there is no single cookbook-style approach to the modelling process that can be adopted for every problem. However, there are several key principles that should govern the building of any new model, and broad steps that are common to the process, namely:

- identification, clarification and conceptualisation of the problem;
- assessment of whether the use of a numeric modelling package is justified or required;
- data acquisition, compilation and review;
- model schematisation;
- calibration and verification; and
- processing and interpretation of results.

While the process of developing, applying and analysing a model is described here as a linear process progressing from one step to the next, there will always be overlap and reiteration of the steps as the model progresses from its initial core form to being fully calibrated and verified. Modellers may need to keep various decisions under review as they work through the process. The steps of problem definition, model selection, and data acquisition tend to overlap and may iteratively influence each other.

In addition, there are heuristic aspects to the modelling approach. That is, to some extent, modelling expertise is built on the gaining of knowledge through experimentation and trial and error. The guidance provided in this chapter then is no replacement for such iterative learning over time of modelling rules of thumb, coupled with a thorough understanding of the underlying principles and governing equations. However, the chapter does attempt to provide an overarching structure of the modelling process in order to better to help the modeller navigate the model development process and its potential pitfalls. This structure is best summarised as shown in Figure 4-1.

The major steps are outlined in this Chapter, with subsequent Chapters providing greater detail.

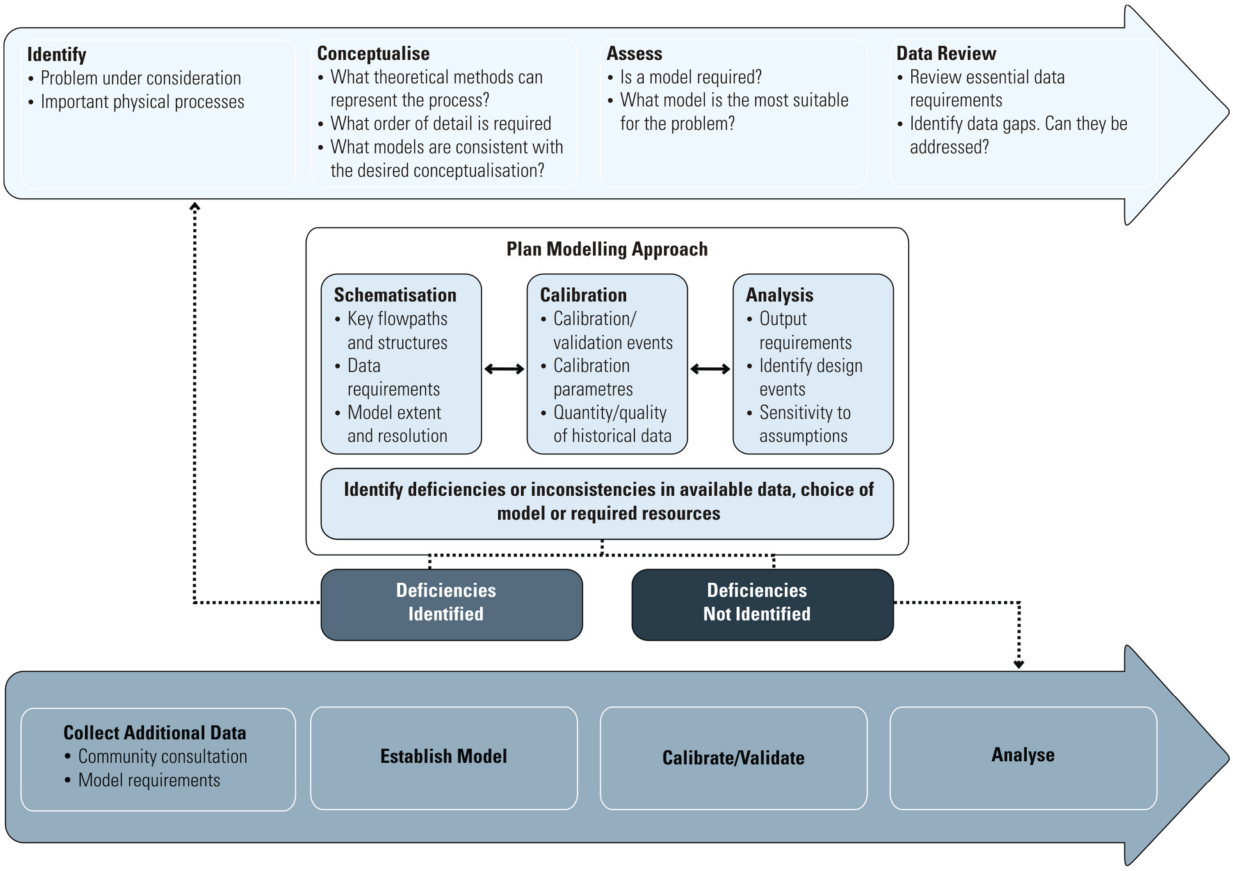


Figure 4-1 Flow Chart of Modelling Process

4.2. Problem Identification & Conceptualisation

The first step in any investigation should be a clear brief or definition of the problem to be addressed. Modellers require a clear understanding of why they are applying a model. It is essential to know the scope and purpose of the analysis in order to rationally decide how to proceed to achieve the desired objectives.

Objectives specifically relating to 2D hydraulic modelling may include:

- analysis or design of drainage systems from simple systems such as a single channel or culvert to complex systems with multiple pit/pipe connections, overland flowpaths, drainage channels and detention basins;
- estimation of peak flood levels, depths, discharges, velocities, etc;
- zoning the study area into broad categories for floodplain management/planning;
- detailed flood behaviour at the scale of individual structures;
- assessment of floodplain management mitigation measures;
- development or analysis of operating rules for water resource management, e.g. for irrigation allocation or wetland hydrology;
- dam break risk analysis;
- impacts of broad-scale development policy or a localised development on flow behaviour; and/or
- environmental impacts on receiving waters such as wetlands and marshes resulting from

catchment urbanisation or flow diversion.

Urban areas typically contain numerous flowpaths. Roads are generally designed to convey flow to pit and pipe networks. These networks then carry flows to trunk drains or natural watercourses. When flows exceed the capacity of these formal drainage paths, overflow between or through buildings can occur. The modeller should consider how critical each path is in terms of the proportion of flow it conveys, and at what level of detail is required to model each path. If a detailed estimate of flow around individual buildings is required, the modeller will need to consider any controls that may affect this flow behaviour, such as fences. However, if details of flows around buildings are not required and these flowpaths are not critical (i.e. do not affect flow behaviour elsewhere), it may be possible to consider simpler overland flowpaths such as roadways only.

Rural areas often present similar challenges of scale. The floodplains of major river systems may be several kilometres wide, requiring a large study area to model the broad scale flow behaviour, yet the focal area for the study may be relatively small, for example the design of a flood mitigation structure such as a levee or spillway, or the operation of a flow regulating structure. It may sometimes be necessary to model the interactions between a river and a tributary with an order of magnitude difference in scale.

Thus, by both identifying the scale and complexity of flow behaviour and considering the outcomes required, the modeller is able to determine which hydraulic features will need to be included in the model schematisation. Typically, the simpler or coarser the output required, the broader the scale of the processes that must be considered and the simpler the schematisation that can be applied.

It is essential that the modeller personally inspect the study area to identify critical features and gain a preliminary impression of likely flow behaviour. If the area is very large, it may be useful and practical to fly over it. For smaller areas, driving and walking through as much of the study area as possible is recommended. Significant drainage and flow management structures should be photographed and closely inspected. Observed information about floodplain features such as such as channels, levees, basins, dams, bridges, culverts, drainage easements, road/rail embankments, vegetation, buildings and obstructions can often prove invaluable as the study progresses.

4.3. Assess Model Requirements

In selecting an appropriate model, consideration needs to be given to the hydraulic behaviour through the study area, data requirements, schematisation, calibration and results processing and interpretation. Early attention to detail can save costs in model development and run time. The modelling package selected must be suitable for the overall task. Chapter 3 contains important information on the strengths and limitations of various modelling schemes. Consideration of the primary flow mechanisms at work is a vital step in selecting a model. Mechanisms inducing flow in the study area can generally be categorised as follows:

- runoff due to rainfall occurring within the study area;
- runoff from upstream areas of the catchment entering the study area either through a subsurface drainage network or as open channel flow;

- runoff from adjacent catchments diverted through the study area via subsurface drainage network or overland flow;
- interflow and baseflow;
- infiltration and evapotranspiration;
- coastal flooding from tides, storm surges or tsunamis; and/or
- infrastructure failure such as dam break.

4.4. **Data Review**

The quality and quantity of available data will affect every step in the modelling process, including modelling approach, schematisation, calibration and analysis. Initially, consider whether the available data meet certain minimum criteria.

- Are there sufficient data to define the fundamental components of the model, such as topographic/ bathymetric data, hydrologic data to define boundary conditions, and historical data for calibration? If impacts of proposed development are to be assessed, are there plans of sufficient detail to model the proposed changes? Major data gaps or inadequacies in these areas are likely to affect the quality of the results. If additional data are required, what are the additional costs in time and money to collect it?
- Is there sufficient information to define any key hydraulic features likely to affect flow behaviour in the study area, such as detailed survey or drawings of works-as-executed?
- Does the extent and resolution of the data cover the study area with the required level of detail? Is the resolution consistent with the conceptualisation of the problem?
- How suitable are the available rainfall and streamflow data? For example, daily rainfall data may not be suitable for calibration in smaller catchments. If streamflow data are available, have sufficient gaugings been undertaken to develop a reliable rating curve for the flow magnitude of interest?
- Are the calibration data relevant to the problem? For example, peak flood levels for large flow events might not be suitable if the object is to estimate low-flow behaviour in a system to inform water resource management decisions.

The collection of new datasets can be time-consuming, so it is important to identify gaps in the data as early as possible. Of course, sometimes it will be difficult to identify all data requirements in advance. Some unexpected aspect of the flood behaviour may only become known during the modelling process or additional areas of interest may emerge. However, a thorough consideration of the data requirements in the early stages of the process should help to minimise stalling of the model development later. Additional survey data can be particularly time-consuming to collect. Chapter 5 provides a detailed treatment of data requirements for 2D hydraulic models.

Above all, avoid the temptation to proceed with the modelling process if the quality of data is insufficient to achieve the outcomes. Remember the often-repeated modeller's maxim: *Garbage in, garbage out!*

4.5. Planning the Modelling Approach

As Figure 4-1 indicates, the modeller should undertake a planning and review phase to obtain a broad idea of how the available data and chosen modelling package will affect the major phases of the modelling process. If the modeller has experience with the chosen modelling software and approach with similar applications, then they should have a reasonable understanding of the data requirements and model development process. Otherwise, the modeller should review the model software documentation to gain a preliminary understanding of the schematisation method for key features. The aim of this stage is to identify deficiencies in the data and/or modelling package that may jeopardise the timely fulfilment of the study objectives.

At the conclusion of the review, if serious deficiencies are identified which cannot be rectified by the acquisition of additional data, it may be necessary to revisit the initial considerations including choice of model, conceptualisation, and possibly the study outcomes.

4.5.1. Schematisation

As mentioned, the modeller should endeavour to determine whether the available data are likely to allow an adequate schematisation and calibration of the model, and is consistent with the requirements of the modelling software. Issues requiring consideration at this stage might include:

- Complexity of flowpaths – Will the major flowpaths be modelled in 2D or 1D, or a combination? If using 2D, what model resolution will be required to provide an adequate definition of the flowpath? Are the appropriate bathymetric data available for the intended schematisation method?
- Key hydraulic features - Can important structures be adequately schematised using the intended modelling approach? Are the appropriate data available to schematise these features?
- The built environment – To what level of detail will common urban features such as fences, gutters, walls and buildings be modelled? Is it necessary to model these features directly, or can their effects on flow behaviour be averaged in a broader conceptualisation?
- Boundary conditions – What type of boundaries will be used and where will they be placed? Will a “direct rainfall” approach be used? If using a combined 1D/2D model, where will the boundary between the 1D and 2D domains be placed and how will the interface be schematised?
- Interpolation/modification of data – Will the Digital Elevation Model (DEM) need to be interpolated or manipulated to be consistent with the model resolution?
- Dynamic topography – Is it necessary to consider changes to topography during the model simulation, such as river geomorphology, entrance changes at ICOLLs, scouring of embankments, or dam failure? The modelling approach should be able to implement the effects of these changes dynamically if they are an important aspect of the study.

Detailed guidance concerning several aspects of model schematisation is provided in Chapter 6.

4.5.2. Model Extent and Resolution

As described in Chapter 2, 2D modelling requires a trade-off between the number of grid/mesh[2] elements and the run time. The number of grid/mesh elements is determined by the size of the grid cell or the average size of the mesh cell and the model domain area. Reducing the size (or average size) of each grid/mesh cell, increases the number of grid/mesh cells for a given model extent. Increasing the number of grid/mesh cells lengthens the run time. To keep run times manageable (less than 2 days and preferably less than 1 day), it is currently recommended that the number of grid/mesh cells do not exceed one million. As computational resources improve (i.e. computers get faster), run times will reduce and the maximum number of grid/mesh cells recommended here will increase. However, there will always be a trade-off between the grid/mesh resolution and the run time. Techniques for maximising resolution and minimising run times are described in the following sections.

It is important to note that the primary consideration will always be that the model resolution enables significant hydraulic features, such as channels, storages and controls, to be adequately defined, such that a change in model resolution does not significantly alter the modelled results.

4.5.2.1. Model Extent

Minimising the 2D model extent will either allow a higher grid/mesh resolution **or** a shorter run time, all other things being equal. The study objectives will primarily determine the model extent. The 2D model may not extend across the entire study area if the modeller considers that parts of the system do not affect flow behaviour, or can be analysed using a 1D model or other approaches. Conversely, it may be necessary to extend the model beyond the study area to incorporate significant controls or suitable locations for boundary conditions. However, it is recommended that the modeller be aware of the importance of minimising 2D model extent.

In the majority of stormwater studies, the upstream catchment area is significantly larger than the study area. In such cases, the traditional approach is to use a hydrologic (runoff-routing) model to generate discharge hydrographs for input to the hydraulic model domain. Alternatively, it may be that the study area is sufficiently small that a direct rainfall approach might be adopted. Chapter 11 has further details on this approach.

4.5.2.2. Model Resolution

The resolution of model output data required often correlates with the size of the study area. For large study areas (such as a long river reach), broad scale results are generally required. Results such as the flood extent, peak flood levels, and proportion of discharge in major flowpaths are typically achievable at this scale. For such purposes, a relatively coarse grid/mesh resolution may be satisfactory if channels, flowpaths, storages and controls are adequately defined.

---

[2] This document refers to the uniform square or rectangular elements of a finite difference model as a **grid** and the quadrilateral, triangular or curvilinear elements of the flexible mesh models (such as finite element and finite volume models) as a **mesh**. Statements that are applicable to both types use the **grid/mesh** format of reference.

Finer resolution results and model elements may be required within more complex study areas. Complex study areas may require detailed analysis of flow behaviour along channels/roads, through hydraulic structures, or flow velocities and hydraulic loadings in and around buildings. In addition, a finer model resolution may also be required such that the hydraulic features are adequately represented.

4.5.3. Run Times and Computational Resources

The availability and type of computational resources will impact directly upon the efficiency and timeliness of any 2D modelling study. The efficiency of a 2D model study will be greatest where model run times are less than 24 hours. The shorter the run time the greater the efficiency. However, depending upon the extent and resolution of the model and the length of the modelled events there may be situations in which run times may be in the order of several days.

Excessively long run-times can introduce a significant bottle-neck in the study timeline and the decision to accept an excessively long model run time should be made carefully. Timeliness may be particularly affected during the calibration phase, where a large number of iterative simulations are necessary, mostly in series rather than parallel. In addition, the total number of runs required can be an important consideration if there are many scenarios to be considered, such as different design storm durations, development scenarios, blockage scenarios or scenarios to parameter sensitivity tests. During the planning stage, the modeller will need to consider the following factors to estimate the efficiency and timeliness of the study.

- The estimated length of time required to complete each run.
- The number of calibration and design events to be simulated
- The number of computers/processors available.
- The ability of the computers to undertake multiple runs in parallel or not.
- The number of licenses available if proprietary software is to be used.

The type and availability of computational resources can provide a real practical constraint. It may limit the number of design runs that can be achieved, the length of event that can be simulated, or the achievable resolution of the model. Such limitations and the resulting implications need to be identified as soon as possible in the process.

Consideration of run times can be particularly important for rural flood studies, or for studies involving continuous simulation of long flow periods. Such studies may require simulation of floods or flow sequences lasting several months. In these situations it may be appropriate to consider the use of a modelling package that can implement an adaptive timestep, using a longer timestep during periods of relatively steady flow conditions, which may significantly reduce computational run times. Adaptive timesteps are discussed further in Chapter 10.

4.5.4. Modelling Log and Naming Conventions

A modelling log is highly recommended and should be a requirement on all studies. The log may be in Excel, Word or other suitable software. An experienced modeller should review the modelling log. It should contain sufficient information to record model versions during development and calibration, file naming conventions and observations from simulations.

Model file naming conventions and locations are important in ensuring that simulations can be undertaken efficiently, with high traceability, and that old simulations can be reproduced as required. They also assist in minimising human errors. Successful model file naming conventions have the following characteristics:

- Files are named using a logical and appropriate system that allows easy interpretation of file purpose and content.
- A model version naming and numbering system (designed prior to modelling) should be included in input data filenames.
- A logical and appropriate system of folders is used that manages the files.
- Documentation of the above in, for example, the study's Quality Control Document and/or Modelling Log.

For example, if the name of the creek being modelled is "Rapid Creek", an "RC" within the filename would be recommended. If the topography (elevations and roughness) used for a particular simulation is that from 1974, the modeller may consider using "T74" within the model filename/s. If the flood event being modelled is the Q50 event, then Q050 may be included. Including the model version number is also important. If it has taken 7 versions of the model to reach the current point then the modeller may consider including "007". Thus, the filename may include the text string "RC_T74_Q050_007". This makes it immediately identifiable to the current modeller, reviewers and any future modellers. While modellers do not need to follow this specific example, it is highly recommended that a modeller does develop a robust naming convention and records this naming convention within the modelling log.

4.5.5. Calibration and Verification

As part of the review and planning process, it is beneficial to plan how the calibration will be undertaken.

- Which historical events are likely to be used for calibration/verification? How much data exists for each event? What is the reliability and relevance of these data? Were the events recent enough for present members of the community to remember? Is the spread in magnitude of the historical events similar to the spread in magnitude of the proposed design events? That is, will the model be calibrated, and therefore considered reliable, for both small and large historical events, or through relatively wet and dry periods for continuous flow sequences?
- What model parameters will be adjusted for calibration of the model?
- Are the available topography/bathymetry/control structure data consistent with the state of the catchment during the calibration events, or have there been major changes due to development, dredging, crossing upgrades or some other cause?
- Based on the available data, how much confidence will there be in the calibration? Will it be sufficient to achieve the desired objectives?

Calibration and sensitivity analysis are essential as they provide an indication of the uncertainty associated with the model results. Poor calibration results can highlight deficiencies in the schematisation of key features, or limitations of the historical data. Sensitivity analysis can provide direction during the calibration process, by indicating the model parameters or inputs on which the calibration results are most dependent, so the modeller can focus on reducing the uncertainty of those inputs. The primary consideration is that the calibration process should reflect the purpose for which the model is intended. Chapter 7 presents detailed guidance on calibration and model uncertainty.

4.5.6. Processing and Analysis of Results

The modeller is advised to consider:

- What modelling scenarios and events are likely to be required?
- Are there any key assumptions against which the sensitivity of the results will need to be checked?
- What outputs will be required from the model?
- What is the acceptable level of accuracy of the results?
- How will the model outputs be collated, analysed, and presented?

The output from hydrodynamic models can be quite complex in itself and the manipulation and presentation of results is a significant factor to consider, and one that is easily overlooked. The modeller should also be confident at the outset that the desired outputs can be extracted from the model, and that they are consistent with the requirements of the study objectives. For example, there are several different methods used in Australia to estimate hydraulic hazard at particular locations, usually involving a consideration of the velocity and depth of floodwaters. If one of the study objectives is to determine hazard, the modeller should give thought to how the hazard level will be determined, and whether the desired outputs are available. Will the maximum velocity from the entire event be used, or velocity at the time of maximum depth? Once again, forward planning at the early stage may prevent delays later in the modelling process. Detailed guidance on interpreting results is provided in Chapter 8.

4.6. Selection of Analysis/Design Scenarios

2D hydrodynamic models have traditionally been used with event-based approaches. Typically, the model is configured and run for scenarios of discrete duration such as a single storm, tide, or dam break event. The popularity of this approach can be attributed to:

- its being generally consistent with the use of design rainfall storms as documented in ARR87,
- limitations of computational resources, and
- availability of calibration data in urban areas is often limited to peak flood levels and/or photographs from large historical storms.

Other less traditional modelling approaches may also be worth considering, depending on the problem under consideration and data availability. These include:

- Stochastic analysis;
- Continuous simulation; and
- Real-time operational forecasting.

However, the inherent limitation given the inclusion of 2D domains is likely to be run time. Although less traditional methods (non-event based) may be desirable in certain circumstances, run times for 2D models currently do not invite their utilisation in continuous or stochastic (Monte-Carlo type) applications.

4.7. Identification of Deficiencies

As prescribed throughout the modelling process, deficiencies in knowledge and/or data need to be identified and either corrected or appropriately accommodated. Deficiencies can create a need to revisit some of the earlier processes to test previous decisions and assumptions. For example, if it is found that model calibration data (such as flood levels) have not been formally recorded, it may be necessary to expand the community consultation process to ensure anecdotal and non-recorded data are captured. Following this, if sufficient calibration data are still not available, it may be decided that the high level of uncertainty in the uncalibrated model outputs may make the original objectives unachievable and the scope of the study may change, before modelling even commences. There are many other examples that illustrate the iterative and cyclic nature of the modelling process and it is important that the modeller recognises that the process is not a linear one.

4.8. References

Pilgrim, DH, (ed)., (1987) *Australian Rainfall & Runoff - A Guide to Flood Estimation*, Institution of Engineers, Australia, Barton, ACT

5. CHAPTER 5 - DATA REQUIREMENTS

5.1. Introduction

This chapter discusses the different types of data required to undertake 2D hydraulic modelling of floodplains. In particular, this section focuses on different data collection methods with their accuracy, practicality, and cost effectiveness assessed. In addition, the role of input data in underpinning the reliability of hydraulic modelling outputs is explored. Further, the gathering of required data from both public and private sector sources is reviewed.

Communication of hydraulic modelling outcomes is an important aspect of flood investigations, improving agency and community understanding of flood risk. A brief discussion of data presentation techniques is provided.

At the most fundamental level, numerical models can be thought of as sophisticated tools for the interpolation and extrapolation of known data sets. This interpolation and extrapolation of data can be both spatial and temporal, and in the case of a flood study, an interpolation or extrapolation of risk probability.

In the context of flooding, 2D numerical models are typically used to expand point water level and flow data measured at discrete locations into 2D maps of water surface level, water depth and distributions of flow velocity across a floodplain. This process, often undertaken as part of a model calibration, expands known data sets over an area of particular interest. Once the reliability of a model to reproduce known datasets is established, the same model is often used to extrapolate flood risk from the observed range to a risk level useful for planning and design (e.g. a 1% AEP risk level).

While the models use detailed and sophisticated methods for data interpolation and extrapolation, the accuracy and reliability of model outputs is intrinsically reliant on the measured data sets used as the basis for the model operation. For instance, if the stated aim is for modelled water level outputs to have a vertical accuracy of +/-0.2m, then it follows that, if everything else was perfect, the input datasets require an accuracy at least equal to, but preferably better than, the accuracy required for the model outputs. The same is true for the spatial resolution and horizontal accuracy of model outputs.

In general, it is advisable that modellers seek out the best available data sets in each instance and assess the accuracy of those data sets in respect of the required outputs.

Data required for hydraulic modelling can be classified by the following purposes:

- Model development
- Model calibration/verification
- Model application/presentation

In the model development phase, a broad understanding of the flood behaviour within and around the area of interest (level, extent and velocity), varying with flood magnitude, is required. Developing a general knowledge of flood behaviour in the study area informs the model development process including selection of appropriate model software and application. Within a given study area, the following datasets can influence flood behaviour:

- Local and floodplain topographic features (*topographic information*).
- Drainage infrastructure (*structure information*).
- Channel and floodplain vegetation and landuse (*hydraulic roughness information*).
- Downstream controls on flood behaviour.

Once a hydraulic model is developed, the validity of the model needs to be assessed. Typically, model calibration/verification involves the comparison of model predictions against observed flood behaviour. Information on observed flood behaviour includes:

- Observed/estimated flow rate and volume (including hydrograph shape and timing) from upstream of the area (*historical hydrologic information*).
- Observed flood levels and extents (*historical flood information*).
- Anecdotal information from local stakeholders describing flooding (whilst caution needs to be used in applying this type of information, it can be critical in areas that lack formal flood records).

The quantity and quality of the collected data has a direct influence on the accuracy/reliability of the model application. Good understanding of the primary flow behaviour underpins effective data collection for model development and calibration.

Generally, hydraulic modelling outputs are applied within a risk management framework. The assessment of flood risk combines the likelihood and consequences over a range of flood magnitudes. The evaluation of the likelihood requires an understanding of flood frequency (*design hydrologic information*) at the study area. The assessment of consequences requires knowledge of the property and infrastructure (*property information*) affected during a given flood event.

The collection of the information discussed above, varies in scope and size (cost) in line with the complexity of the flood behaviour characteristic(s) of primary interest. It is important to emphasise the need to understand the nature of flood behaviour during the project conceptualisation so that data requirements can be focussed on adequately supporting the required study outcomes. Data collection for a large and complex floodplain investigation may be extensive and costly, hence opportunities to source/share data from a range of agencies is desirable (e.g. a sewer authority may have substantial floor level survey records and drainage infrastructure details).

5.2. **Data Requirements Fundamentals**

5.2.1. Overview

When considering the data requirements for 2D hydraulic modelling, the following three fundamental principles are suggested to provide guidance:

- A sound understanding of the flood behaviour and hydraulic processes leads to the scoping of robust and rigorous data collection;

- The (expected) reliability of hydraulic modelling estimates is dependent on the collective quality and quantity of all the data inputs; and
- Sharing well-documented data between agencies leads to cost-effective hydraulic modelling.

This section discusses the application of the above principles to 2D hydraulic modelling.

5.2.2. Understanding Flood Behaviour

The scope and focus of hydraulic modelling investigations varies with the primary flow/flood behaviour of interest. Typically, flow/flood behaviour is characterised by the following:

i. channel flow behaviour in watercourses (in-bank flows in rivers and creeks);

ii. floodplain behaviour (overbank flooding from a river or creek channel)

iii. structure/culvert/bridge/pipe performance

iv. overland flow behaviour (including flash flooding/drainage typically in urban environments)

The requirement for hydraulic modelling is most often driven by a need to quantify flooding impacts on development, whether that occurs in either an urban or rural setting and for both existing and future planned development.

The second and third of the above flood behaviours are principally concerned with flow magnitude up to channel/pipe capacity. Given this concern, the data requirements for model construction, calibration and application, are centred on the channel/pipe characteristics and capacity, and the hydrologic conditions that produce a flow magnitude that exceeds the channel/pipe capacity.

The longitudinal scale associated with the above characteristics also influence data collection. The influence of the hydraulic performance of a structure is generally limited to the immediate surroundings. Hence, data collection should be concentrated on the immediate surrounds of the structure and, on the hydrologic behaviour at the structure.

Once channel/pipe capacity is exceeded, during large flow events, the characteristics of the overland/overbank areas strongly influence flood behaviour. This is also true for overland flowpaths that develop by the accumulation of rainfall runoff in upper catchment areas prior to discharging into a more formalised drainage channel or pipe.

Representation of flood behaviour on overland flowpaths and storages relies on adequately defining the topography of the affected overbank area and any drainage features that might influence the passage or storage of these flows.

In urban areas, data collection needs to consider the pipe/pit infrastructure, topography and land use of the overland flowpaths. Significant complexities in flood behaviour along an overland flowpath arise from the influence of obstacles to the flow such as buildings (residential, commercial and public), fences, roads, earthworks and embankments. Data collection needs to account for the key hydraulic controls. This may result in an extensive and costly collection process, particularly for topographic data.

In rural areas, data collection similarly needs to consider the topography of likely overland flowpaths and any infrastructure (e.g. road embankments) or drainage features (e.g. levees and drainage canals) likely to influence flow passage.

5.2.3. Data Quality and Hydraulic Model reliability

Common questions asked about hydraulic modelling outputs are:

- How reliable are the modelled flood levels and extents?
- What is the uncertainty surrounding the modelled flood levels and extents?

As often the hydraulic modelling outputs are utilised within a risk management framework, the answers to the above questions fundamentally underpin investment decisions made by government agencies and private sector parties.

The reliability and uncertainty of the modelled flood behaviour is influenced by the collective quality of the hydraulic model input and calibration/verification data (Bishop and Catalano, 2001). For example, the reliability of the design flood extents may be compromised by uncertainties in hydrologic data (design flood estimates), even though a good quality topographic data has been employed in the model development. Further, if limited historical flood information is available to calibrate/verify the hydraulic model, confidence in modelled outputs is reduced.

At the beginning of a hydraulic modelling application, it is important to set realistic expectations of the hydraulic model outputs from an understanding of the available data. It is also important to recognise that data collection is an expensive exercise and that typically, additional orders of data accuracy are reflected in a respective increase in data acquisition costs. A holistic review of the ultimate use of the model outputs, the available historical data for generating catchment runoff (hydrological data) and data available for model calibration should be integral to study planning and directly influence the required accuracy (and costs) of supplementary data collected to complete the essential model data set.

5.2.4. Elevation Data Accuracy

The accuracy of elevation data are usually referred to in terms of a +/- value. For example, one set of elevation data may have a stated vertical accuracy of +/-0.1m. Typically, unless stated otherwise, stated accuracy values reference one (1) standard deviation (also known as 1 sigma or 1σ). That is, the accuracies of the survey data collected should form a normal distribution bell curve with the average accuracy being 0m and the standard deviation being 0.1m. It is not within the scope of this document to explain statistical distributions in detail and the modeller must seek their own references if further statistical information is required. However, consider the simple example of 100 points, each with a *true* elevation. If the elevation of these 100 points is measured by a uniform means with a measurement procedure capable of achieving an accuracy of +/- 0.1m, then the resulting 100 *measured* elevations will have an accuracy of +/-

0.1m, to one standard deviation. This means that about 68% (1 standard deviation) of the *measured* elevation points will be within 0.1m of the *true* elevation. The remaining 32% of the *measured* elevations will be more than 0.1m higher or lower than the *true* elevation. This is a significant number of measured elevations outside the stated accuracy and this fact must be well understood by the modeller when dealing with elevation datasets. Horizontal accuracies also follow the probability distribution laws and are stated to one (1) standard deviation.

In relation to the above example of the 100 measured elevations, it is also important to consider the location of these points. To extend the above example, let us say that the method of measuring the elevation of these points was an airborne technique (refer to Section 5.3.1.5 for further details on airborne techniques). Let us also say that the 100 points lie across a golf course that has a heavily wooded gully between the fairways. Half the points lie on the fairways and half the points lie between in the gully under the trees. Given that aerial methods generally provide the most accurate results in the open on smooth/hard surfaces, it would be no surprise that the 50 measured elevations that lie on the fairway are all well within the one standard deviation stated accuracy of 0.1m. However, the measured elevations in the gully under the trees are far less accurate. Only 18 of the 50 measured in the gully lie within the 0.1m standard deviation. The remaining 32 measured elevations are outside the 0.1m standard deviation and some are more than 1m higher and lower than their true elevation. The point of this over-simplified example is to demonstrate that statistical accuracies can very often be skewed in relation to location and the modeller will need to beware of assuming that the stated accuracy applies uniformly across the study area. In reality, post-processing of the aerially-collected data *should* exclude/ignore locations where the ground is not sighted (e.g. under heavy tree cover). This process *should* leave the dataset almost blank in this heavily wooded region. However, if the blank area is not supplemented by survey data from another source, the Digital Terrain Model (DTM) that is subsequently created will fill this heavily wooded blank region by interpolation between fairway elevations. What is in reality a gully, becomes an interpolated fairway. The difference between the DTM elevations and the true elevations can be substantially different, and bear no resemblance at all to the stated accuracy in that area.

Survey data sets, especially data sets measured by aerial (remote sensed) techniques may also exhibit a bias. That is, a constant over or under estimation of the actual surface value. In aerially sampled data sets, bias in the data may occur due to errors in the raw data measurement. For example, due to poor ground control or a fault with the instrument or may be a function of the sampling techniques used to thin the data set.

The stated accuracy of a dataset is a critical concept to understand, as in some studies the modeller may have responsibilities that include:

- Scoping and writing the survey brief;
- Checking stated accuracies in datasets provided by another party;
- Scoping a survey specifically for the purpose of checking another dataset;
- Deciding which dataset should have priority in overlapping regions;
- Using overlapping regions to check datasets;
- Advising whether a dataset obtained for a purpose other than a flood/flow study is suitable for use for a flood/flow study based on stated accuracy;

- Explaining apparent discrepancies in input topographic data;
- Explaining why recorded historical peak flood levels are sometimes below the level of a DTM; and
- Explaining why historically inundated areas, are not predicted as being inundated within the modelled results.

All references to accuracy values in this document are to one (1) standard deviation.

5.2.5. Coordinate Systems and Datum

All topographic data applied in the development of a hydraulic model require indexing both horizontally and vertically in space to an adopted origin. Typically, spatial data are either specified with reference to an (approximately) horizontal plane using Cartesian coordinates or to the spherical earth by using latitude and longitude. More often than not topographic data will use a Cartesian coordinate projection.

Survey data with Cartesian coordinates is referenced to a specified projection of a grid on the earth's surface with local axes of zero in the east–west and north-south directions. Survey data are then referenced to the defined origin of the plane in as either single x, y (horizontal) and z (vertical) points or, in the case of contours or 3D polylines, strings of x, y, z points. The 'horizontal' plane is typically referenced to a defined surveying spheroid of the earth's surface and survey coordinates referenced to a defined origin on the spheroid. Cartesian distances from the grid origin are measured in metres. Typically, data provided by surveyors in Australia is referenced to a local Australian standard Cartesian coordinate system. These coordinate systems divide the country up into a series of grids, each referring to a local 'false' origin for horizontal coordinates. The extents of these local grids are sized so as to minimise errors due to the curvature of the earth, but significant errors in position due to curvature can occur as distances extend from the grid origin.

Several Cartesian coordinate systems have been in common use in Australia in the preceding decades. These systems such as the Map Grid of Australia (MGA – in use from 2007), the Australian Map Grid (AMG66 in use from 1966 and AMG84 in use from 1984) have origins based on Universal Transverse Mercator (UTM) projections. Other local systems such as the Integrated Survey Grid (ISG) in NSW have also been in common use. While all these systems use similar projections, care should be taken when converting from one coordinate system to another as subtle errors can occur. Extra care should be exercised when assigning projections to data within GIS and CAD systems. The simple error of mixing AMG data with MGA projections can lead to significant errors in data location.

Typically, vertical heights are referenced to Australian Height Datum (AHD), which is a reference that was derived in 1971 by fitting a surface through the mean sea level at thirty tide gauges around the country. Prior to 1971, numerous local datums were used for vertical measurement and some bathymetric surveys are still, though less commonly, referenced to the local tide gauge datum.

When developing a hydraulic model, it is possible that the data sourced for the model may come from numerous surveys, from various eras with differing horizontal coordinate systems and vertical datums. Extreme care should be taken to firstly identify without doubt which coordinate system and datum each data set refers to. The modeller should then adopt a single coordinate system for model development and study output. While some modelling software systems and GIS software allow for the combination of data sets in various coordinate systems automatically, it is good modelling practice to independently verify the spatial reference of the various data sets. While there are no 'rules of thumb' for comparison various coordinate systems have subtle origin offsets and grid rotations which can introduce subtle, but nonetheless critical errors into model topographies and/or calibration data sets. Where possible, a spot check of overlapping data in known locations e.g. kerb and gutter on a particular intersection should be undertaken to validate the data coordinates. While not always a simple exercise, it often pays dividends in the long run to convert all data to a single coordinate system so as to expose these errors and minimise further potential confusion.

5.2.6. Data Sourcing and Sharing

As mentioned, data collection for hydraulic modelling may be extensive and costly, depending on the scope and focus of particular investigations. The reliability and uncertainty of the modelled outcomes is founded on the quality of the data employed in the investigation. Hence, cost-effective data collection can enhance model reliability for a given investigation budget. In fact, investment in data collection usually improves reliability of model output.

Various agencies and private organisations have the need to collect data relevant to flow/flood behaviour investigations, particularly topographic data. This data may have been collected without hydraulic modelling as the focus. For example, a road authority may collect topographic data to enable the design of a proposed road. This data could contain waterway cross-sections and/or floodplain terrain that may be utilised for hydraulic modelling. Water authorities collect data for management of irrigation areas or for water resources assessment and this data may also include flood data that is relevant for the requirements of this type of project. The Bureau of Meteorology and local authorities provide flood forecasting and warning systems and this data may also be valuable for model development and calibration.

It is important that a range of agencies are consulted early in project development to ensure that as much useful data as possible is included. Sharing data is therefore important and all agencies gain from this sharing. Therefore, this sharing should be encouraged.

5.2.7. Data Handling and Metadata

2D numerical models addressing a floodplain of any size typically require a relatively large amount of data. These data are primarily spatial in nature and require specialised spatial analysis software with suitable algorithms and techniques for efficient data viewing, sampling and analysis, and storage.

In the process of undertaking a 2D modelling study, a modeller can reasonably expect to start with a large data set, rationalise this into a more workable but still large series of data files while developing the model, and then multiply this data set many times as output data to each model simulation.

It is sound modelling practice to catalogue these numerous files by both naming them consistently and tagging them with “metadata”. Effective file naming conventions ensure files are stored in an ordered manner. Metadata are summary index data fields that can be more easily located and interrogated. Effective metadata systems allow for easy location of appropriate data sets without relying only on a system of data file naming or the time-consuming alternative of opening and viewing/interrogating short-listed files.

Methods and functionality for tagging and cataloguing model files and base data are included as part of the databasing and user interface of some modelling software systems. Often though, the model interfacing is aimed at the core functions of model development (which often assumes base model data is presented in a pre-processed format) and model simulation and a limited amount of model output result post-processing.

Databasing, cataloguing and manipulation of core data sets is often better handled by a specialist spatial analysis system such as a Geographical Information System (GIS) or a Computer Aided Design (CAD) system, which address these metadata and databasing requirements.. GIS and CAD have various strengths and weaknesses. Typically, the key differences between them are:

- GIS has a more powerful functionality when it comes to relational comparisons of two or more data sets. For example, comparing flood extents to a property database or a land use data set, whereas
- CAD systems are better able to produce engineering designs from flood model outputs and in some instances are more efficient at handling large DTM data sets.

2D modelling systems typically use a pre-processed subset of ground surface data as a core input. Other than when prepared especially for a model to a very tight specification, available ground surface data sets most often need to be processed into a resolution and a format compatible with the modelling system.

Most modelling systems have methods for pre-processing of spatial data into a compatible format. However, as the core focus of most modelling systems is on the computational solution of the flow equations, systems often have more basic processing tools. GIS and CAD systems as previously stated are designed specifically for handling a range of different data forms and have specialised and customisable algorithms for producing both irregular and regular gridded surfaces, which service most 2D model formats.

In summary, the focussed data handling of both raw spatial data and metadata, the ability to efficiently interrogate and resample data, and the superior ability of GIS and CAD in respect of data presentation and mapping means that most modelling systems currently perform more effectively when used in tandem with GIS or CAD.

GIS and CAD have traditionally been focussed on static plan/map data and have not typically handled temporal (time series) data effectively, if at all, without significant customisation of the software. While this aspect of GIS is rapidly changing, at the time of writing it is more than likely that storage of temporal data such as water level, flow or rainfall time series will occur separately from the other spatial data sets required for model development.

5.3. Model Development Data

5.3.1. Topographic and Infrastructure Data

5.3.1.1. Overview

At their core, 2D hydrodynamic models solve a mathematical approximation to the physically based laws of water flow. It follows then that the physical inputs to the model have the most influence on the model outputs, with the model solution scheme and its varied parameters assuming a secondary influence on outputs.

Topographic and infrastructure data are the primary inputs needed to construct a hydraulic model. Typically, topographic data collection can be 40 to 60% of the project budget for a flood investigation where no existing data are available. Proper scoping of topographic and infrastructure data collection can have a significant impact on the cost effective delivery of hydraulic modelling investigations.

The scope of the required topographic and infrastructure data is driven by the nature of flood behaviour for a given area. The desired elements of topographic and infrastructure data can include:

- channel cross-sections,
- waterway structures (weirs, levees, regulators, culverts and bridges etc),
- overland flowpath definition, and
- drainage infrastructure (pits, pipes etc).

The mix of the above elements used in a particular hydraulic modelling application is determined from the scale of the key flood behaviour processes. These elements can be collected via airborne (remote sensed) and field techniques. Further discussion of the elements, specific techniques and their application is provided in the following sections.

5.3.1.2. Field survey elements

This section discusses the scoping of the field survey component for a flood investigation. The focus is on which features need to be captured with survey. Key elements requiring field survey for a flood investigation may include:

- Channel and overland flow path cross-sections (and potentially long-sections)
- Structures – weirs/regulator, bridges, culverts, siphons
- Drainage infrastructure
- Road and rail embankments, Levees
- Property data (floor level, type, size)

Sourcing existing field survey data from other agencies may offset scope and cost. With data sourced from other agencies, it is important to obtain relevant metadata (date of capture, methods, accuracy etc.) to assess whether the data are suitable for the purpose of hydraulic modelling.

Cross-sections

The scope of the cross-section survey depends on the nature of the hydraulic modelling application. The lateral extent and longitudinal spacing of cross-sections are the key parameters in scoping cross-section survey.

The lateral extent of the cross-section must be sufficient to include key in-bank elements. Erring on the conservative side is wise with the cross-section extending up to at least top-of-bank height. When surveying in-bank cross-sections, good field notes and/or photos describing the nature of the channel are vital for proper interpretation afterwards. There are numerous references such as Stewardson and Howes (2002) that describe flood study cross-section survey requirements in detail. The overriding principle being that the cross-sections adequately describe the shape and slope of the channel so that the flow conveyance capacity of the channel can be calculated.

The influence of in-bank features on floodplain flow behaviour tends to reduce as flood magnitudes increases. For example, bank-full capacity may represent 100% of a 20% AEP flow but less than 10% of a 1% AEP flow. However, the capacity of the channel determines the engagement of the floodplain. Cross-sections in a floodplain investigation can be more widely spaced than for in-bank investigations. Ideally, cross-sections are provided at break-out points to the floodplain and at all locations where there is a significant change in channel characteristics, such as conveyance. Survey breaklines along the tops of the banks provide additional aid for the hydraulic modelling of overbank breakouts.

Field practicalities such as bank vegetation, access to the channel and water depth may influence the actual location of the cross-sections surveyed.

Structures and drainage infrastructure

Structures in the waterway and on the floodplain may have a significant influence on flood behaviour. These can include:

- Levees/embankments (such as road and rail)
- Bridges/culverts/weirs etc.
- Drainage infrastructure (pipes and pits)

Often such structures constrict and/or obstruct flood flow. The effects on flood behaviour may be intentional (such as a weir) or unintentional (such as a blockage due to a road embankment). Regulators generally act to control flow behaviour in accordance with a particular management strategy. The impact of these structures may vary with flood magnitude. A levee or embankment may prevent inundation in a given flood, but in larger floods the levee may be overtopped and/or fail with significant inundation behind the levee.

Geometric details of these structures are required for the hydraulic model development. These details include:

- Levees/ embankments
- Crest elevation
- Condition
- Continuity
- Cross-section
- Bridges / culverts / weirs
- Dimensions (width and height)
- Invert and obvert elevations
- Adjacent embankment elevations
- Elevation and type of handrails and/or guardrails
- Deck Overtopping level(s)
- Number of Spans/Barrels
- Geometry/Levels of any suspended services present across the inlet or outlet.
- Drainage infrastructure
- Stormwater pit/outlet coordinates
- RL of kerb
- RL of lintel/grate
- Size of lintel/grate
- Stormwater pit dimensions
- RL of bottom of pit
- Invert of incoming pipe
- Size of incoming pipe*
- Offset of incoming pipe
- Invert of outgoing pipe
- Size of outgoing pipe*
- Offset of outgoing pipe

* Note that the angle of pipe into/out of pit can be calculated from pit coordinates.

Other information, such as condition of hydraulic infrastructure, may also be collected. While this type of information may not be directly relevant to the study, the client (such as an LGA) may wish to update or create an infrastructure database to assist them with long-term planning.

Again, field practicalities such as vegetation, access and water depth/flow may influence the location and details surveyed for a given structure.

5.3.1.3. Field Survey Techniques

The techniques used by the data collectors, typically surveyors, to capture the elements presented in the previous section are discussed here. Typical accuracies for each method are provided in

Table 5-1. The techniques to measure topography and other survey features generally fall into two main categories:

i) Direct measurement where the survey technique involves a ground based instrument measuring features by physical contact and relating the measurement directly related to know ground control such as a State Survey Mark (SSM); or

ii) Remote sensing where features are measured without physical contact with the object, and generally refers to measurement by an airborne or satellite mounted instrument.

Traditional Ground Survey

Traditional survey (that is, survey collected by total station ground survey techniques) is the most accurate survey technique with vertical and horizontal accuracies as shown in

Table 5-1. However, the method is manual and labour intensive, making it best suited to smaller or difficult areas.

In the context of a flood study, traditional ground survey methods are often used for:

- checking remote sensed data sets, and/or
- supplementing remote sensed data in:
  - areas that are impenetrable from the air (i.e. satellites and aeroplanes cannot sense the ground)
  - areas that are critical to the operation of the model (i.e. critical hydraulic controls such as levees and weirs) where accuracy of ground level data are of prime importance.
-

Real Time Kinematic (RTK) GPS / Differential GPS

RTK GPS involves the coordinated use and comparison of two GPS (Global Positioning Systems) using the same satellite signals. The first GPS, sometimes known as the “base station”, is set-up over a permanent mark and maintains a continuous record of the location relative to numerous satellite positions. The second, or roving GPS, which can be hand held or mounted to a car, boat or all terrain vehicle, simultaneously collects information defining the roving GPS position from the same satellites. The accuracy of the roving GPS position is then enhanced by comparison of the satellite signals for the base station GPS. Figure 5-1 shows the principle behind the RTK GPS.

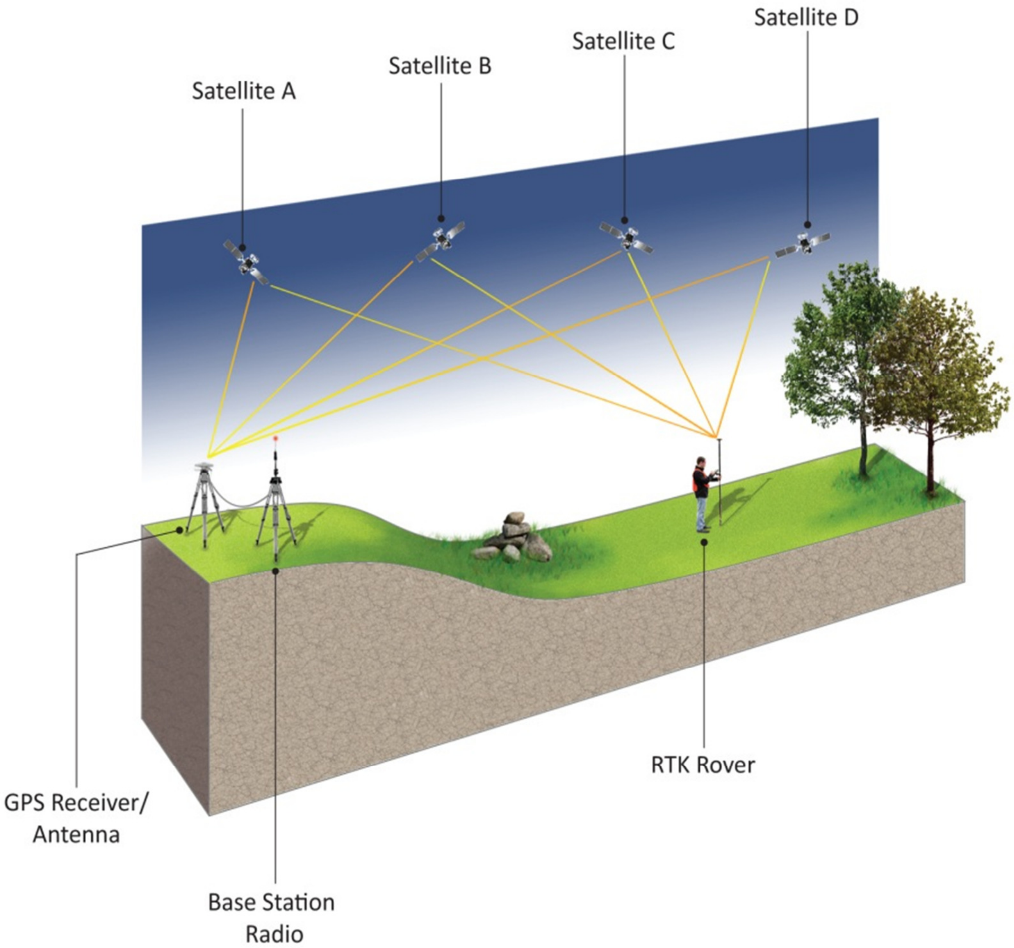


Figure 5-1 Principle of the RTK GPS Method

Table 5-1 contains typical accuracies for RTK GPS. RTK GPS has the advantage that it can collect a reasonable amount of data at higher accuracy than remote sensed data and much faster than by traditional means. The disadvantage is that if the vehicle in which the RTK GPS is mounted is unable to access an area, the system may have to be dismantled to gain access by some other means or measurement of data is not possible in the inaccessible area. RTK GPS methods also rely on the instrument having line of sight access to an array of satellites. This can be a limitation to the technique in areas underneath a tree canopy.

Table 5-1 Typical Accuracies of Field Survey

Survey Technique	Nominal Accuracy (+/- m)	
	Vertical	Horizontal
Traditional Ground Survey	0.01	0.01
RTK GPS	0.05	0.05
Photogrammetry	0.1 – 0.3	0.2 – 0.5
ALS (LiDAR)	0.15 – 0.4	0.2 – 0.5

5.3.1.4. Bathymetric (Underwater) Techniques

Many methods for collecting ground surface data are not applicable to collecting bathymetry in permanent or semi permanent water bodies. Where a relevant water body within the study area has not been adequately surveyed previously, a specific survey will be required to supplement the ground surface data for the purposes of the study.

If the water body is shallow or small then a traditional survey technique may be suitable. The survey staff may be deployed by a surveyor’s assistant in waders or from a stable platform like a bridge or boat.

In deeper, larger water bodies, a specialised sonar system may be required to measure the bathymetry. Instruments such as echo sounders, side scan sonar systems and acoustic doppler profilers can measure equivalent data sets for underwater surfaces. These data will then need to be carefully merged with overbank data sets as discussed further in Section 5.3.1.6.

5.3.1.5. Airborne techniques

Overview

Flood behaviour within an area containing overland flowpaths requires extensive spatial capture of topographic data. Aerial techniques are well suited to the capture of topographic data across a broad area. Typically, in a 2D fixed grid hydraulic model, overland flowpaths are represented as a regular grid of spot elevations. It is important to ensure key linear features such as levees, embankments and other infrastructure are adequately represented. These features can be incorporated into the topographic description using field data as breaklines.

The capture of aerial survey data is considered a more cost effective technique than field-based methods over extensive areas. Before embarking on any aerial data capture exercise, it is worth liaising with other agencies (rural water authorities, local council etc.) to check on existing aerial coverage.

In Australia, there are two principal techniques used in aerial survey:

- Photogrammetry
- Airborne Laser Scanning (ALS) also known as LiDAR (Light Detection and Ranging)

Generally, photogrammetric topographic data consist of a spot elevations plus linear breaklines to define changes in grade. Raw ALS data consist of a dense cloud of spot elevations classified into ground and non-ground strikes. The raw ALS data usually has the non-ground strikes removed prior to provision and is also thinned to yield gridded data. Further processing can produce breakline data, but this is not straightforward and is discussed in detail in the forthcoming “ALS” Section. Neither method can penetrate water surfaces. Only the water surface level is able to be measured. If the bathymetry under the water surface is relevant in the context of the numerical modelling, bathymetric data must be collected and incorporated separately. Similarly, neither method can penetrate dense vegetation (such as trees, sugar cane and mangroves) to produce ground elevations. Although, as photogrammetry spot heights rely on aerial photographs, ground spot elevations can sometimes be manually extracted if the ground is visible between the stands of vegetation. However, ground survey by field techniques may be necessary to fill the gaps in elevation data under heavy vegetation.

Further discussion of these two aerial survey techniques is provided in the following sections.

Photogrammetry

Photogrammetry is a measurement technique where the 3D coordinates of an object are determined by measurements made from a stereo image consisting of two (or more) photographs. This “stereo pair” are usually taken from different passes of an aerial photography flight. In this technique, the common points are identified on each image. A line of sight (or ray) can be built from the camera location to the point on the object. It is the intersection of its rays (triangulation) which determines the relative 3D position of the point. Known control points can be used to give these relative positions absolute values. More sophisticated algorithms can exploit other information on the scene known as priori.

The accuracy of the photogrammetric data is a function of the flying height, scale of the photography and the number and density of control points. Typically, the accuracy requested when scoping photogrammetric data collection for flood study purposes ranges from +/- 0.1 m to +/- 0.3 m.

As the technique is based upon the comparison of photographic images, shading and obscuring of the ground surface by vegetation can reduce coverage in specific areas. However, as photogrammetric analysis can utilise manual inspection of the stereo pair of photographs, the photogrammetrist is sometimes able to pick the odd ground surface visible through tree or crop cover. In this way, photogrammetry is sometimes able to provide some reliable points in vegetated areas.

Figure 5-2 shows a region over which photogrammetric data coverage and ALS data coverage will be demonstrated. Figure 5-3 shows a typical sample of raw data from a photogrammetric collection technique with Figure 5-4 showing the raw data in finer detail for the zoom area indicated on Figure 5-3. These figures demonstrate the following specifically in relation to photogrammetry:

- The measured points are well spaced, but not always on a grid. Some manual manipulation has occurred in locating these points when necessary.
- Breaklines (seen as intervals along which measured elevations form the vertices of the interval) are evident along tops and bottom of banks. These are most likely to have been manually derived by viewing the stereo pair of photographs.

Figure 5-2 Aerial Photo of Example Region A

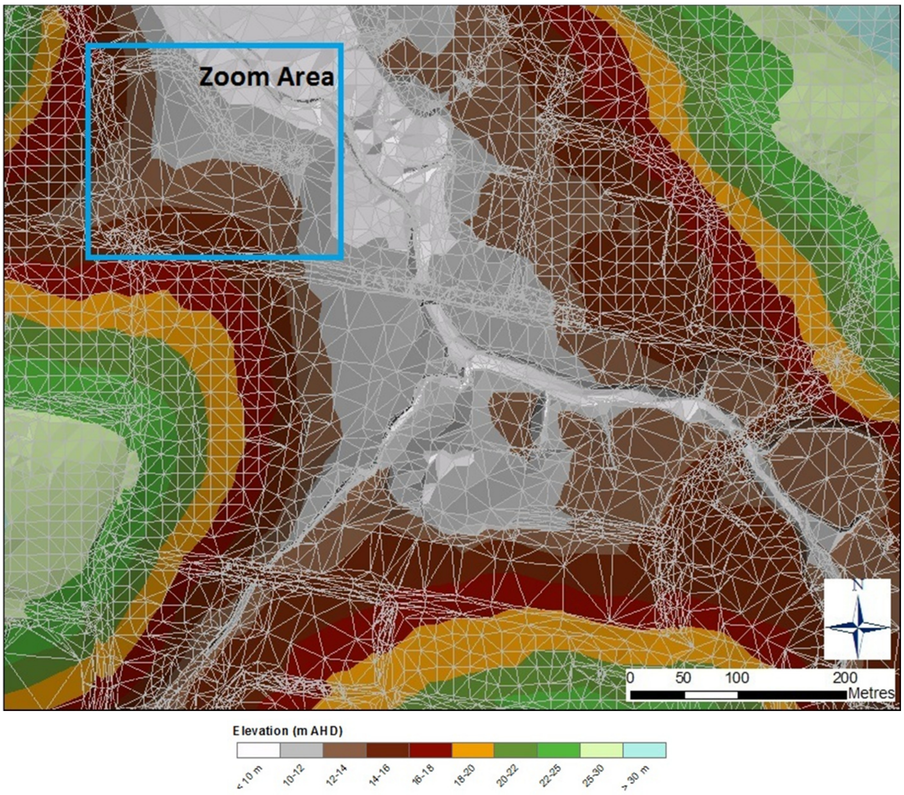


Figure 5-3 Sample of Processed Photogrammetry Dataset (Region A)

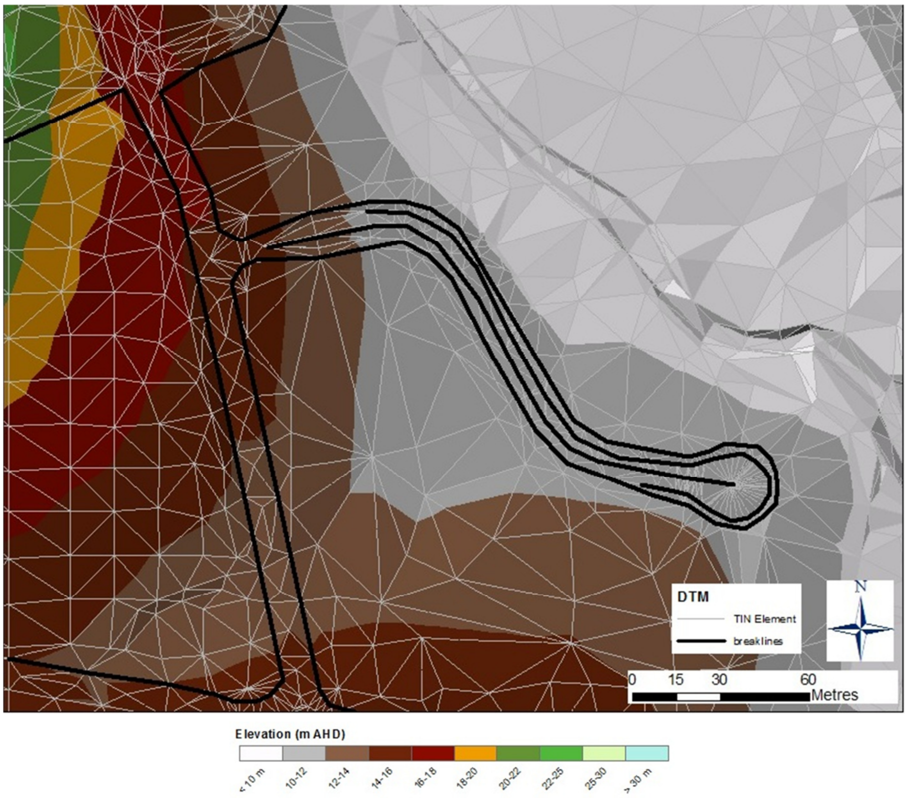


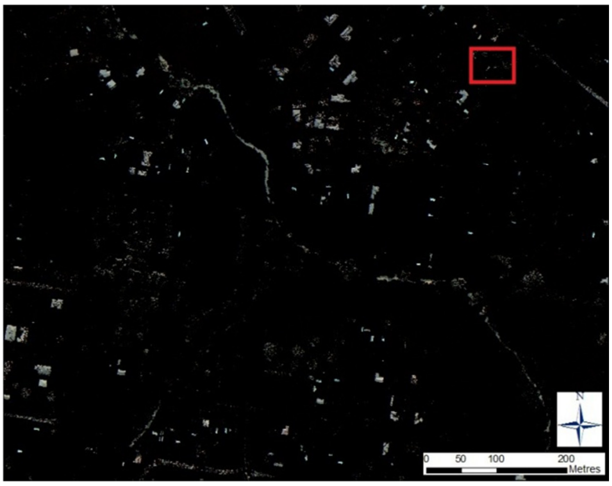
Figure 5-4 Sample of Processed Photogrammetry Dataset (Region A detail)

Photogrammetry is also often used to develop land surface contours directly as polylines with an attributed elevation to create an elevation dataset.

Airborne Laser Scanning (ALS) (or LiDAR)

Airborne Laser Scanner consists of a high frequency laser emitter and scanner, coupled with a Global Positioning System (GPS) and an Inertial Measurement Unit (IMU), all mounted in fixed winged aircraft. Rapid pulses of light are fired toward the earth by the laser instrument. These light pulses subsequently bounce back once they have rebounded from a target. The scanner records the time differential between the emission of the laser pulses and the reception of the return signal. The time taken is used to determine the distance between the emitter and the target. The position and orientation of the scanner is determined using differential kinematic GPS and the Inertial Measurement Unit (IMU) to account for aircraft pitch.

ALS produces a dense cloud of points (See Figure 5-5), which can be classified as ground or non-ground points. ALS requires little ground control in acquisition. However, while inaccessible or sensitive areas may be readily scanned without the need for ground survey, accuracy of the collected data may be low. That is, ground control is important for quality control.



(a) (b) Zoom area indicated by red square in (a)

Figure 5-5 Sample of Raw ALS Dataset (Region A)

The vertical and horizontal accuracy of ground surface level measurement by ALS is a function of the laser specification, flying height, ground control and surface coverage. Hard road surfaces, for example, are normally measured accurately, but other types of surface must be treated with caution as to whether the required levels of accuracy are being met. Note that all accuracy values stated for ALS data are in reference to clear hard ground. For clear, hard ground (i.e. ground with no surface coverage), the following provides guidance on nominal accuracy based on typically applied technology in Australia:

- Horizontal accuracy: 1/3000 x altitude at which the aeroplane is flown (i.e. for a flying height of 1000m, the horizontal accuracy is about +/-0.33m)
- Elevation accuracy:
  - <+/-0.15m @ 1100 m flying height
  - <+/-0.25m @ 2000 m flying height
  - <+/-0..4m @ 3000 m flying height
- Typical swath widths vary with flying altitude as follows:

Typical swath (m)	Altitude (m)
800	1100
1456	2000
2184	3000

A disadvantage of the ALS data capture method (compared to say low-level photogrammetry) is the absence of breaklines in the data to define distinct, continuous topographic features and significant changes in grade. While the horizontal density of points is generally quite high (say average point spacing of 1 to 2 m depending on flying height and sampling frequency), features such as narrow banks/levees or channels will only be resolved if the data are sampled on a very small grid (say 1 to 2m grid). This can result in large and unwieldy base terrain files.

There are a number of approaches that can be taken in relation to the treatment of breaklines in ALS data sets:

1) Sample the entire survey area at a fine resolution (say on a regularly spaced 1m Digital Elevation Model (DEM) grid) and manually identify important salient topographic features and hand enter into hydraulic models.

2) Use local knowledge, GIS, aerial photos, satellite imagery or historic plans to identify locations of important features and hand-digitise over the fine resolution DEM as in approach (1) and then drape values from DEM to develop 3D breakline strings.

3) Use observations as in approach (2) to determine locations of key features and then field survey them to develop 3D breakline strings.

4) Use auto-processing/filtering algorithms to extract breaklines from the raw ALS data.

Through experience in such exercises, a combination of techniques approaches (2), (3) and (4) is favoured. While approach (4) nominally provides the widest coverage and makes the most of the ALS data, these processes are somewhat unreliable as there is no rational way of testing the validity of the breaklines produced. Although requiring more manual input, it is considered that approaches (2) and (3) are able to provide more reliable estimates of levels at critical locations in the floodplain. This is due to the manual checking that occurs along the way. Long-sections from the ALS can be checked for consistency and then a sub-sample tested against field measurements as a verification process.

ALS data capture yields a number of points from various surfaces (ground, vegetation, buildings etc). Thorough processing of the raw data is required to remove non-ground points effectively and to provide a true representation of the ground surface.

Raw ALS data files containing all returns (ground and non-ground) can be very large leading to difficulty with data storage and processing. “Thinned” ALS data have been processed to remove the points that provide limited additional definition of terrain. Figure 5-6 shows the form of the processed thinned ALS dataset. This figure demonstrates the following specifically in relation to ALS:

- The measured spot heights are gridded. This grid has been created by “thinning” the raw ALS point cloud dataset. The grid can be of any selected dimension. The smaller the grid, the larger and perhaps more unwieldy the dataset, but the topography is of higher definition. Conversely, the larger the grid, the smaller the dataset but the topography is of lower definition.
- There are no breaklines. Breaklines can be created according to the approaches discussed previously but they are not typically determined from the ALS data alone.
- Blank areas in which there are no measured points. In these areas, the ground surface is not visible due to heavy vegetation or there is a water surface. These points are removed as “non-ground” points. There are times when a true non-ground elevation point (e.g. a point that has hit the canopy of trees) is not removed correctly by the post-processing algorithms. The spot elevation remains in the dataset and is treated as a ground elevation. In some

cases, these types of spot heights are obvious and can be removed manually. In other cases, they may not be obvious and form part of the DEM (Digital Elevation Model – refer to Section 5.3.1.6) used in the model. This may have significant impacts on the operation and reliability of the model, and calibration may be affected.

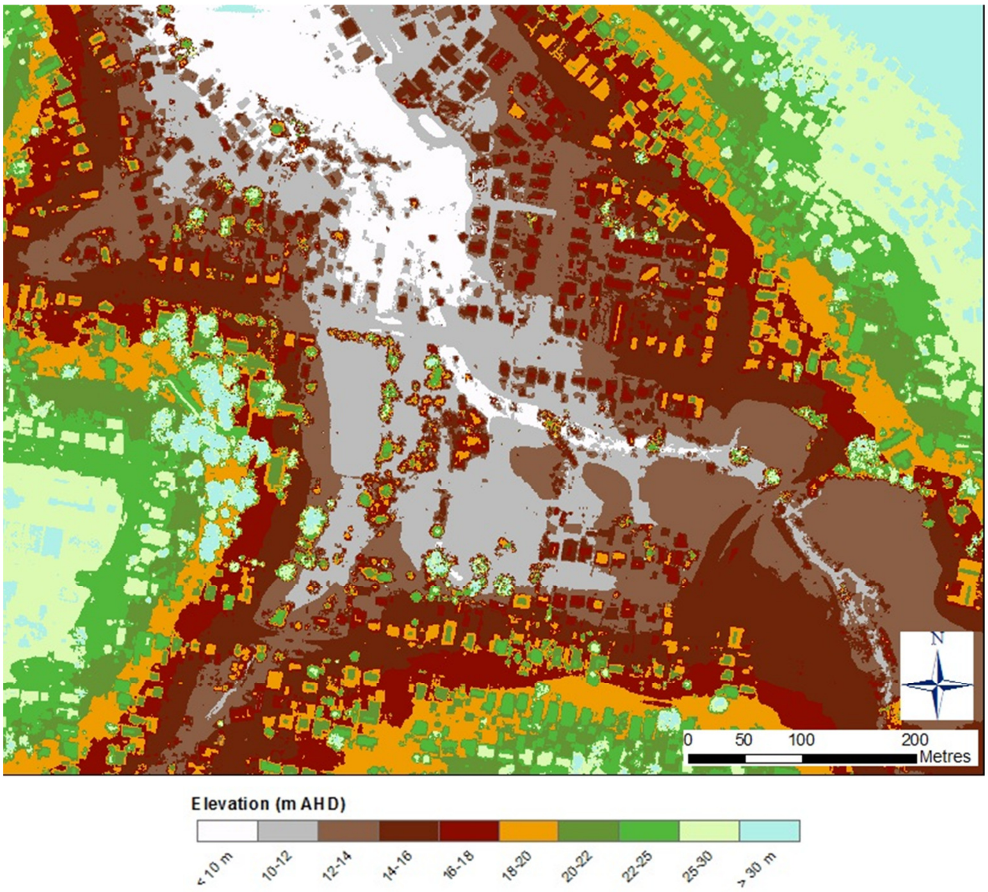


Figure 5-6 Sample of Processed ALS Dataset (Region A)

Aerial survey quality checks

As discussed in Section 5.2.4, which provides an essential background on accuracy, all survey data have a stated level of accuracy. The surveyor that has collected and processed the data typically advises accuracy of each of the survey data types. Aerial survey is no different. As discussed in the previous two sections, accuracy is dependent upon a number of factors, including flying height and the number of control points. The aerial surveyor is generally advised that certain accuracy is required before the data collection commences and the surveyor then determines the factors required to achieve the desired accuracy.

The aerial surveyor will check the accuracy of the collected data through a theoretical statistical analysis with some on-ground validation. Accuracy stated for aerial survey typically does not apply to the resulting DTM but rather to the individual elevation points collected/determined. This is an important distinction if the modeller or data client wish to independently check the accuracy of the aerial survey data.

There are a number of methods that may be used to check the accuracy of the aerial survey.

1) POINT vs DTM SURFACE: Independent field surveys of selected quality check points can be compared to the DTM. However, this surface has been interpolated from the aerial elevation data and, unless the accuracy stated by the aerial surveyor referred to the DEM, the correlation in accuracy values is not guaranteed. But, this method can provide a preliminary indication of accuracy, particularly if the independent field survey points with known accuracy are already available from another source.

2) POINT vs POINT: Independent field surveys of selected quality check points can be compared to the individual aerial data elevation points. The selected check points must exactly match the coordinates of the aerial data points to ensure that a valid comparison is being made. To do this, the aerial survey must first be received in order to select the points at which comparison will be made, which may slow the data collection phase. Alternatively, early liaison with the aerial surveyor may allow the location of a number of points to be known before data provision, which may save time.

3) STRING vs DTM SURFACE: A more appropriate approach may be to field survey a number of breakline strings along key linear features. These strings may be some 50 m – 200 m in length with points at regular spacing (say every 5 -15 m). A comparison can then be made of the profiles along the feature determined from both the aerial and field survey. In addition, if the strings are across conveyance paths (i.e. they are cross-sections or partial cross-sections), the modeller can check that conveyance cross-sectional areas are adequately represented. These comparisons enable a qualitative assessment of aerial survey accuracy for a given region within the study area. A number of surveyed strings may be required across the study area to gain an overall appreciation of accuracy.

5.3.1.6. Digital Terrain Models (DTM)

The topographic survey elements described in the preceding sections can be merged to create a Digital Terrain Model (DTM). While the topic of model topography and resolution is discussed in more detail in Chapter 6, it is worthwhile repeating some of the key elements of a DTM for model use here.

A DTM is an unstructured network of elements of variable size and sometimes shape. The variable nature of elements in an unstructured mesh allows for greater flexibility in the surface definition from the perspective that more elements at a finer resolution can be defined in parts of a terrain where more detail is required e.g. around an embankment. Each element in an unstructured grid is defined by 3 or more vertices, each vertex defined by x,y,z coordinates. Unstructured grid elements can be defined by anything from 3 nodes (basic triangle corners only) to 9 nodes (quadrilateral with corners, mid-side and centroid nodes). The multiple grid coordinates defining the extent and orientation of each element also define the slope of each element. An example of an unstructured grid is presented in Figure 5-7.

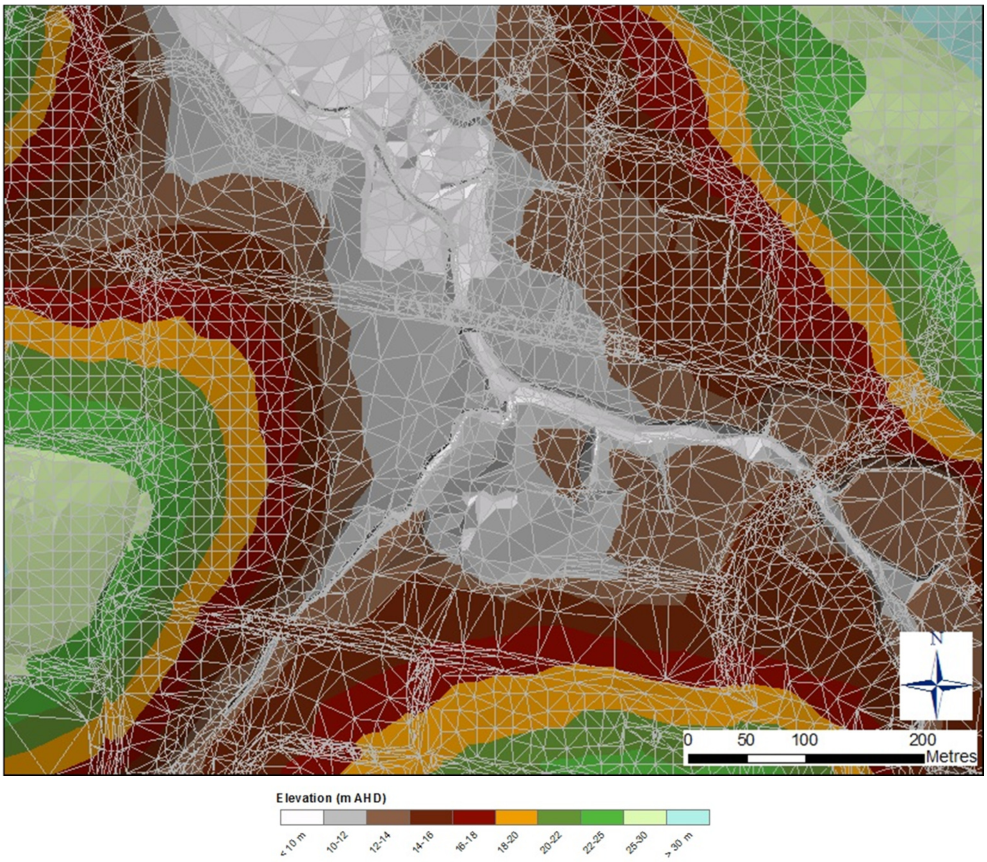


Figure 5-7 Example of an Unstructured DTM Grid (Region A)

This definition of a DTM is in contrast to a Digital Elevation Model (DEM), which represents a topographic surface through a series of three-dimensional coordinate values on a raster or regular grid. In this sense, a DEM is a simplification of a DTM.

A DEM (structured) grid is a raster surface defined by elements of regular shape at regular or semi-regular intervals as shown in Figure 5-8. Each DEM grid “square” is defined in space by cartesian (x-coordinate, y-coordinate) coordinates from the grid origin with the vertical attribute (z-coordinate) of each grid square an averaged value applicable over the area defined by the grid size (e.g. 50m by 50m). By definition, structured DEM grids require sampling of an averaged (weighted) z-coordinate. A range of methodologies (e.g. nearest neighbour, distance weighted etc.) are available for sampling the z-coordinate for each grid square.

The level of detail able to be represented in a structured DEM grid is dependent on the DEM grid resolution. For example, a rapidly varying terrain with fine scale details like incised, meandering gullies requires a fine DEM grid resolution to represent the terrain details as shown in Figure 5-8.

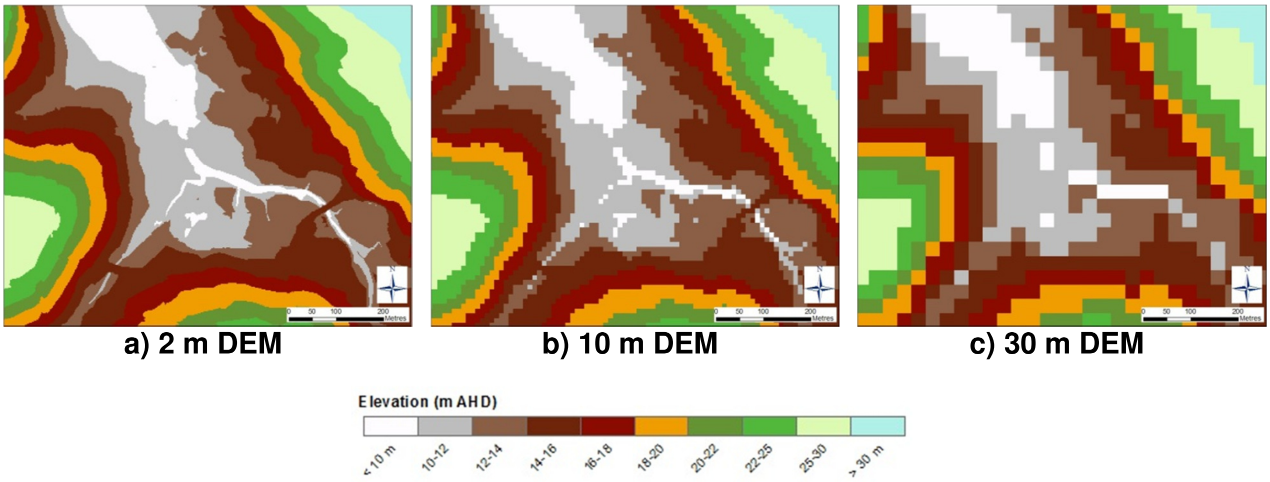


Figure 5-8 Example of Terrain Detail at Various DEM Grid Resolutions (Region A)

The physical representation of the floodplain characteristics as a DTM (or DEM) is paramount to a 2D model determining accurate and reliable outputs. In fact, the development of DTM technology has assisted the implementation of 2D hydraulic modelling.

The DTM as applied in a hydraulic model needs to be representative of the floodplain surface at the chosen model resolution. It needs to contain all potential physical influences on floodplain flow including flow controls like elevated embankments, bridge/culvert openings, and other floodplain flow obstacles if a model is to provide reliable and accurate representation of flood behaviour.

Typically, a DTM used in a model, irrespective of whether it is a fixed grid or flexible mesh model, will be a rationalised sub-set of the total ground surface data set available. Often the methodology and techniques for thinning and rationalising DTM data sets are left to the modelling practitioner to determine. The domain, format and resolution of data can be determined either:

- prior to receiving the data by setting the requirements of the data set through a survey brief; or
- post receiving the data sets by using a range of computer algorithms or more manual techniques to rationalise the data.

Both these approaches to data rationalisation have merit. The option of setting a tight survey brief has the advantage of requiring the surveyor to thin what is often a very large set into a much more manageable data set for a model, thereby allowing the modeller to start simulating and producing flood results in a more timely manner. The risk with this approach is that the survey brief may be deficient in one or more aspects (e.g. a particularly important hydraulic feature may be poorly represented or missed altogether in the supplied data set). From this point of view, the risk of obtaining a flawed or limited data set is real, suggesting that this approach is for the more experienced modeller. The second approach holds less risk of a limited data set, but requires that the modeller, or an associate, have a sound understanding and the requisite software to process the raw data into a state and format suitable for input to the model.

If time and budget constraints allow, the most sound approach is a combination of both these methods. That is, obtain all “raw” survey data and a thinned data set to specifications.

When specifying survey requirements, a sound approach is to specify point or perhaps contour data in areas of the study domain where ground surface slopes are constant or changing slowly and vector “breaklines” where rapid changes in terrain occur. The definition of breaklines to define floodplain features allows for accurate representation of floodplain features without opportunity for misinterpretation. Breaklines can be either 2D (i.e. vector lines without an elevation component) or 3D (i.e. vector lines with an elevation assigned at each vertex, that can be interpolated between vertices). Typically, breaklines are used to define continuous features on a floodplain such as embankments or channels. In cases such as these, a sequence of breaklines is used to define the feature. For example, around four to five breaklines can be used to define an embankment: bottom of bank on both sides; top of bank on both sides; and bank crest. Breaklines are able to be used automatically in many gridding / triangulating routines and therefore avoid the instance of features being interpolated in a discontinuous manner as is illustrated for the creek channel in Figure 5-9.

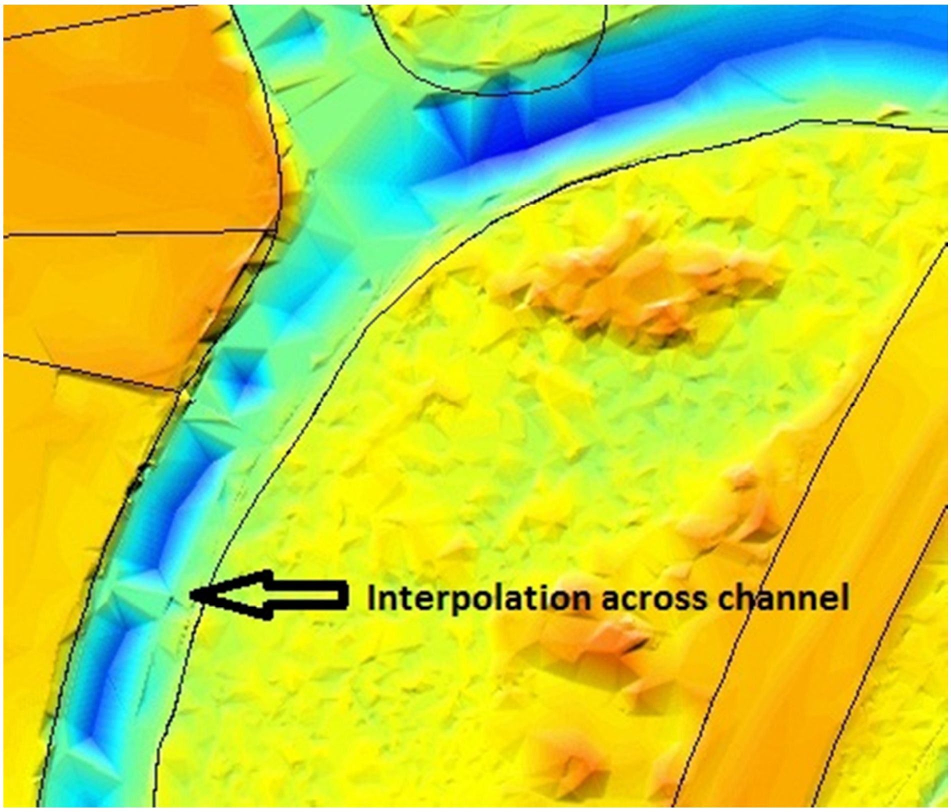


Figure 5-9 DTM Without Breaklines

In Figure 5-8 the black lines indicate the cadastre boundaries and the coloured scale the DTM surface. The automated routines used to “triangulate” the DTM surface have, in some instances, created surface elements that span the creek channel as highlighted in the figure. The occurrence of these artefacts of automated routines can be avoided by using the techniques described in the forthcoming section on creating a DTM.

While a flood study is primarily focussed on flood flow behaviour in overbank areas, the distribution of flow between in-bank and overbank areas is important. The representation of flow conveyance within flow channels needs to be adequately represented by survey, preferably of a comparable accuracy and resolution as the overbank areas. As topographic data collection is a significant component within any flow/flood investigation, separate scoping and commissioning of this component is considered worthwhile.

Creating a DTM

It is often necessary to merge two or more topographical data sets together in order to develop a DTM that provides full coverage of the study area. A common example of the requirement for merging topographic datasets is where an overbank floodplain topography needs to be joined with a creek channel survey from a second data source. This situation commonly occurs where permanent water, heavy vegetation or difficult terrain in the channel alignment requires a specialised survey data set to supplement a broader survey of the easier to access overbank areas.

Merging of data sets is often a complicated exercise made more difficult by adjacent topographies having variable accuracy and spatial resolution. Data set merging is best managed if a focussed supplementary survey can be planned and specified after careful review of the data gaps required to fulfil the primary data set. In this case, the supplementary data can be specified to completely cover all required areas without the need for extensive extrapolation or interpolation of data with data overlaps managed in an ordered manner.
If the two or more data sets have been derived without such coordination, then the issues are often twofold:

i) Which method to use to fill gaps where there are no data specified?; and

ii) Which data set to adopt / give more weight to where the data sets overlap?

There are numerous numerical methods available to address these issues that are handled best by CAD, surveying and GIS software packages with 3D surface capabilities. Each of the numerical methods has their merits depending on the size and scope of the area to be merged. Whatever the method eventually adopted, it is highly recommended to compare the final, merged topography with the original measured data. 3D visualisation software can also assist in checking that the general topography reflects the digital dataset. While the merged dataset may be a compromise between the original data sets by necessity, the resulting merged topography should be physically realistic and reflect the observed shape of the topography on site. Additional, manual editing to fine-tune the automated outputs may be required to ensure a realistic result.

5.3.1.7. Aerial Photographs

Aerial photographs are an important source of qualitative data and can be collected during an aerial survey. Geo-referenced (or ortho-rectified) aerial photos, that is, aerial photos which have been registered with coordinates are often supplied as part of a photogrammetric survey, but typically are an additional extra to other types of survey including ALS.

It is important to recognise that the raw aerial photograph is spatially distorted, being a planar image of a curved surface of variable height. In orthorectifying the image, the image is scaled, rotated and stretched so that various reference locations move to their correct coordinate locations. Thus, the location of features on an aerial photograph may have a degree of uncertainty. For example, if the rectified aerial photo is to be used to locate a flood mark, the attributed location is itself subject to a tolerance. In an area of high flood gradient this can result in differences between observed and simulated flood levels that do not accurately represent the true differences.

Aerial photos, while not providing quantitative data directly, can provide additional information about flowpaths, flow obstacles and floodplain vegetation that is not always immediately evident or accessible on a site inspection. Aerial photos can be a useful guide when defining floodplain roughness coefficients and can be used directly to develop a spatial map of floodplain roughness. Aerial photography can also be used to digitise building outlines or fence lines where these are to be included within the model grid and can be a reasonable source of information for imperviousness mapping. Aerial photography is also available in historical data sets making it a useful reference for indexing catchment development or sourcing information on floodplain development for historical calibration events.

Where other topographic mapping is not available, aerial photography also provides a useful background for referencing model predicted flood inundation.

5.3.2. Historical Topography & Infrastructure

All data collection methods covered thus far in this chapter are concerned with measuring present day data: what the catchment is like now. However, calibration floods are historical events and the catchment and floodplain conditions at the historical point in time at which the event occurred may not be the same as present day. In addition, if a number of calibration events are to be simulated, changes to conditions may also have occurred between events. Conditions at *each* of the relevant historical points in time must be established and used in the model. Changes to conditions that may affect flood behaviour include dam construction, changes to initial dam storage levels, dredging or siltation of river entrances, levee construction, road raising or duplication, new road embankments, new culverts or bridges, upgraded drainage networks, development on the floodplain, different crop types or growth stage, and so on. The modeller will need to be vigilant for these changes. Depending upon the length of time since the occurrence of the calibration event, record of these changes may be only available anecdotally.

This consideration is especially important if major historical events are used for calibration. For example, there may be some anecdotal or even good formal measurement evidence of a record flood that occurred 100 years ago. Because of the size of this event, it may be essential that it should be included in the calibration, but use of the data must proceed with considerable care.

For example, the 1918 flood in Mackay or the 1893 flood in Brisbane were both extreme events where considerable data are available for the flood. However, they were sufficiently long ago that a range of data, such as the status of dredging in the river for example, was poorly recorded. On the other hand, the 1998 flood in Katherine was sufficiently recent that there is good information available and the river and floodplain conditions are well recorded and understood.

While it is impossible to overcome all of the possible problems with the data from historical events, care must be taken in the application of the observed data.

5.3.3. Hydraulic Roughness

Flow and flood behaviour is influenced by the hydraulic resistance due to channel and floodplain vegetation and landform. This resistance (essentially an energy or head loss) is quantified in hydraulic modelling by hydraulic roughness parameters. Various forms of hydraulic roughness have been developed and include Manning's 'n' and Chezy coefficient 'C'.

Appropriate values of the hydraulic roughness for given vegetation and land use can be evaluated via:

- Aerial or satellite imagery
- Site inspections
- Textbooks and guidelines

Useful references for hydraulic roughness values for Australian streams and floodplains are provided in Land and Water Australia, 2009 and Arcement and Schneider (no date). Care should be exercised when interpreting published roughness values from these references for a numerical modelling application, especially when applied to a 2D model. The roughness values in these texts and similar are generally derived from in-channel flows and based on a 1D interpretation of those flows. As such, they are not directly applicable to a 2D model and should be regarded as an indicative starting point for model calibration.

Through the model calibration process, hydraulic roughness may be adjusted to optimise the simulation of historical flood behaviour. The adopted hydraulic roughness values should be checked, where possible, against published and/or accepted values for the given vegetation and land use. Where calibrated hydraulic roughness values vary significantly from published or expected values, significant care should be taken. Typically, such aberrations often suggest there is a problem with one or more of the following:

- Incorrect hydrologic boundary conditions either in terms of flow (upstream) or level (downstream)
- Problem with model schematisation (incorrect representation of physical geometry of river/floodplain)
- Errors in, or interpretation of, calibration data (e.g. wrong datum).

In some hydraulic modelling applications, the seasonal variation of hydraulic roughness with vegetation is important. This effect can be difficult to quantify given the lack of flood information at this temporal resolution. The influence of hydraulic roughness at site, reach and catchment scale on flood behaviour is further discussed by Rutherford *et al.* (2007). This reference provides a useful insight to the effect of waterway and catchment re-vegetation on flood behaviour.

Similarly, the effect of vegetation on the passage of flow along a channel or overland flowpath may vary with the depth and velocity of flow during a flood event. For example, a channel with dense reeds growing in it may have a high roughness at low flows when the reeds are standing tall, but at a certain point the combined depth and velocity of the flows may cause the reeds to bend over and lie down in the creek bed with a corresponding dramatic reduction in hydraulic roughness.

It is also important to note that hydraulic roughness is often calculated using a Manning's equation approximation, for example. In this approximation, the basic assumptions are that the flow is relatively deep, the longitudinal water surface slope gently sloping so that the flow remains in the sub-critical range. In very shallow flows, such as those encountered in direct rainfall applications (see Chapter 11), these assumptions are likely voided and the Manning's roughness coefficients that are representative of certain vegetation types at deeper flows invalid.

5.4. **Model Calibration Data**

5.4.1. Historical Hydrologic Data

Numerical hydraulic modelling requires the specification of inflows (peak flow and flood hydrographs) to the study area. Historical hydrologic information is utilised in the model calibration/verification phase. The derivation of the historical inflows requires the use of the following hydrologic data:

- Historical rainfall (Daily and pluviographic)
- Historical streamflow (Daily and instantaneous)

The quantity and quality of this historical information has a primary influence on the model reliability. Poor quality and/or limited quantity of data will reduce the confidence in model predictions. Ideally, historical streamflow data from several flood events of varying flood magnitudes would be available upstream of the study area. However, the more likely situation is that no, or only estimated, historical streamflow is available at the upstream study limit.

The availability of historical hydrologic data for model calibration/verification must be understood during the scoping and conceptualisation of a hydraulic modelling investigation. Such understanding will enable the formation of reasonable and achievable expectations of hydraulic model reliability. Streamflow data can be in one of two forms:

1. Calibration Event Gauging - Gauging of the calibration event has been undertaken at a location relevant to the study area such that measured flows can be incorporated into the calibration process. Further details are provided in Section 5.4.1.1.

2. Derived Flow Hydrograph – Flows can be derived at a water level gauge location using a stage-discharge (rating) curve. An understanding of the quality of streamflow data can be gained from the examination of rating tables and gaugings developed for a gauge. The reliability of the streamflow data for a large flood event is limited by the amount of extrapolation required beyond the highest gauging of the rating curve. Further details are provided in Section 5.4.1.2.

5.4.1.1. Calibration Event Gauging

In the unlikely event that there is one or more actual flow gaugings obtained during a calibration event, it is important to recognise that it is not really a certain flow at a particular point in space or time.

The gauged flow is calculated across the measured flow width. One or more velocity measurements are taken (depending on the measured depth) at one or more points across the flow width (depending on the magnitude of the width). The depths are combined with the distances between them to define unit cross-sectional areas, which are multiplied by the velocity measurements to produce unit discharges. The sum of the unit discharges add up to the cross-sectional discharge. There are therefore errors and uncertainties in all of the measurements and the calculations.

In addition, it takes time to take each depth sounding and then each velocity measurement and to then move to the next measurement point. Gauging can therefore span a significant time interval (relative to the model timesteps) as several measurements are taken across the flow width. The calculated discharge did not therefore occur at a unique point in time or model timestep.

The measured flow width will also probably span several model nodes and is unlikely to correspond exactly with the model resolution. So that neither the unit nor the cumulative cross-sections forming the gauging will correspond to unique nodes or grid/mesh locations in the model. The same goes for the location of the velocity measurements.

There is therefore no guarantee that the nodes (or points in time) at which model calculations are made will be at the same location (or time) that field measurements are made. The modeller will need to decide which model outputs are closest in time and space to which field observations. As both field observations and model calculations contain uncertainties, which are unlikely to be the same in both data sets, an exact match would suggest that the model produced the “correct” value for the “wrong” reasons.

5.4.1.2. Derived Flow Hydrograph

It is important to note that flow/discharge hydrographs (streamflow data) are not *recorded* flows but rather *derived* flows. A stream gauge records water level (stage), not flow. Flow is derived from the water level records using a rating curve (stage-discharge relationship). A stage discharge relationship is specific to each gauge site and is typically determined by physically gauging (rating) the site during a flow event. That is, as the flood rises and falls, measurements of flow and stage are taken. A description of a physical gauging process is provided in Section 5.4.1.1.

Stage-discharge (rating) curves may be constructed from one or more physical ratings of a gauge site during one or more events. Rating curves should be updated by the responsible government agency if changes have occurred to the flow behaviour at the gauge site (e.g. construction of a bridge downstream).

The event/s for which the rating was undertaken may not be the largest event on record. The rating curve is physically accurate only up to the maximum rated water level. Rating curves are usually extrapolated beyond the maximum rated water level using engineering judgement of gauge site topography/cross-section and water surface slope. This is not a reliable process and may introduce substantial error into flow predictions beyond the limit of gauging. The modeller should note whether the flood level for the calibration event/s fall into the extrapolated area of these curves. This will provide an indication of the likely accuracy of the derived flow.

Some rating curves are well-documented in terms of their history and construction, others are not. Given the importance of these rating curves to each hydraulic modelling study, rating curve information should always be sought as a high priority and used with appropriate caution.

5.4.2. Historical Flood Behaviour

Historical flood information is particularly important as it can often provide a significant improvement in the quality and reliability of the study outcomes. While data on historical floods may be difficult to obtain at times, efforts expended in finding and analysing these data are extremely valuable.

Typically, information on the historical flood behaviour is collected at the commencement of the study. Sometimes the responsible agency has already collected some of the quantitative information before the study commences in order to estimate the likely number of calibration events to be modelled. Regardless, there is no doubt that the quality of the modelling will improve by including a formal data collection process within the study scope. It is recommended that the data collection process involve significant community and stakeholder consultation. In undertaking the data collection, there are many types of data that may be found and these are discussed in the following sections.

5.4.2.1. Continuous Water Level Recorders:

Historical records for continuous water level recorders are typically stored by government agencies. These records are very important as, if intact, they will show the complete hydrograph (i.e. the rise, peak and fall of the flood). Care needs to be taken with the datum of these records as they are typically not to AHD (Australian Height Datum) and some conversion may be required. In large events, these recorders can fail and the data need to be inspected for "flat" areas, which may indicate lack of data due to failure of the gauge. Typically, each stage record has an accuracy code assigned and these should be noted before use. As explained in Section 5.4.1.2, flow is derived from stage using a rating curve and it is critical that the maximum gauged flow is known so that the extent of extrapolation underlying the 'recorded' flow is clear.

5.4.2.2. Maximum Height Gauges

Maximum height gauges simply record the peak flood level reached during a particular event. Again, data are held by a government agency and care should be taken converting the gauge datum to AHD. Failure of these gauges is difficult to detect as they are simply recording the peak level. If the gauge fails before the peak of an event, it may still provide a "peak level" value but it will be the flood level reached prior to the peak at the time of gauge failure. Time of peak is sometimes also available and it is recommended that this is checked to ensure that the peak was recorded at approximately the correct time.

5.4.2.3. Peak Level Records

If the flood event has been of a significant nature, it is likely that government agencies have been able to collect some actual flood levels at a variety of locations. This is typically done by mobilising agency staff to place markers (paint, stakes, nails, surveyor's tape) either indicating maximum flood extent (e.g. spray paint on a road, stake in ground) or peak flood level (e.g. nail in a tree). Ideally, each marker should have the time and staff member's name recorded at the marker site as well. Following the event, surveyors can measure x,y location data and z flood level data at each of these markers. It is also useful to photograph the site and to record ground level. Surveyors should also include the time at which the markers were placed, by whom and type (e.g. nail in tree) in their metadata. Residents often also record peak flood levels, particularly if the flood has inundated any buildings on their property. An assessment as to the reliability of these levels can only be made after viewing the marks themselves and noting the care with which the recording has been made. Have different event dates been recorded by the resident or is the resident relying on memory to determine one event from another? Has the location of the marks changed in any way since the record was made? For example, if the marks are made near the front door, has the house been raised at any time since? Detailed discussion with the resident can often unearth important details otherwise unknown.

5.4.2.4. Debris Marks

Debris marks are best measured as soon as possible after the event, when the debris or scum line is still fresh. This ensures that the mark is attributable to the event of interest. The longer it takes the surveyor to measure the mark, the more chance there is for errors as debris marks may be from an older event or an event in a tributary rather than the main channel of interest. Debris marks can be inaccurate for a number of other reasons. They can be influenced by dynamic hydraulic effects such as waves, eddies, pressure surges, bores or transient effects, which may not be accounted for in the model. For example, if the debris mark is located within a region of fast flowing floodwater it is possible that the floodwater has pushed the debris up against an obstacle, lodging it at a higher level than the surrounding flood level. More common though is the fact that debris often lodges at a level lower than the peak flood level. The reason for this is that for debris to be deposited it needs to have somewhere to lodge and this elevation is not always at the peak flood level. For example, the classic place for debris lodgement is a barbwire fence with horizontal strands of wire. If the flood level almost reached the top strand of barbwire, debris will not lodge in the top strand but rather on the second from top strand, which may be about 0.3m lower than the peak flood level. It is recommended that the surveyor be asked to record as much information as possible about the mark itself (e.g. debris on barbwire fence spacing 0.35m) so the modeller is able to consider reasons for discrepancies in the calibration process, if they arise.

5.4.2.5. Watermarks on Structures

As with debris marks these are best measured as soon as possible after the event to ensure that they are attributable to the event of interest. If this has not occurred, care needs to be taken as to whether the structure has remained the same since the watermark was made. For example, a mark on the side of a house that has subsequently been raised will have no value. Detailed discussions with stakeholders are needed when considering watermarks that have not been recently made.

5.4.2.6. Anecdotal Information

*Anecdotal in*formation is usually qualitative in nature but can be very valuable in determining flow behaviour and subsequently verifying that the model behaves in a similar manner. Photograph and video evidence can also be beneficial in this regard and can often assist long-term residents remember details of historical floods long past. The modeller will need to be mindful of the fact that memories can sometimes fade or be skewed by other events that have occurred between. In addition, information providers may not be able to provide unbiased information due to a vested interest (e.g. pride or financial gain etc) in the level to which an historic event reached. Again, detailed discussions with residents and stakeholders can provide the modeller with a general feel for the reliability of all anecdotal evidence. Inconsistent facts have to be identified and discarded and discrepancies have to be studied and explained.

With all historical flood behaviour data, the modeller must consider the nature of the collected data when making an assessment of the reliability of the point and the priority it has during the calibration process.

5.4.3. Sources of Historical Data

Some data sources have been included in the previous sections. However, the following is a stand-alone list of potential sources of historical hydrological and flood behaviour information:

- Previous studies and reports;
- Consultation with, and/or survey, of residents, property owners and local stakeholders;
- Council representatives including members of floodplain risk management committees;
- Council records including flooding complaints databases;
- Local and/or National newspapers;
- Historical records, societies and libraries;
- Bureau of Meteorology;
- State Water Agencies;
- State Emergency Services;
- Other Federal and/or State Government Departments;
- Water Corporations responsible for trunk drainage assets; and/or
- Road and/or rail authorities.

5.5. Model Application Data

5.5.1. Property data

In order to assess flood risk and hazard, property data (including building type, condition and floor level) are typically required. As well, agricultural development is important in many rural areas, and road and rail closures affect the damages costs of disruption to transport.

Property databases form the basis of most flood damage assessments. These databases typically require a description of the property attributes and features on a property by property basis. Typical information required for each residential property includes:

- Street address;
- Representative ground level;
- Floor levels;
- Building construction type (e.g. brick veneer, timber, slab on ground, on piers etc.);
- Building age;
- Single/double storey;
- House size.

Commercial and industrial properties require similar information, but also require information on the type of business undertaken at the site as this can have a significant bearing on the value of flood damages from business to business.

Ideally, this data are collected via field survey. However, it can be a costly process depending upon the number of properties for which data are required. Alternatively, there may sometimes be records available from the local authority, other government agency or the census. For broad assessments, property data may be estimated. A panel of people with relevant skills should review the method of estimation for soundness. As an example, property data may be estimated from aerial photography or from a general understanding of local conditions.

An example of a typical property data collection form is provided in Figure 5-10.

5.6. **References**

Arcement, G.J. and. Schneider, V.R. ‘Guide for Selecting Manning's Roughness Coefficients for Natural Channels and Flood Plains’. *United States Geological Survey, Water-supply Paper 2339, Metric Version.*

Bishop, W. A. and Catalano, C. L., 2001. ‘Benefits of Two-dimensional Modelling for Urban Flood Projects’, *Proceedings of Hydraulics in Civil Engineering*, *Engineers Australia, Hobart, 2001.*

Rutherfurd I., Anderson B. and Ladson A., 2007. ‘Managing the effects of riparian vegetation on flooding’. In Lovett S & Price P (eds), *Principles for Riparian Lands Management*. Land and Water Australia, Canberra.

Stewardson M.J. and Howes E.A., 2002. The number of channel cross-sections required for representing longitudinal hydraulic variability of stream reaches. *Proceedings of the 27th Hydrology and Water Resources Symposium. Engineers Australia p21-25*

Land and Water Australia, 2009. ‘An Australian Handbook of Stream Roughness Coefficients’. Product code PN30109. Land and Water Australia, Canberra.

Address	Land Use	Building Type	Building Age	Building Size	Construction Type	No. of Storeys	Ground Level	Garage Level	Habitable Floor Level	Split Level	Step Up	Flood Level	Comment
Example													
31 West St, Southville	Commercial	Shops	Medium	3,000m^2	Block	1	12.1	12.1	15.5	N	N	16.2 in 1984	Northside Shopping Centre
43 North St, Eastern Hills	Residential	Detached	New	Medium	Brick Veneer	2	15.5	15.5	16.1	Y	2.5	None	
78 North St, Eastern Hills	Residential	Units 6	Old	Large	Brick Veneer	2	15	15.2	17.5	Y	2.5	None	
Notes: All Levels to AHD (m) Record Details as specified in each column.	Residential	Detached	Old	**Residential:** Small	Brick / Block		At Entry	If lower than habitable floor	Lowest Level of habitable floor (does not include laundry or garage)	Are the habitable floor levels split over 2 levels? Y or N	Difference in floor levels if split	Any recorded levels at site? Specify level & year of event	
	Commercial (Specify)	Semi-detached	Medium	Medium	Brick Veneer								
	Industrial (Specify)	Flats/Units - specify #No.	New	Large	Weatherboard								
	Vacant	Other		**Commercial/ Industrial:**	Fibre Cement								
	Other			Record Area (m^2)	Other (Specify)								

Figure 5-10 Example of Building Survey Form

6. CHAPTER 6 - MODEL SCHEMATISATION

6.1. Introduction

Schematisation of a numerical hydraulic model is the process of converting the conceptual model of the physical system (refer to Chapter 2) into a numerical model. The process involves discretising the continuous physical system into a series of discrete elements. Schematisation is the process that characterises the profession of numerical modelling. The physical system being modelled may be schematised in many different ways depending on the selection of model elements within the modelling tool and the choices made by the modeller. The accuracy, reliability and usefulness of the model are significantly influenced by the skill of the modeller in completing this process.

The choice of schematisation is dependent on a range of considerations. The primary considerations that apply are the:

- type of model to apply;
- model extent;
- mesh or grid resolutions and orientation (spatial schematisation);
- simulation timesteps (temporal schematisation);
- specification of specific hydraulic features and controls; and
- types, location and design of boundary conditions.

This chapter provides an overview of each of these considerations and provides guidance on some of the critical issues that need to be addressed. In practice, it is necessary for the modeller to ensure that all objectives for the model can be met using the schematisation chosen.

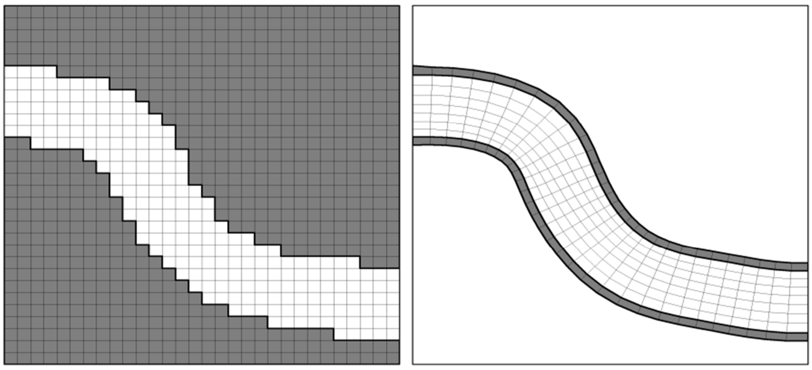
6.2. Selection of Model Type

The selection of the type of model is the first decision in the model schematisation process. A detailed description of the model types is provided in Chapter 2.The choice of model for practical applications is generally limited to two fundamental techniques. These techniques involve the discretisation of the physical system using a fixed regular grid or as a flexible mesh. The flexible mesh approach generally utilises triangular and quadrilateral elements either exclusively throughout the model or in combination. The fixed grid is generally based on a fixed dimension rectilinear grid with constant grid spacing (i.e, all of the elements are of the same dimension and shape). The choice of model type is dependent on the specific circumstances, and general guidance on the options is provided in the following sections.

6.2.1. Fixed Grids

The fixed grid approach is usually based on a finite difference solution to the governing equations. This approach requires that a grid of same size and shaped elements is developed as illustrated in Figure 6-1. A variant on the rectilinear grid is the “boundary-fitted coordinate” or “finite difference curvilinear” method. Generally referred to as a “curvilinear” model, this method allows the grid to be slightly distorted so as to align the elements with the streamlines or flow paths of importance.

The solution schemes for fixed grid models are numerically more efficient to solve than their flexible mesh counterparts, especially those schemes that are implicit and utilise matrix solvers. This is due to the “well-organised” nature of the coefficient matrix that must be solved mathematically at each time step (refer Chapter 3). Generally, the fixed grid models are between 4 and 8 times faster to solve than flexible mesh models where the same number of elements exist in each type of model. In some cases, flexible mesh models have the ability to reduce the number of elements (and thereby reduce run times) by utilising larger elements in areas away from the area of interest or where larger elements adequately depict the hydraulic behaviour. The trade off between resolution and model efficiency must always be considered when choosing a model type.



a) Regular Grid Mesh b) Curvilinear Grid Mesh

Figure 6-1 Regular Fixed Grid and Curvilinear Grid

6.2.1.1. Rectilinear Grids

The rectilinear fixed grid technique is particularly popular for flooding and urban drainage studies as they have historically been more stable and faster than finite element and finite volume models that use a flexible mesh. The rectilinear models are relatively simple to develop compared to the flexible mesh model as there is no requirement to manually develop a model mesh. This can save significant time and effort in the model development process. However, in some situations the rectilinear models are not suitable for resolving channel flow due to the poor representation of flowpaths by the grid. For example, Figure 6-1 shows how the channel becomes staggered where the alignment is angled to the fixed grid. This problem can be resolved by increasing the resolution with a finer grid size but it comes at the significant cost of increased computational time. Generally, the reduction of the grid cell size by a factor of 2 will lead to an 8 fold increase in the model run time. If the resolution of channel hydraulics is important objective for the model then the use of a 1D representation of the hydraulic feature can be included with linkages made to the 2D grid (see Chapter 9).

The number of grid cells chosen to represent a channel is dependent upon the objectives of the modelling exercise and the importance of the channel’s conveyance in the conceptual model. A minor or insignificant channel may not require accurate representation and thus be modelled as a simplified channel with fewer grid cells. The model's ability to represent the conveyance of a channel will also be dependent upon the orientation of the grid to the direction of flow in the channel. A meandering channel through a fixed grid may require more grid cells than a relatively straight channel directly aligned with the grid orientation. Generally a minimum of five (5) grid cells across the width of a channel is required to produce an adequate representation of channel flow.

6.2.1.2. Curvilinear Grids

A curvilinear grid model solves the governing flow equations on a transformed reference system based on polar coordinates. This allows the model "grids" to curve slightly to fit within predefined flow paths as shown in Figure 6-1. The use of curvilinear grids is preferred where the flow is predominately uni-directional such as a river or drainage channel with generally unidirectional flow in floodplains. In these situations, the total number of computational points can be significantly reduced compared to a rectilinear grid whilst still maintaining the benefits of having a finite difference solution scheme. In addition, the grid is aligned to the flow path, which enhances the representation of the channel conveyance. A typical example of a curvilinear model on a river system is shown in Figure 6-2.

The curvilinear mesh has some restrictions that must be considered when selecting the model type. The first consideration is the grid dimension. The overall grid dimensions must be maintained throughout the model domain, as is the case in a rectilinear model. Therefore, the number of cells in the longitudinal direction and the transverse direction must be maintained throughout the model domain. If a large number of cells are required to the represent a large floodplain in the transverse direction in one part of the model; the same number of cells must be maintained in the transverse direction through more narrow parts of the model. This can produce a very small grid resolution size in the narrow part of the model and result in significantly large numbers of grid cells. This in turn requires a reduced model timestep, resulting in longer run times.

The second consideration is the development of the mesh. The curvilinear models require that the angle between crossing grid lines is maintained at 90 degrees for all elements and the aspect ratio (the grid cell length to width ratio) for the distorted element should not exceed five, with three being a recommended value. This geometry restriction requires greater effort from the modeller in developing the mesh.

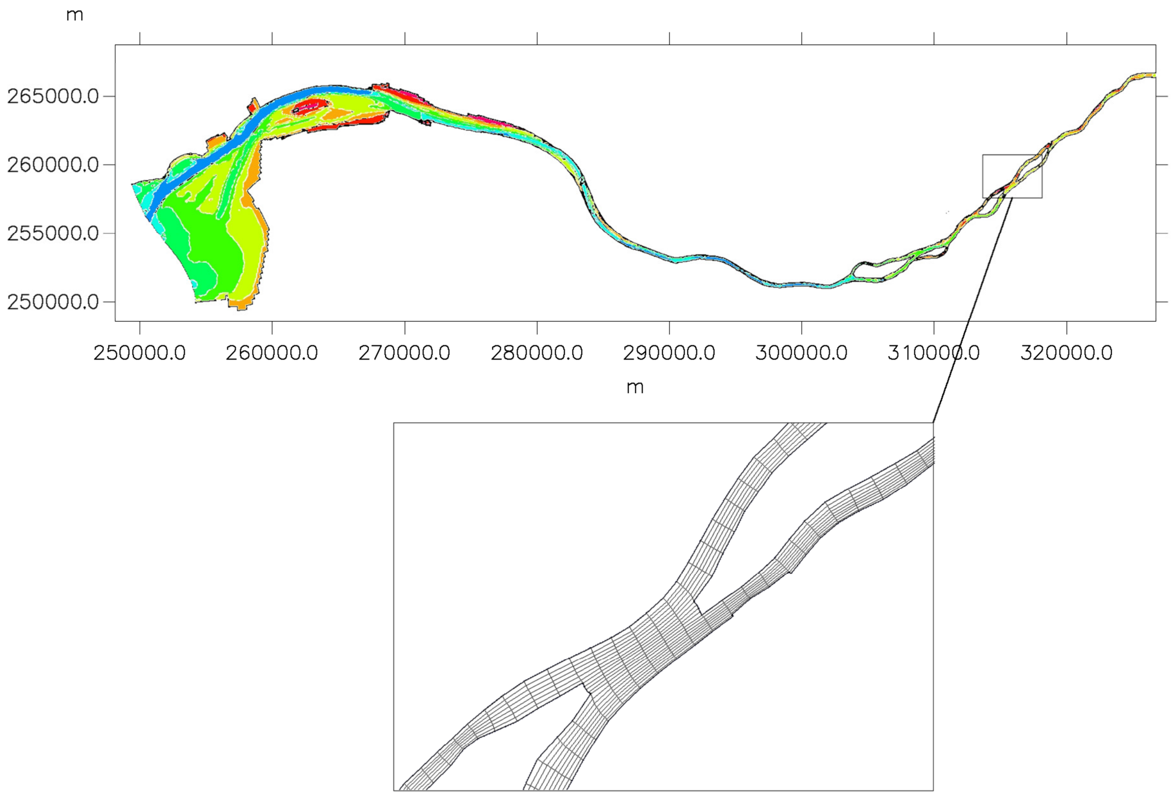


Figure 6-2 Example of the Use of a Curvilinear Grid

6.2.2. Flexible Mesh

Models that utilise a flexible mesh are becoming increasingly popular, particularly those using a finite volume solution. The flexible mesh method involves the discretisation of the model domain into triangular or quadrilateral elements with a single element type being applied exclusively or in combination with other types. The elements applied in a mesh can be of varying size (area), as shown in Figure 6-3. The flexible mesh technique provides a great amount of flexibility in the representation of a complex geometry and enables the mesh to be customised to resolve specific details such as boundary conditions, physical features and channels. Higher resolution can be provided with smaller elements in areas of interest or where there are rapid changes in flow behaviour. In other areas such as wide open floodplains, larger elements can be developed to reduce the number of computational elements and thereby reduce the model run time.

However, the numerical solution scheme used for flexible mesh models is not as efficient as that used for finite difference models. As previously stated in Section 6.2.1, the flexible mesh models are between four and eight times slower than an equivalent finite difference model with an equivalent number of computational points. Nevertheless, flexible mesh models can be an attractive alternative when the number of computational elements can be reduced to the extent that they offset the reduced numerical efficiency.

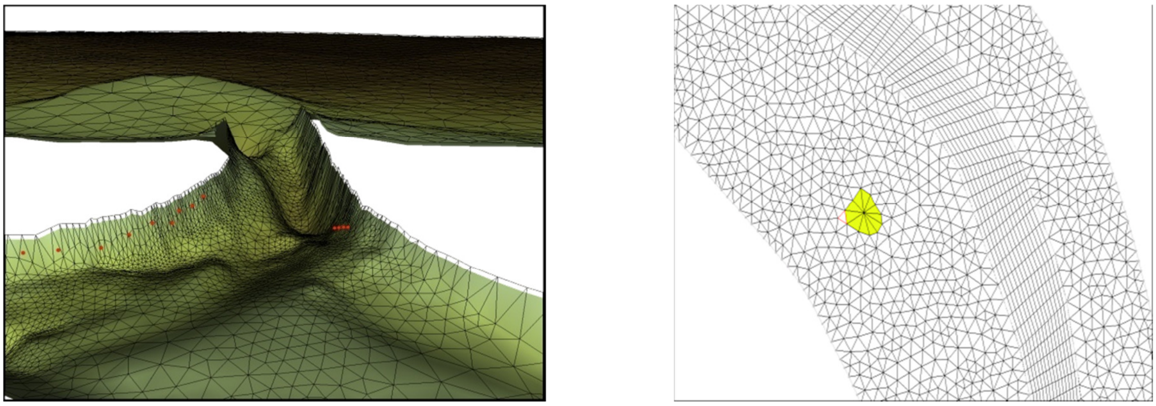


Figure 6-3 Examples of Flexible Mesh Elements

6.2.2.1. Finite Volume versus Finite Element

Flexible mesh models can be sub-divided into two further categories, which are based on the solution to the governing equations. These are described previously in Chapter 3. However, it is important to understand the practical impact of the differences between the two alternatives.

6.3. Model Extents

The primary goal in selecting a model extent is to represent hydraulic behaviour within the area of interest without significant influences driving hydraulic behaviour from areas outside the model extent. Key considerations include, but not limited to:

- Ensuring that the model extent is sufficient to cover the likely inundation extent of the largest event to be modelled. The key here is that for the largest event to be modelled (typically the Probably Maximum Flood), the model extent does not artificially restrain water movement at its boundaries, and that the topographic data within the model extent also extends beyond the inundation areas.
- Ensure that boundary conditions are located sufficiently far away so as to not unduly influence results within the area of interest.
- Minimise the inclusion of unnecessary (flood-free) areas, as this produces excessive results, impacts on computer memory requirements, increases model output file sizes and reduces efficiency.

If the likely maximum extent of the inundated area is difficult to define (e.g. very flat terrain or dam break studies) defining the extent can be an iterative procedure. A recommendation is to always start with a large model area and then narrow the model domain based on feedback of model results, as this is far less problematic than the reverse process. Using a coarser grid/ mesh resolution to reduce run times during these earlier stages of the modelling process can be an effective and efficient approach, especially for large model areas.

6.4. Grid or Mesh Resolution

The grid or mesh resolution has significant implications for the model stability, reliability and accuracy of the model. It is therefore important to carefully consider the size of the model grid, the area of mesh elements and the angle of the model grid and mesh elements to the likely flow streamlines.

6.4.1. Numerical Stability and Accuracy

A range of components selected in the model build process impact on numerical stability and accuracy. The stability of a model is best described by examining the Courant Number (Cr), also called the Courant Criteria. Cr (as utilised in 2D modelling applications) is calculated as below.

$$Cr = \frac{\left(v + \sqrt{gd}\right) \times \Delta t}{\Delta x}$$

Where v is velocity (m/s), g is gravity, d is depth, Δt is the timestep and Δx is the grid/mesh dimension.

Theoretically, it is only explicit models (refer to Chapter 3) that must satisfy the criteria that Cr shall be no greater than one (1) in order to avoid "unstable" or inaccurate results. A fully implicit scheme should be able to utilise Courant numbers that are infinitely high provided that the timestep is sufficiently small to depict the shape of the boundary timeseries. Semi-implicit schemes, such as those used for fixed grid finite difference solutions are able to run at Cr conditions greater than one but, are unlikely to produce stable and reliable results for Cr values greater than 10.

Note that Cr varies throughout the model domain, so it is the areas that experience the highest Cr value that limit the timestep. These areas are usually the deepest flow areas where the hydraulic friction from the bed of the model is at a minimum within the model domain.

However, for implicit solutions, the actual Courant number can be much smaller in practice and may approach one (1), depending on the application. The introduction of 1D elements can introduce into an implicit model a Cr limitation much smaller than for the rest of the domain.

In general, model stability and accuracy is compromised where the Cr value is too high for the numerical scheme being used. On investigation of the Courant equation, this typically occurs when the velocity, depth or timestep is too large in combination with an element size which is relatively small.

A key issue in the model development process is to gain an understanding of the timestep that is likely to be required (and hence overall run time) for a specific mesh/grid resolution. This analysis will require an estimate of the likely modelled flow depth and velocities.

Model accuracy can be compromised when a Courant Number higher than one (1) is utilised in conjunction with a bathymetry that is irregular. For Courant Numbers less than one (1), a volume of water will only move between adjacent elements in a single timestep. Depending on the wetting and drying algorithms used by the numerical scheme, for Courant numbers greater than one (e.g. 2, 3 etc) a volume of water may move across multiple elements that are non-contiguous during the period of a single timestep. If this is occurring, and the bathymetry is irregular (i.e. not a smooth surface), some error transfer is likely and the degree of error transfer is proportional to the irregularity of the bathymetry.

Two good examples of where utilising a smaller Courant Number are likely to improve stability and accuracy of model predictions include:

- a non-straight channel, one grid/mesh element wide is modelled;
- a key hydraulic control (embankment, levees etc.) is defined in the model topography using a single element.

Employing single lines of cells in a 2D topography to model features that behave as broad crested weirs is not recommended and these should be defined using 1D elements that are inserted within the model grid.

6.4.2. Orientation of the Fixed Grid

Rotation of the model fixed grid is sometimes applied for a finite difference method in urban applications. It is very common in coastal and offshore applications where considerable benefits in the use of a model can be achieved through rotation, whilst for flooding applications the benefits are more limited. In general, the purpose of rotation in flooding applications is to align the grid axis to the dominant flow direction or to flows in the area of interest. There are two advantages that are gained:

1) A larger simulation time step can be adopted.

2) The topography or bathymetry can be more accurately developed to represent the flowpath.

The ADI (Alternating Direction Implicit) numerical schemes (refer to Chapter 2) that are commonly applied in finite difference methods have an implicit solution when solving along the grid axis. The alignment of the flows with the highest CFL (Courant–Friedrichs–Lewy) condition to the numerical grid ensures that the maximum timestep can be achieved.

The definition of flow paths such as channels and floodways within a DEM are poorly represented when they are aligned at 45 degrees to the grid alignment. This can lead to poor definition of the hydraulic conveyance and therefore limits the accuracy of the model

In practice the rotation of model grids is quite common and advantageous for coastal and offshore applications where the model boundary conditions are difficult to define. In such cases, the model is often rotated to improve the boundary definitions. In floodplain applications the flow directions vary widely and there is more flexibility with definition of boundary conditions. Consequently, the advantages in rotation of the grid are more limited, making the use of this technique limited to specific and unique conditions.

6.5. Temporal (Time) Schematisation

The temporal schematisation of the model is the process of selecting a simulation timestep and an interval for saving the model results. The temporal schematisation is critical for successful modelling outcomes.

6.5.1. Timestep

The model simulation timestep is dependent on the model grid/mesh resolution and the schematisation of features in the model. As a consequence, the impact of poor model schematisation can lead to inefficiently small timesteps which in turn will produce excessive run times. The impact of excessive run times should not be underestimated and in practice it becomes impractical to calibrate and apply the model effectively.

There are generally 2 choices for selecting a model timestep which are:

- a fixed regular timestep, or
- an adaptive timestep.

The fixed regular timestep allows the modeller to pre-determine the model run time and to set the saving step (in which model results are saved) as a regular multiple of the simulation time step. However, the timestep will need to be set at the shortest time interval necessary for stability of the model during the most energetic or deepest flows during the simulation. This typically occurs for only a very short period of time during the peak of the flood hydrograph. Consequently the model simulation time is longer than is necessary as it is fixed for the entire simulation. However, the modeller can be sure that the simulation will complete within predetermined run time.

The adaptive time step allows the model to determine the appropriate timestep necessary to maintain stability as defined by the Courant condition. The modeller will typically set a maximum and minimum timestep allowable. This allows the model to timestep at relatively longer timesteps when the flow is shallow or less energetic and shortens the timestep during the peak of the flow event. In theory, this should allow the shortest run time for the simulation to be achieved whilst maintaining model stability. However, in practice the adaptive timestep method can often lead to excessively long run times. This is due to the impact of a few minor locations in the model where short lived energetic fluctuations in the flow can lead to the minimum timestep being selected for excessively long periods of time

Run times can also become excessive if the period that it takes for the flood wave to propagate through the model is very long. For example, simulations of large river systems or of flat terrain where the critical rainfall duration is long, will have propagation times in the order of days, if not weeks. However, small catchments with short critical durations may only have propagation times in the order of hours. Therefore, some idea of the likely propagation period is needed before finalising the model resolution and extent.

6.5.2. Save Step

The model save step is an important issue to consider during the model schematisation process. As 2D models will typically produce very large results files if all the results are saved, there is a requirement to select an appropriate saving step for the results.

The model saving step needs to be sufficient short to be able to define the shape of the hydrograph in time. The model save step also needs to be sufficiently short to enable the observation of stability issues that may occur during the simulation. If a model is being saved at a longer time interval than a higher frequency oscillation in the model then it would not be easily identified and could be missed. It is important that the model is checked thoroughly by saving all time steps at specific points or at small regions in the model domain. This allows for the observation and checking of stability issues without the need to save the entire model at all timesteps. It is generally impractical to save all results at all timesteps in a 2D model and it will typically exceed the limitations of most computer storage and hardware to do so.

6.6. Boundary Conditions

The development of the boundary conditions is an important stage in the schematisation of the model. Three steps need to be carried out for appropriate schematisation.

1. Location of Boundary Conditions

2. Choice of Boundary Condition Type

3. Schematisation of the Boundary

6.6.1. Location of Boundary Conditions

The location of the boundary conditions is of critical importance. In general, boundaries should be located as far away from the study area as is practicably achievable. Any boundary condition on a hydraulic model requires a description of the water level, flow rate and velocity, flow direction and water surface slope across the boundary. In most situations, these flow conditions are rarely available as input data time series. Consequently, a range of assumptions are made in the definition of these conditions. While some models provide the ability to specify these explicitly, most models have generic assumptions incorporated into the model system to facilitate the automatic calculations of the range of parameters required.

As an example, the water level at a boundary condition is typically defined as a time series of recorded level. The other flow conditions are assumed or calculated based on the general assumptions or predefined conditions, such as an assumed flow direction across the boundary and assumed water surface slope. In this example, these assumptions, when combined with the water level time series, allow a discharge to be estimated across the boundary.

The specification of these conditions on the boundary introduces errors into the model predictions. Over time, these errors propagate through the model domain and may eventually pass through the model domain and out through another boundary. In a well-developed and tested model, these errors become dampened as they propagate through the model domain. If the boundary conditions are located remotely then the errors become insignificant at the area of interest.

As an example, if a high flow rate is introduced through a topographic boundary condition that has small conveyance (restricted flow capacity) then high velocities and a significant velocity head results. This may cause large errors in the momentum flux into the system leading to errors in the flow patterns, water level and velocities downstream from the model boundary into the model domain. In this case, provided the boundary is located well away from the area of interest so that these effects have fully dissipated, the presence of these unrealistic flow patterns can be considered acceptable for the purposes of the investigation.

6.6.2. Type of Boundaries

The types of boundary conditions that are applied are important in determining the results produced by the model. The boundary conditions can be defined into two broad categories of;

- external boundary conditions, and
- internal boundary conditions.

The most common boundary conditions applied in hydraulic models are external boundary conditions with a flow or discharge boundary defined along the upstream boundary of the model and a water level defined at the downstream external boundary.
The boundary condition type can be described using one of the following for specifications:

- Flow time series specified which is distributed across the model boundary grid/mesh points,
- Water level time series which is assumed to be constant across the model boundary,
- Flow and water level specified in combination as an input time series and distributed along the boundary,
- Flow or water level specified as a 1D line of values along the boundary for each timestep,
- Transfer boundary where the water level, flow, velocity and water surface slope are provided from another model,
- Rating curve along a model boundary (combination of water level and flow)

The combination of boundary types is critical and must be considered in combination with the specification of initial conditions. In general, the boundary conditions for flooding models should be designed with upstream inflow or discharge boundaries and downstream water level boundaries. This ensures that any errors or uncertainties associated with initial conditions are "washed out" of the model. If other combinations of boundary conditions are used then the initial conditions will not necessarily be "washed out" of the model. The initial conditions will then significantly affect model simulation results and the results may not be reliable.

6.6.3. External Boundary Conditions

The schematisation of the external boundary conditions can vary across the range of model types and even within specific model software packages. The schematisation of external boundary conditions is therefore highly dependent on the specific case and software being used and it would be inefficient to describe all types of boundary conditions in detail. However we can define some general principles for schematising boundary conditions that are important to consider.

If general the modeller should approach the schematisation of external model boundary conditions in a similar manner to how a boundary condition would be conceived for a physical model. The modeller should consider the physical flow characteristics at the boundary in the real world and should attempt to schematise so as to minimise any artificial flow behaviour that is induced by the boundary condition. Issue that should be considered include:

- Align the model grid to be normal to the boundary flow streamlines if possible;
- Avoid rapid transitions in flow regime at the boundary;

- Avoid placing the boundary where turbulent flow are likely to be crossing the boundary;
- Minimise the wetting and drying on the boundary if the flooded boundary changes in width substantially during the simulation;
- Ensure that the boundary condition does not restrict or expand the flow substantially at the boundary.
- Preference for specifying an upstream inflow discharge boundary and a downstream water level (or rating curve) boundary in combination.

As discussed, the boundary conditions should be located as far from the area of interest as possible. This will minimise the possibility of boundary effects and errors influencing the model results within the study area. The specification of the boundary conditions will therefore have a significant influence on the grid/mesh resolution. In general, the boundary condition should be identified as the first task that is carried out when conceptualising and schematising a model.

6.6.4. Internal Boundary Conditions

Internal boundary conditions are specified to control either the flow or the water level at grid/mesh element(s) within the model and not along the edge of the model grid. There are generally two types of internal boundary conditions:

- Internal inflow points (sometimes called sources or sinks),
- Internal flow or level controls.

The primary issue in defining internal inflow boundary points is to ensure that the flow rate is compatible with the grid or mesh resolution. There should be sufficient conveyance into or out of the element(s) where the boundary condition is specified to allow the model to accept the flow without introducing significant disturbance to the natural flow streamlines. If a large flow is forced as a boundary through a relatively small cell element with limited flow area; the model will produce an excessively large velocity and water level gradient to achieve continuity with the flow volume. If this occurs then significant momentum can be artificially introduced to the model at this location which will then influence water levels and flow patterns for a relatively large distance away from the boundary cell.

Internal control boundary conditions are a special form of boundary condition and are generally not recommended unless there is a strong compelling case for their use. An internal boundary condition will force the model to reproduce a predefined hydraulic behaviour within the model domain. The most common internal boundary condition is a forced rating curve at an internal cross section of a 1D model. These boundary conditions are highly "reflective" and will introduce distortion and disturbance of the flow behaviour far from the actual boundary point. We would not recommend the use of this type of boundary for most flood modelling applications.

6.7. Hydraulic Controls

Hydraulic controls are defined in a 2D model by either embedding 1D representations of the flow control and linking the 1D structure with the 2D domain or replacing the momentum equation in the 2D model with an equation that represents the hydraulic structure.

Hydraulic structures are incorporated as 1D elements within the 2D domain to achieve a better representation of their hydraulic behaviour. Generally for structures, there are well-accepted equations to describe the hydraulic behaviour that have been confirmed by lab testing. Many of these equations have empirical coefficients that account for the energy loss due to expansion and contraction.

It is common practice to use hydraulic structures to represent weirs, culverts, flow control devices and bridges. These types of structures typically involve 3D contraction of the flow upstream and expansion of the flow downstream of the structure. Contraction and expansion of flow and the resulting energy loss may be fully or partially accounted for in the 2D model, depending upon the grid/mesh resolution. The finer the resolution, the more likely it is that the contraction and expansion losses are accounted for within the 2D domain. This is also dependent upon the model appropriately representing turbulence effects via an eddy viscosity value (refer to Chapter 10). However, if the grid/mesh resolution is coarse, additional losses may need to be applied to the 1D structure itself to compensate. Generally, only weirs are implemented as 2D hydraulic structures with the momentum equation being replaced by a weir equation. The majority of hydraulic structures, including weirs, can be implemented in 1D in most modelling packages.

While 1D structures are generally easy to incorporate into a 2D domain, the way in which they are linked to the 2D domain is an important factor in ensuring they perform as they should. Most model software allows 1D structures to link to the 2D domain on the basis of water level and continuity but neglect the velocity of the flow. In addition, the orientation of the 1D structure to flow is not able to be specified. There are options in some models to transfer momentum between the 1D and 2D domains but this type of link is generally only used in very specific applications and can be challenging to implement. Water level compatibility is generally a reasonable assumption when velocity is low or the connection is perpendicular to the main flow direction.

Chapter 9 provides detailed guidance on linking of 1D and 2D domains and some of the advantages and disadvantages of different approaches. Chapter 10.10 contains practical guidance and discussion on how to represent structures in 2D models. The reader may also be interested in Chapter 10.11, which presents discussion on incorporating sub-grid features, such as fences and buildings, into a 2D model.

7. CHAPTER 7 - CALIBRATION AND SENSITIVITY TESTING

7.1. Introduction

Calibration of a hydraulic model to historical floods is a critical and important stage of the model's development. Calibration demonstrates that the hydraulic model is capable of reproducing flood behaviour within acceptable parameter bounds. In the absence of historical flooding information, the modeller needs to cross-check the model against other modelling or desktop analyses.

Regardless of hydraulic model type or complexity, the calibration process is critical to ensure the model is capable of adequately representing the physical system and, in doing so, producing reliable results. While 2D hydraulic models provide a superior numerical solution, accurate results are not guaranteed. Calibration is just as important for 2D model applications as it is for simpler models. This chapter discusses some of the particular issues related to the calibration of 2D hydraulic models. While the discussion focuses on 2D modelling, the principals generally apply to all types of hydraulic models.

During calibration the full impact of the trade-off between model resolution and run time (discussed in Chapters 2 & 3) is felt. Calibration runs and the calibration process can be very time-consuming and costly. The “benefits” of a finer resolution model may be negated by the fact that the excessive run times limit the number of calibration runs able to be undertaken, resulting in a model that is not as well-calibrated as it could be. As calibration is an iterative process with no perfect answer; therefore the modeller needs to be critically aware of acceptable ranges for model parameters and the uncertainties of the model's inputs. The modeller also needs to be aware of the software's limitations, and how varying the schematisation affects results. The modellers, client and other stakeholders also need to be very clear on what constitutes an acceptable calibration.

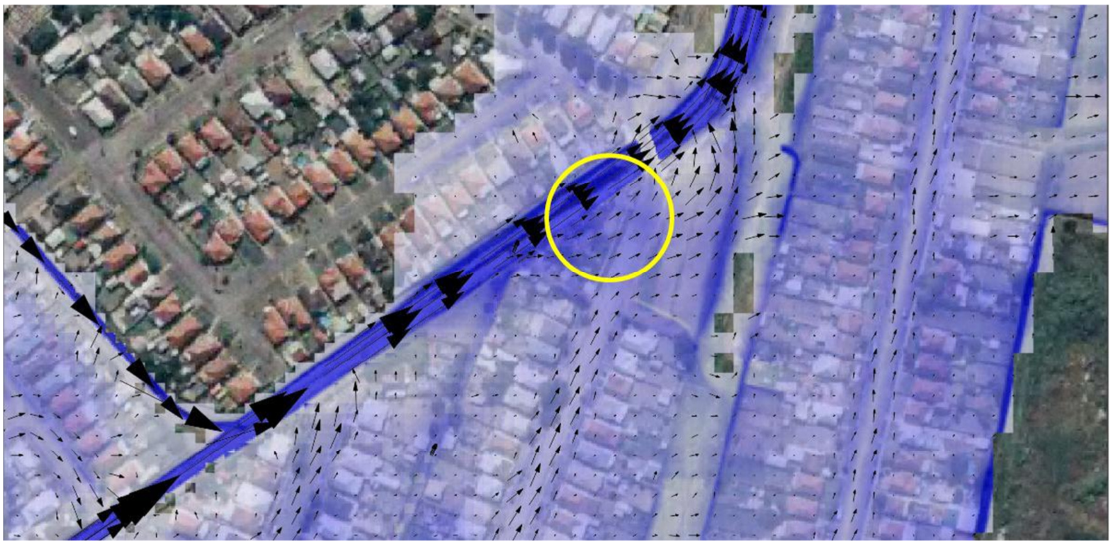
The longer run times of 2D hydraulic models means that model calibration can take significantly longer to complete than for 1D models. However, correctly schematised 2D hydraulic models have less uncertainty and require less engineering judgment than 1D models, so fewer calibration runs are usually required.

Verification (in the context of this document) is the undertaking of additional model simulations or analyses after the model's calibration to “independently” proof or verify the model.

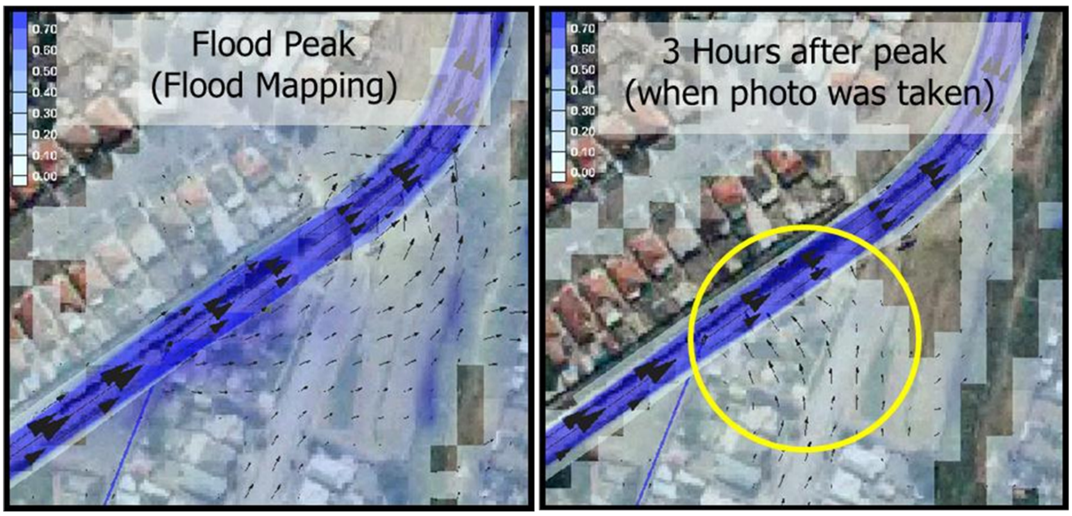
At the end of this chapter a FAQ (Frequently Asked Questions) section is provided to offer some guidance whilst calibrating a model. Brief stories of past calibration experiences are also provided to help readers appreciate some of the interesting and challenging incidents that may occur when calibrating a model!

The photo below was discovered after the model was calibrated, so in effect was a form of model verification. Taken 3 hours after the flood peak it shows water entering the channel from the left.

However, the mapping of the flood peak showed the flow direction parallel to the channel as shown in the yellow circle.



An animation was created showing the changing flood behaviour and flow patterns during the event. The image on the right shows the floodwaters returning to the channel 3 hours after the flood peak as indicated by the photo above.



7.2. Calibration Data

7.2.1. Selection of Calibration Events

Prior to collecting and analysing all data for a calibration exercise, suitable historical events need to be identified and selected. The modeller should primarily consider the:

- Amount, type and quality of suitable data available for each event.
- Magnitudes of the events as to whether they are of a similar size to that of the primary design events.

Each calibration event must have sufficient historical flood observation and reliable topographic information and boundary data at the time of the flood. Often this means that events used for calibration are relatively recent, as the data sets are likely to be more complete. Larger floods that may have occurred longer ago may not be suitable for calibrating to due to the lack or scarcity of key data sets.

Calibration events should ideally also span the magnitude range of the intended design events with a preference for the more important design floods (eg. 1% AEP event). This instils confidence in the ability of the model to replicate flow behaviour over the full range of event magnitudes. For example, a small flow event that is confined to the channel and drainage infrastructure will have a substantially different behaviour to a large flood event that has broken the banks and is flowing overland. If the model has only been calibrated to the in-bank flow magnitude, confidence in its ability to replicate overland flow will be lower.

For tidal sections of a flood model, a tidal calibration is a useful additional calibration step, and is particularly recommended where storm tide inundation and interaction with catchment flooding is important. Tidal calibration data often exists, or can be readily measured, and is usually an accurate data set. It also provides a check that the model can reproduce any tidal amplification.

> The 1998 flood in Katherine was larger than a 1% AEP event. There were extensive water level measurements taken throughout the town, many photographs and videos and the flood discharge was gauged at the gauging station. Therefore, the data available for calibration at Katherine for this event could be regarded as ideal: a large recent event with a reliable and extensive dataset.

7.2.2. Types of Calibration Data

Types of calibration data are detailed in Chapter 5 and include:

- Historical changes to topography, land-use, structures and drainage infrastructure.
- Records (photographs) of bed, bank and floodplain vegetation levels to assist with interpretation of roughness and provide record of prevailing conditions.
- Rainfall records (daily and pluviograph records), including in adjacent catchments.
- Gauged water level hydrographs, rating curves and derived flow hydrographs at stream gauge sites.
- Stream flow gauging at gauge sites and over the side of bridge structures (rare, but useful).
- Tidal level records if in a tidal area.
- Flood mark levels, location and measure of reliability. For example, debris marks, watermarks on/in buildings.
- Descriptive anecdotal information and past reports of flood behaviour in general.
- Observations of the rate of rise of flood waters and the time of peak.
- Photographs or videos of historical floods.
- Records or observations on water speeds and/or flow patterns.
- Records of blockage at hydraulic structures such as culverts and gully traps.
- Records and photography of the extent of inundation, noting the time of the photos.
- Information on road/rail closures.

A flood occurred whilst calibrating a model. One of the local landowners phoned and asked if there was anything he could do? Make as many flood marks as you can, and if possible try to record when the marks were made. The local diligently went round hammering nails into trees until the flood peaked. After several weeks trying to calibrate to this fantastic data set, the modellers were desperate, and visited the landowner. The model is always showing much higher levels than you've recorded. After a while the landowner took them over to the creek bank and showed them a levee hidden amongst the trees. Don't tell anyone he says, as I'm not sure if it's legal. In the end he agreed to have it surveyed, and lo and behold the model calibrated beautifully!

7.2.3. Anecdotal Information

While sourcing the quantitative data, it is important to also source descriptive or anecdotal information. This information can be just as valuable in the calibration process, especially where the quantitative data are scarce. Anecdotal information is best sourced through:

- Discussions with local residents on their recollections and observations. For example, they may have experienced a flood event and have noted features such as flow directions, water speeds and the timing of the flood's rise and fall. This information can be valuable to help check that the model's representation of flow behaviour is realistic.
- Information from stakeholders. For example, a road or rail authority may be able to advise how frequently a crossing is inundated and/or for how long. While this may not provide event specific quantitative data, it could be useful as to whether the model is in the right general area of performance.

An old timer recalled how his grandfather remembered a large flood in the 1860s that broke across a ridge in two locations. Today, this would isolate the hospital and be a significant flood risk to homes. The 1% AEP flood did not show this flood behaviour, however, when the 0.2% AEP event was run, these floodways developed. This helped convince the old timer that the modelling was good, and the local council incorporated these floodways into their flood risk management planning.

During a resident survey a local shop owner took the modeller to look at a tree. "See that fork up there; well that was where a pig got stuck." Fortunately, the modelling for that event showed flooding to that height, and was proof to the local that the model was "doing the right thing".

7.2.4. Changing Conditions

The catchment condition data used in a hydraulic model is typically that of the current day. This is due to the fact that an airborne and infrastructure survey is usually undertaken close to study commencement. In using this current day dataset, there are a number of potential calibration issues that the modeller needs to consider:

As described in Chapter 5, catchment conditions at each of the relevant historical calibration/verification periods must be established and used in the model. Changes to conditions that may affect flood behaviour include:

- dam construction
- changes to initial dam storage levels and/or operations
- dredging or siltation of river entrances
- levee construction or raising
- road/rail raising or duplication
- new road/rail embankments
- new culverts or bridges
- upgraded drainage networks
- development on the floodplain
- different crop types or growth stage
- changes in stream bed and bank profiles
- changes to vegetation including seasonal variations

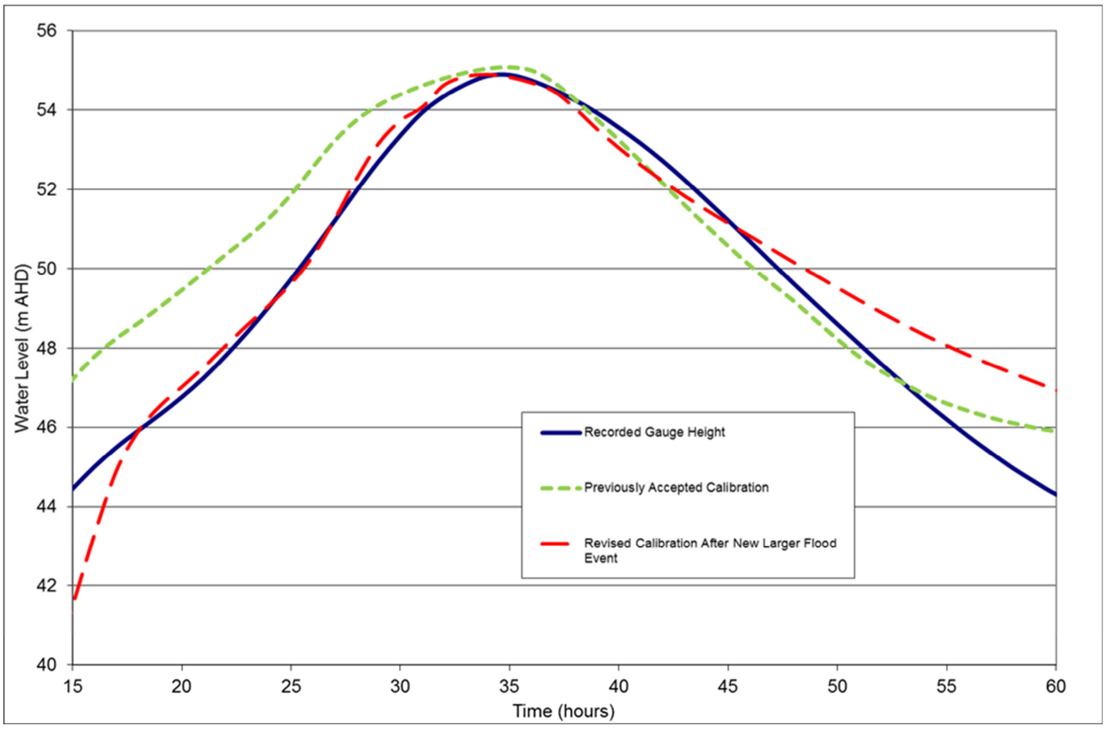
The last major river flood in one coastal area occurred in 1974 and resulted in extensive inundation of the floodplains. . At this time, the floodplain was mostly utilised as grazing land. That land is now developed with extensive canal and flood mitigation works. While model calibrations for these rivers must rely on data from the 1974 flood, the drastically changed conditions mean that calibration results must be treated with appropriate caution.

A 2D model was constantly producing flood levels that were too low in the upper tidal reaches of one branch of a coastal river. However, modelled flood levels matched recorded well in all other locations. Not even extremely high Manning's 'n' values would lift modelled levels to those recorded. It was initially suspected that the recorded levels were erroneous, but this was proved incorrect when the recorded flood levels were independently resurveyed and found to be accurate. It was later revealed by a long term resident that a weir that had been installed to prevent saline water penetrating upstream, had never been completely removed and was still controlling flows. Once this partial weir was included in the model, a good fit was obtained with the same parameters used elsewhere in the model.

A 2D/1D flood model was calibrated to 3 events, with all events smaller than the 5% AEP event. Calibration results for the 3 events were deemed satisfactory and the flood study was completed on this basis.

A few years later the area experienced a larger flood event (about 2% AEP) and the model was recalibrated to this event. The recalibration indicated that the floodplain upstream, which was represented by a 1D domain, needed to convey more water in larger events. Adjustments were made to the 1D domain schematisation upstream.

This resulted in a good calibration to the larger event and an improved calibration to the other 3 smaller events. An example of the improved calibration to the stage hydrographs for one of the smaller events is shown below.



7.3. Calibrating 1D/2D Hydraulic Models

The process of calibrating a model involves the adjustment ("tuning") of the model so that it reproduces, within an acceptable degree of precision, recorded or observed behaviour.

7.3.1. Defining an Acceptable Calibration

Before commencing calibration, it is strongly recommended that the criteria for achieving an acceptable calibration are clearly defined in agreement with the relevant stakeholders. For example, the criteria could be simply that modelled peak water levels are to be within 0.2m of the peak flood marks, and the timing of the flood is to be "consistent" with the observations.

However, the modeller and stakeholders need to understand and appreciate the sources of uncertainty in hydraulic modelling and be open to the impacts of these uncertainties on the modellers' ability to calibrate the model. There can be significant uncertainties associated with the input data, recorded information, hydrologic modelling, model schematisation and some model software. During calibration it is important that the modeller and stakeholders engage in constructive dialogs about these inaccuracies and their affect. **It is far more important to understand why a model may not be calibrating well at a particular location than to use unrealistic parameter values to 'force' the model to calibrate.** It is worth repeating that the goal of a calibration is to produce a model that is capable of adequately representing the physical system and, in doing so, producing reliable results.

Following a large flood event that occurred in 1984, Council organised the survey of over 400 peak flood marks across the floodplains of the affected catchment. These were primarily flood debris marks. Prior to model calibration, Council specified that the calibration criteria was for modelled peak water levels to be within 300mm of recorded. However, calibration was accepted with 50% of points meeting this criterion in recognition of significant proven uncertainties in debris mark levels (refer to Chapter 5) and some of the model inputs.

When calibrating a model to peak flood levels for one historic event, a good match between modelled and recorded was obtained for all levels with the exception of the one recorded by the most upstream automatic gauge. The datum of the offending gauge was checked and no problem was found. In order to match this gauge, Manning's 'n' values needed to be set at values that were outside the normal range and very different to elsewhere in the model. In addition, the peak level at this gauge looked out of place on a longitudinal plot of the river profile. Despite a strong desire to have the model calibrate well to this one gauge level, the client accepted the modeller's advice that confidence in the accuracy of the recorded level was low and it would be compromising the model to fit the data. Not long after the study was complete, a larger flood occurred and the model fitted all gauge data very well, including the troublesome gauge. It was concluded that something had gone wrong with the automatic gauge in the earlier event.

Sensitivity testing of inputs and parameter values is a good way of understanding and resolving the importance of the input/parameter on the model's calibration results. This is discussed further in Section 7.4.

A 2D model was being developed across a catchment with no pluviograph records and limited daily rainfall data. During calibration to an historical event the model consistently produced levels that were too low despite Manning's 'n' values being on the high side of accepted bounds. A sensitivity test was undertaken by increasing flows by 20%, resulting in a much improved calibration. In this case it is possible that the rainfall that actually occurred within the catchment was higher than that indicated by the limited rainfall data available.

7.3.2. Hydrologic Method

The hydrologic or rainfall runoff processes are usually represented by hydrologic modelling, and/or direct rainfall on to the 2D hydraulic model. This document does not provide guidelines on hydrologic modelling methods, however, the calibration and understanding of the uncertainties in the hydrology is critical to achieving a good hydraulic model calibration. Where direct rainfall application on to a 2D hydraulic model is used, there are a number of critical issues that the modeller needs to be aware of as discussed in Chapter 11.

It is strongly recommended that the hydrologic and hydraulic modelling be carried out and calibrated in unison. This is often referred to as a joint calibration.

During the calibration of a 2D model, one recorded peak flood level on the floodplain was proving difficult to match. The modellers had investigated the source of the level and had confidence in its accuracy. However, the model predicted water levels that were too high at this point. Anecdotally the modellers discovered that the large event they were calibrating had caused stripping of riparian vegetation along that section of river. Sensitivity testing of lower roughness values along the banks in that region to mimic the stripping of vegetation showed a closer match between recorded and modelled water levels.

7.3.3. Adjusting the Hydraulic Model

Since numerical models operate by solving equations containing a range of parameters, model calibration is an iterative process of testing combinations of parameter values. The modeller uses experience and judgement to select starting values and compares model results with recorded or observed flood behaviour to judge the appropriateness of the input parameter values. Depending on how well the model results “match” the recorded data and observations, the modeller adjusts the value of one or more parameters, re-runs the model and again compares model results to observations, whilst considering the influence of the parameter changed.

Depending on a modeller’s experience with, and understanding of, a model package, they may adjust one parameter at a time or several between each run. Initial changes to parameter sets should be undertaken in a ‘bold’ manner (i.e. small changes in parameters in the early stages of a calibration should be avoided). The benefits of this are that the modeller will:

- gain an early appreciation for the sensitivity of the model to certain parameters;
- gain an early appreciation for the parameter bounds within which a calibration may be achieved, and
- not waste time on small incremental changes over many iterations that are eventually found not to influence results.

Fine-tuning of parameters is appropriate in latter stages of a calibration. This iterative process is repeated until the modeller and stakeholders are satisfied that an optimum parameter set has been achieved.
The calibration of a hydraulic model is achieved through the adjustment of model inputs and/or a number of model parameters including:

- Hydraulic roughness parameters.
- Energy losses at structures/bends (1D and 2D – refer to Chapter 6 and 10).
- Inflow hydrographs (it is recommended that the hydrologic and hydraulic models are jointly calibrated).
- Downstream boundary location and assumptions, particularly stage-discharge boundaries.
- Model schematisation to more accurately depict flow behaviour.

- Sensitivity testing uncertainties in the input data.
- Inlet rating curves and runoff split between flow to the pipes and the ground surface.
- Blockage of inlets and hydraulic structures.

Failure to reproduce recorded and/or observed behaviour may be due to errors in supplied or collected data or from erroneous entry of data into a model. Cross-checking and sensitivity testing of key model inputs will help ascertain the influence of uncertainties. Section 7.5 provides some advice on the data checking that can be undertaken if difficulties in calibration are experienced.

Initial estimates of values of the hydraulic roughness parameter is usually derived by application of recognised procedures based on physical characteristics of the land-use surfaces, and through discussions with experienced modellers who have successfully calibrated other models. Chapter 10 provides guidance on accepted roughness values.

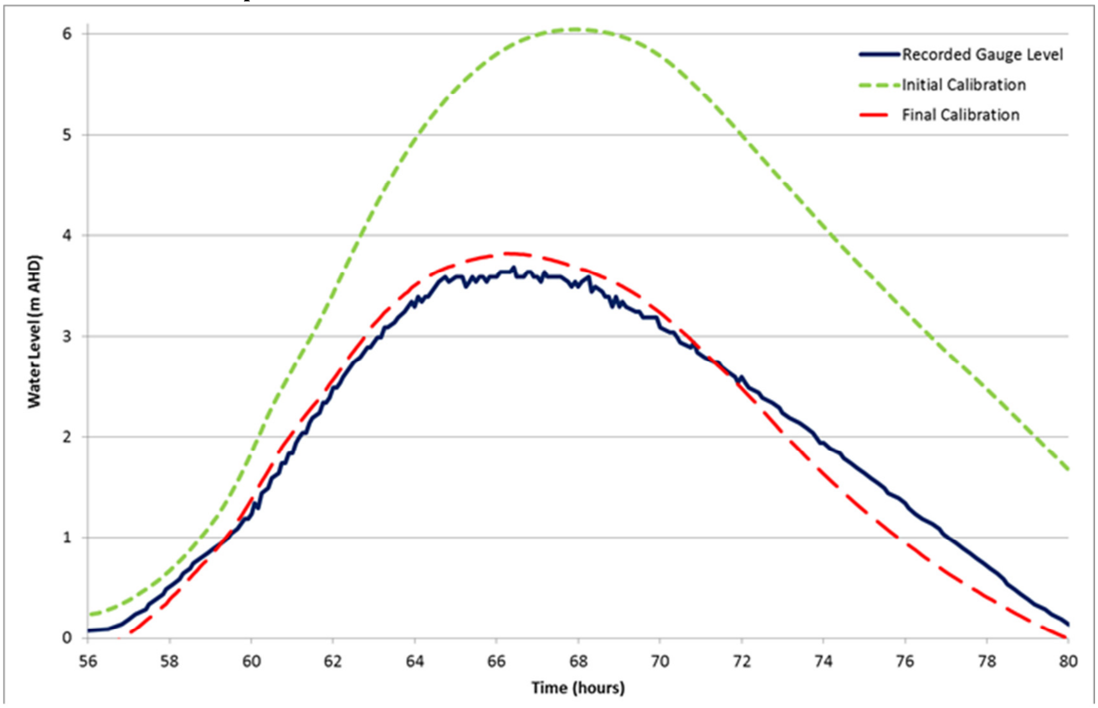
It is to be expected that calibration of a hydraulic model should be possible without significant variations of the roughness parameter values from initial (industry standard) estimates. Values that are grossly inconsistent with the physical characteristics of the waterways and overland flowpaths are a cause for concern, and dictates caution in the use of the model for the prediction of flooding under conditions that are substantially different to the calibration event/s.

Initial calibration of a large river represented fully in 2D, yielded peak modelled flood levels that were significantly higher than gauged water levels. Investigation of model details around one of the bridges revealed a number of separate issues that were all contributing to the higher than desired water levels.

The first of these was due to errors in input data – the bathymetric survey around the bridge was incorrect as it showed an increase in channel invert levels, which in reality did not exist.

The other issues were both due to modeller error: a) the obvert of the bridge had been incorrectly read from the bridge plans and had been entered into the model 0.8m lower than it was, and b) the form loss coefficients had been entered incorrectly.

Once these errors were corrected, peak water levels dropped significantly and calibration fell into place as shown below.



7.3.4. Matching Timing And Magnitude

Ideally, a hydraulic model is calibrated to recorded water level marks and hydrographs. Recorded marks are usually at the flood peak and often spread throughout the model domain. Calibrating to these marks shows that the model is capable of reproducing the peak water level distribution. However, especially if the model only covers a small extent of the overall river/creek system, this does not necessarily mean that the model is well calibrated.

Also, fundamental to a good calibration is the demonstration that the model reproduces the timing of flood events. This may be achieved through calibrating to recorded water level hydrographs (if available), and to observations by locals (e.g. "the flood peaked around midday"). Water level hydrographs give the added benefit of showing whether a model is reproducing the shape (rise and fall) of the flood.

Calibrating to information on the timing of the flood shows that the flood dynamics are being reproduced, and this only occurs if the model's input data and schematisation are satisfactory, parameter values are within typical ranges, the software is suited to the application, and most importantly, the hydrologic method is also reproducing the correct timing. The latter is particularly important when it comes to calibrating a hydraulic model. If the hydrologic method is inaccurate with respect to timing and/or magnitude, satisfactory calibration of the hydraulic model will be difficult, if not impossible. For this reason, jointly calibrating the hydrologic and hydraulic modelling is always recommended.

If parameters such as hydraulic roughness are outside standard values, the calibration may be "acceptable" for that particular event, but will very likely be compensating for inaccuracies in the hydrological modelling, input data and model schematisation. In this case, the "calibrated" model is not suited to representing floods of smaller or larger size than the calibration event, and will be of limited use.

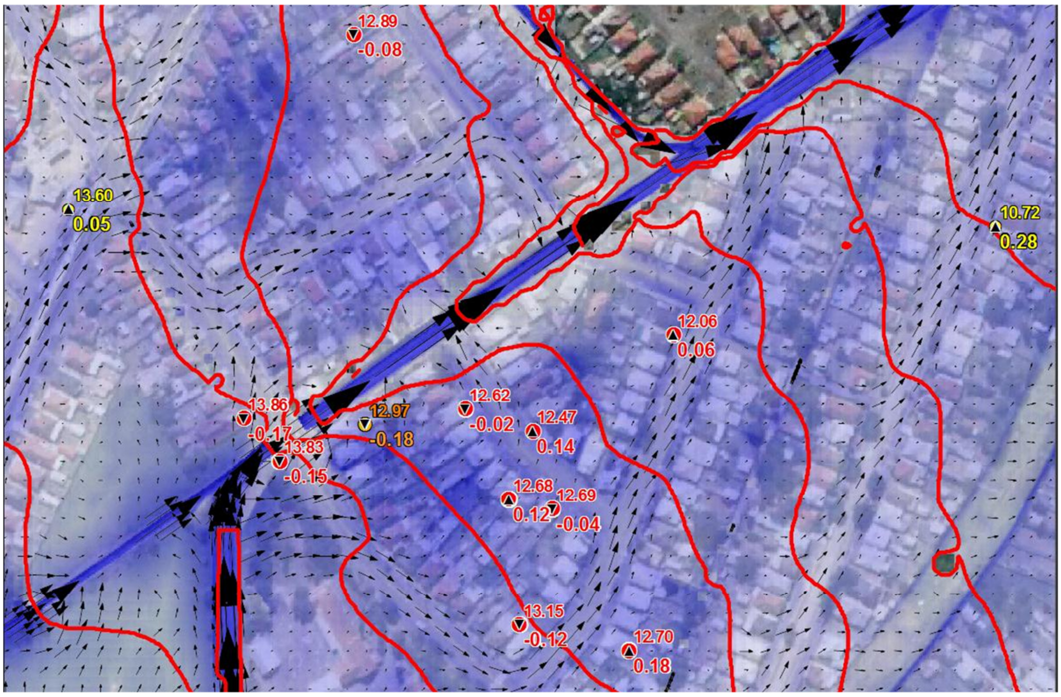
It is important to note that should flow/discharge hydrographs exist for a study area, the flows are not *recorded* but *derived*. A rating curve is used to convert the water levels recorded by the stream gauge into flows. Details on this process and its limitations are provided in Chapter 5. However, it is worth reiterating that the reliability of discharge data is limited by the number and quality of manual gaugings undertaken at the site, the extent of extrapolation beyond the highest gauging of the rating curve and the means by which the rating curve is developed by the hydrographer. In undertaking a calibration using flow discharge hydrographs, it is essential to consider the quality and reliability of the rating curve used to derive the flows. Inaccurate rating curves produce inaccurate flows that will potentially mislead the modeller into using inappropriate parameter values.

7.3.5. Benefits of Community Consultation during Calibration

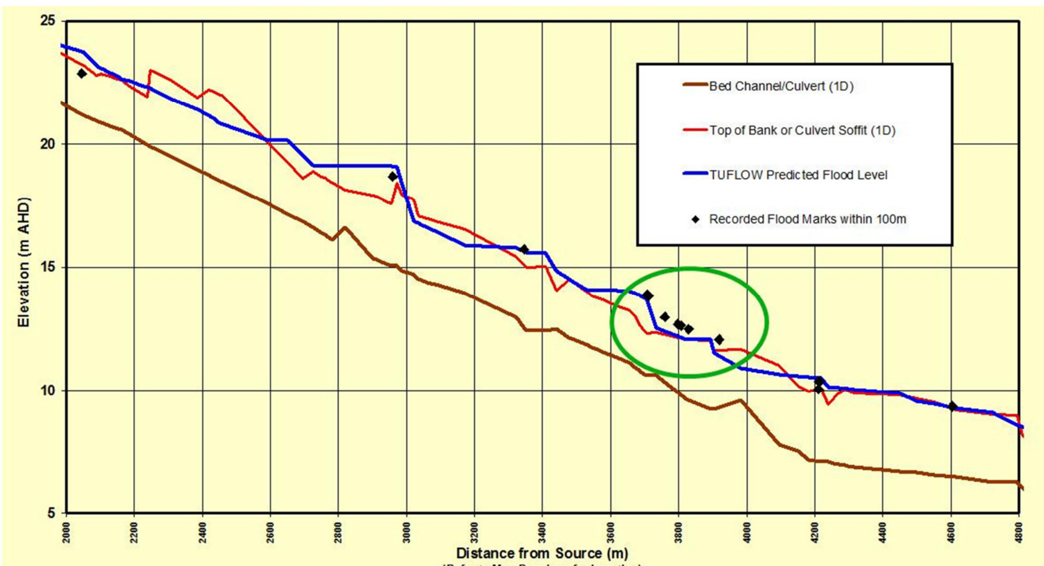
Involving the local community is usually an essential step in gaining community acceptance of the modelling. Participation of the community in the calibration process helps proof the model, and provides them with a sense of "ownership". Constructive community involvement is invaluable in achieving a well-calibrated and reliable model. It is also important that the community has confidence in the model as it will be used to determine aspects of development and mitigation that may impact their livelihoods and properties in the future.

Community members and other stakeholders may have firsthand experience of observed flow patterns during a flood. Interacting with the community through workshops and one-on-one discussions to learn from them, and having the community scrutinise model predictions of flood extent and flow patterns, are very effective ways of involving the community and improving a model's calibration. In this regard, 2D models are significantly more effective community consultation tools than 1D models due to their more accurate representation of complex flow patterns and ability to readily produce visual outputs (e.g. flood maps and animations) of flow behaviour.

The image below shows an example of plan view mapping of the calibration of a 1D/2D model to recorded flood levels. The value to the top right of each point is the recorded value and the value below is the difference (ie. modelled minus recorded). The different colours indicate different types of flood marks (red reliable, orange less reliable and yellow indicative). The red lines are the modelled water level contours at 0.5m intervals. Arrows are 1D or 2D velocities with the largest arrows in the concrete lined channel up to 7m/s.



Traditional forms such as longitudinal profiles remain a good method for viewing and presenting model calibrations. Care should be taken in interpreting results as the levels along the channel where the profile is taken can be quite different to the levels on the floodplain. In the example below, flood marks within 100m of the main channel are shown on the profile, with the recorded levels within the green oval being those in the map image above. The profile suggests the model is under predicting these flood marks by up to 0.5m, however this is not the case as the levels on the floodplain are higher than those in the channel as shown in the map view above.



In previous community meetings, 1D modelling results of calibration events had been presented to a sceptical audience without much success or enjoyment. In the interim a 2D model was developed, and calibrated to the satisfaction of the technical committee. At the next community meeting, whilst showing an animation of one of the events, a local got up pointing to an area with high velocities and commented "that's where our house was washed away". And following the flow patterns, showed how his brother survived the night clinging on to the fridge, which came to rest in a backwater on the opposite bank. The scepticism vanished and the community were on board.

7.4. Sensitivity Testing

Sensitivity testing of model parameters, uncertainties in input data and the model's schematisation (resolution) should be a regular part of a modeller's activities, especially for inexperienced modellers, whilst calibrating a model. It also plays a useful role for establishing the uncertainty of uncalibrated models.

For models that are well-calibrated to a range of flood events and later verified, considerable confidence can be had in the model's ability to reproduce accurate flood levels. This in turn means that factors of safety such as the design freeboard applied to flood planning levels can be kept to a minimum.

However, for uncalibrated or poorly calibrated models less confidence can be had in the model's accuracy, and greater factors of safety (eg. larger freeboards) should be applied to reflect the greater uncertainty. To quantify these uncertainties, sensitivity testing should be carried out where a model's calibration is non-existent or poor.

Examples of sensitivity testing to help quantify a model's uncertainty are:

- Adjust hydraulic roughness parameters values up and down by 20%.
- Increase inflows by 20%.
- For downstream boundaries, not at a receiving water body such as the ocean, vary the stage discharge or water level upwards to check that the water levels in the area of interest are not greatly affected.
- Apply blockages and greater losses to hydraulic structures and inlets.
- Apply lower discharge coefficients across embankments such as roads.

Other useful sensitivity tests include:

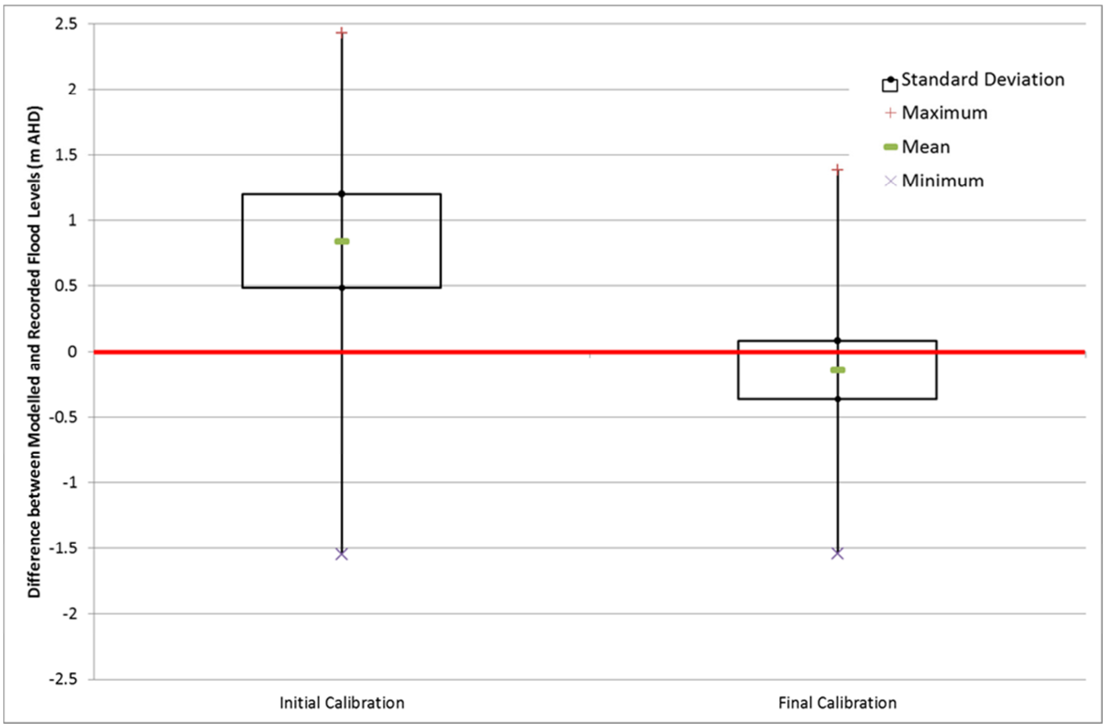
- Making the model's resolution finer to check that results do not demonstrably change.
- Varying the timestep and other computational parameters.

Sensitivity testing is also a very important part of developing a modeller's knowledge base and should be encouraged wherever possible.

After a few weeks of pulling their hair out trying to calibrate to a well-defined flood mark in a house (the model was calibrating well elsewhere), the modellers called the owner of the house. After chatting for a while the owner suddenly remembered "my Dad had the house raised after that flood". Once the flood mark was adjusted by how much the house was raised, a good calibration was revealed! The modellers regretted not making that call a few weeks earlier…

A meandering river modelled in 2D was predicting water levels above that which had been both recorded and observed anecdotally for a recent large event. The model was originally developed without bathymetric data. Bathymetric data were collected and included in the 2D model and modelled water levels dropped by up to 1m.

The image below shows the standard deviation (box) and range (black line) of the modelled peak water levels minus the recorded levels. The red line is a zero difference, so ideally the box is centred over the red line. The left box and whisker plot is for the initial calibration and the right for the final calibration.



7.5. FAQs

Frequently asked questions and possible solutions that may arise during model calibration are presented in this section. The list of questions and topics is not exhaustive, but should provide some guidance on issues that may be encountered.

The golden rule is that if it is difficult to achieve a satisfactory calibration then one or more of the below are likely to apply:

- Inaccurate input data.
- Inaccurate recorded calibration data and observations.
- Unrealistic parameter values.
- The model resolution or schematisation is inadequate.
- Modeller error in developing the model.
- The hydraulic modelling software is operating beyond its limitations.

Therefore, it is paramount for a modeller to understand the: influence of model parameters through sensitivity tests; uncertainties of the model's inputs and recorded calibration data and observations; effect of the model's resolution/schematisation; and limitations of the software.

An experienced modeller had spent weeks trying to calibrate a 1D/2D model to several events. In all calibration events the model was showing lower flood levels. In sheer frustration, he sought another opinion. Within a short time the reviewer noted that the 2D cells covered by the 1D representation of the creek were active, thereby effectively duplicating the conveyance between the creek banks. Deactivating the 2D cells quickly resolved the problem. It happens to the most experienced of modellers – cross-check even the most obvious; seek opinions from others; simply do something else for a while and look at the problem with fresh eyes. If things don't stack up, look outside the box.

Table 7-1 FAQ's

Question	Answer
How should I adjust my parameters?	Don't inch toward a calibration by slightly changing the parameters up or down. During the early stages, test the effects of parameter values at their lower and upper bounds to gain a feel for the parameter's influence on the model's results and ability to replicate the calibration data. When sensitivity testing, it is generally best to change one parameter at a time so that the influence of each parameter is understood.
What parameters should I focus on?	Roughness values (e.g. Manning's 'n') are the most influential parameters in the vast majority of flood models, so these should be the primary focus. Use of non-standard 'n' values indicates that something else may be wrong. At hydraulic structures or where additional energy losses occur that the 2D model is not able to reproduce (for example, fine-scale losses around bridge piers, losses in the vertical at bridge decks or sharp river bends), additional energy or form loss parameters can be adjusted if the 2D software package supports these. Alternatively, Manning's 'n' values could be locally increased, although this is not ideal. Occasionally, eddy viscosity values and formulae may be changed if the 2D scheme solves this term, but as for 'n' values, eddy viscosity coefficients should remain within acceptable bounds. For areas that have significant infiltration effects, a suitable infiltration algorithm and the parameters of that algorithm will be important.
Should my 2D Manning's 'n' values be similar to 1D 'n' values?	Yes, Manning's 'n' values are usually similar for 1D and 2D domains. Exceptions or reasons why different 'n' values may result include: • Where there are rapid changes in flow direction and magnitude (eg. at a structure, sharp bend or embankment opening), 2D model software that solve the full 2D equations will lose energy due to the water changing direction and magnitude. As 1D schemes do not model these changes in flow behaviour, they will not model these losses (hence the need to use entrance/exit losses at a structure in a 1D domain). Therefore, the 'n' value in a 1D model at, for example, a sharp river bend maybe higher than for the 2D model. • As mentioned above, in addition to bed resistance (eg. Manning's equation), 2D schemes simulate energy losses associated with water changing flow direction and magnitude. However, they may need some minor additional energy or form losses applied for fine-scale and/or 3D effects, for example, for modelling bridge piers or sharp river bends with significant 3D circulations. Increasing Manning's 'n' is one way of modelling these additional losses. • 2D schemes typically, by default, apply no side wall friction. Where there is significant wall friction, a 2D scheme may require a slightly higher Manning's 'n' than a 1D scheme, if the 1D scheme is using a

Question	Answer
	hydraulic radius or other formulation that includes side wall friction.
Where should I adjust my Roughness value?	Roughness values are most influential where the unit flow (q or VxD) is highest. Typically, this is along the main flowpaths such as the river, creek or open channels. Adjustment of ‘n’ along these flowpaths will have the most effect on your model’s results. When adjusting ‘n’ values during calibration focus on these areas. Changing ‘n’ values in areas that act as a backwater (low velocity) or are very shallow, that is they have a low q value, will have little effect on results.
Is it okay to use high values of Manning’s n, eddy viscosity or other energy loss parameters to stabilise a model?	It’s never ideal to artificially increase resistance terms or induce increased viscosity to achieve stability, and should be a last resort. Always first cross-check that your schematisation, topography and other inputs are correct in the area of the instability. If used, the increased parameter values should be restricted to localised areas and, preferably, their effect sensitivity tested to establish their influence (for example, double the parameter values and evaluate the impact).
My model results match but my roughness parameters are outside accepted ranges?	For small, localised flood models it’s possible to achieve an acceptable calibration to peak flood levels using unacceptable ‘n’ values. However, it is most likely that this is an indicator that something else is wrong. The most common cause is the inflows are too low if your ‘n’ values are high, or too high if your ‘n’ values are low. Review the inflows to the model. Other reasons could be: • the recorded flood marks are not at the peak or are inaccurate; • if near the downstream boundary review the boundary setup; • inaccurate topographic data; • significant scour and deposition occurred; • check for any seasonal variations in cropping or land-use changes at the time of the event; • high debris loads and/or blockages occurred; • poor schematisation (see Chapter 6); and • the software is operating beyond its limitations. If calibrating to water level hydrograph(s), especially if more than one, it is usually not possible to achieve an acceptable calibration using unacceptable ‘n’ values. Similarly, for models of large systems using unacceptable ‘n’ values will change the dynamics of the flood. For example, high ‘n’ values will delay the flood propagation and produce higher levels upstream and lower levels downstream, while low ‘n’ values will have the opposite effect. High ‘n’ values are sometimes used to represent additional energy losses at, for example, bridge piers, obstructions, blockages or sharp bends. This can be acceptable provided the higher ‘n’ values remain localised to where the additional energy losses occur. Alternatively, try using additional energy or form loss parameters to represent these effects.
Are the stream gauged water levels	The data may have a quality code - check this. Plot the time series hydrograph to make sure all water levels look sensible. Check the datum (if you have the right shape, but the range is offset, good chance it’s a datum problem). Compare

Question	Answer
accurate?	the gauge levels with any nearby flood marks. Has the gauge always been at the same location – it may have been relocated since the event you're modelling? Has the gauge's hydraulic control changed over time? (e.g. erosion of banks/bed, deposition of sediment or calcium carbonate build-up (tufa), or new bridge piers etc).
Is the peak flood level accurate?	The accuracy of recorded peak levels is a common source of uncertainty. Issues and suggestions to be aware of include: • Obtain as much information on the levels as possible, and preferably categorise levels as to their reliability. For example, a water mark inside a house would be a very reliable peak water level, while a debris level may have occurred during the flood recession, not at the peak. Debris in saplings can be misleading as the sapling could be bent over during the flood if located in moving water. • Is the level anecdotal? People's memories may not be reliable, especially if the flood peaked during the night. • Was it truly measured at the peak? If it's not certain the mark was representative of the flood peak, label it as such – the calibration objective on these marks is to be at or higher than the flood mark. • Treat isolated levels with caution – it is always much better to have several levels in the same vicinity to validate each other. • Is it in an area of high velocity? If so, water levels can significantly vary. Where the water is surcharging against an object the level maybe more representative of the water's energy level as some or all of the kinetic energy has been converted to potential energy (eg. against a bridge pier). • Is it on a structure that has changed since the event? • Check for consistency between flood marks by mapping the levels, and/or plotting on longitudinal profiles along flowpaths. Also see Chapter 5 for detailed discussion on accuracy of peak flood levels.
Is the rainfall data accurate?	Usually gauged rainfall data has little uncertainty at the location where it was measured. However, reasons for considerable uncertainty to occur include: • Significant temporal and spatial variability in the rainfall. The rainfall gauge is just a point on a map, and is not necessarily representative of the rainfall elsewhere. • Check with rain gauges nearby and other sources if available. • Radar imagery may assist in helping quantify the temporal and spatial variability of the rainfall, but use with caution as the imagery may not be representative of the rainfall actually hitting the ground. • Daily rainfall totals can be a good source of information to help establish the rainfall distribution.

Question	Answer
	• If the rainfall data is gridded data (e.g. SILO), compare with actual measured data to verify interpolation of gridded rainfall values. SILO gridded rainfall data is available on a daily timestep only and this may not suit the scale of the catchment (i.e. vital temporal variability in rainfall is missing).
My shape looks good, but the timing of the flood is inconsistent with the recorded hydrographs?	If calibrating to a time series hydrograph (either level or flow), check that the hydrologic modelling and/or rainfall has been assigned the correct start time. Check the rainfall and water level/flow recorded timings are correctly synchronised. If calibrating a large catchment using daily rainfall records as input, note that daily records are typically given for the 24 hour period from 9am. This 24 hour period may not coincide with the 24 hour period over which the statistics for the water level/flow record (e.g. average flow or peak water level) have been taken. Always try to use the same period of time to extract statistics for comparison otherwise a temporal shift in the modelled and actual hydrographs may be evident.
Could the ground elevation data be inaccurate?	Yes! A common assumption by modellers is not to question the accuracy of the ground elevations. All ground elevations have an uncertainty. Some of the pitfalls to be aware of include: • LiDAR data's vertical accuracy can be misleading (see Chapter 5). • Aerial surveys cannot (yet) produce ground elevations under surface water. These areas need to be surveyed using other means. • Aerial surveys, particularly LiDAR, can be misleading in areas of thick vegetation. Have these areas cross-checked with ground surveys. • Focus your review of aerial survey data on the main flowpaths. Most models are conveyance dominated so minimising the uncertainty in the main flowpaths is critical to developing a model that you can calibrate with minimal difficulty. • Could the elevations have changed since the historical flood event you're calibrating to? If so, sensitivity testing of the effects of changed topography is recommended to understand the effects on the calibration levels and flood behaviour. See Chapter 5 for a discussion on quality controlling ground elevations.
Are the hydraulic structures and controls well represented?	Is the information from a recent survey or design plans? Has the structure been upgraded since the event (e.g. new road embankment, new culvert, levee re-levelling)? Was there a change to the hydraulic structure during the event? (e.g. embankment eroded, culvert/s blocked with debris, handrails blocked, bridge washed out)
Has the land-use changed since the event?	What exists now may not have existed at the time of the event. As calibration is primarily aimed at setting the model roughness parameters, it is best to select events for which the actual roughness matches the current roughness. But this is

Question	Answer
	not always possible and the following points might assist. Long periods of time between floods can result in heavy unchecked vegetation growth in channels while conversely; a large flood can decimate vegetation, including large trees. What was the case for the calibration event in question? Significant changes in land-use such as urbanisation can have a major influence. Crops change with the seasons and sometimes over the longer term if agricultural practice in the area changes. For example, depending upon the season, sugar cane fields can range from being fallow (minimal roughness) to full-grown (extremely rough). What crops were grown at the time of the event? At what growth stage were they?
Can my roughness value change due to the size of the event?	Yes, for example, tussocky grasses may offer considerable roughness in smaller events but offer a very smooth flowpath in larger events. Well-spaced tree trunks may offer little obstruction in smaller events but once the floodwater reaches the branch height, roughness can dramatically increase. Larger events have a higher debris load, which can result in higher roughness and greater hydraulic losses at locations where debris may be trapped (e.g. closely spaced trees, fences, culverts etc). Try varying your roughness with depth for these land-uses.
Is the schematisation of the model appropriate?	Are the catchment's hydraulic controls well-represented by the model? Are there sufficient grid/mesh cells across the watercourse? Are all crossings modelled? Is there 1D representation of those structures or watercourses too small for representation within the 2D domain? Refer to Chapter 6.
Is the model stable?	Check a time-series output of level to ensure that there are no signs of instabilities.

8. CHAPTER 8 – INTERPRETATION OF RESULTS

8.1. Introduction

One of the most important and commonly overlooked tasks of a numerical modeller is the interpretation of model results. There is a growing tendency amongst modellers and clients to accept results as absolute. However, all results are approximations and need a level of checking and interpretation. Modelling is by nature a simplification of very complex systems relying on many assumptions that need to be considered during the interpretation of results. The modeller is typically the person best suited to undertake this task with no local bias. The modeller needs to draw upon their understanding of hydrodynamics, their experience with the study area, the assumptions the numerical model makes in its calculations and the assumptions they have made during the model development. Those that fail to do this run the risk of being "black-box operators".

In cases where existing model results are to be reinterpreted for some purpose other than that for which the model was originally intended, great care must be exercised to ensure that the results are fit for the new purpose. A common example is the use of a catchment-wide flood study model for detailed analysis at an individual site within the model domain. Care should be taken to ensure that the original model conceptualisation, schematisation and calibration produce results of sufficiently fine resolution and precision for the new task. It may be necessary to make adjustments to the model and run a revised scenario rather than reinterpreting results from the original runs.

This chapter aims to provide guidance to numerical modellers on the interpretation of results, not to define floodplain management policy.

8.2. Results produced by 2D Models

8.2.1. Introduction

Models produce a range of outputs that fall into several categories, each of which will generally have time and space coordinates attached to them. The main categories of model output are:

- Primary variables - usually water level and velocity in the x and y directions;
- Secondary variables - results calculated from interpolation of the primary variables and the ground elevation grid such as depth, flow, velocity-depth relationships, and Froude Number; and
- Simulation output relating to the numerical "health" of the model - such as the number of iterations, convergence achieved and mass conservation error.

Depending on the numerical model scheme used, some results will be calculated at different points in space. For example, water level is often calculated at the centre of a grid/mesh element and velocity at the edge of two grid/mesh elements. Care needs to be taken when combining these variables as some interpolation will be required. This can particularly affect the results when the parameter or terrain is varying rapidly compared to the depth of the flow.

Similarly, results might sometimes require interpolation across time as well as space.

The output from an unsteady 2D hydrodynamic simulation is three-dimensional, having a time dimension in addition to two spatial dimensions. It is difficult to present such information in a conventional report. Results from unsteady 2D simulations are therefore often presented either as spatial maps across the study area, or as a plot of the variable against time at points of interest.

8.2.2. Scalars and Vectors

It is necessary to consider the difference between scalar and vector quantities when interpreting and presenting model results. Scalar quantities have only a magnitude, while vector quantities also have a direction. For example, speed is a scalar quantity in that it has a magnitude, indicating how fast something is moving but with no specified direction of travel. Velocity on the other hand is a vector quantity that has both a magnitude and a defined direction of travel. 2D models typically calculate a velocity component in both the x and y direction which can then be combined to resolve the resultant velocity at that calculation node.

Scalar results (or the magnitude components of vector results) can be presented well using a raster map (i.e. a picture where each pixel represents a data point). The data points can be colour-coded to produce a readily understandable view of how the data change spatially.

However, it is important not to overlook the direction component of vector results, particularly in the reporting of velocity, flow or momentum. The direction at a data point can often be represented by arrows (either of varying length in proportion to the magnitude, or of the same size) to indicate flow direction. These arrows can be overlaid on a raster which indicates the velocity magnitude to present the full amount of information provided in the results.

8.2.3. Output Intervals

Results are typically output to a file at an output interval specified by the modeller. The output interval is usually larger than the computational timestep in order to avoid unwieldy output data file sizes. Care should be taken that the output interval is not so large such that peaks and other important features of the computation are missed.

The modeller should also be aware of the method of calculation of secondary variables, such as velocity-depth products. For example, does the model calculate the maximum velocity-depth product only at the time of the peak depth, or does it take the maximum value of each velocity-depth product calculated throughout the event? This can sometimes be an important distinction and significantly alter interpretation of model results. Details on the method used in each specific model should be sought within the model software user manuals.

8.2.4. Methods of Presentation

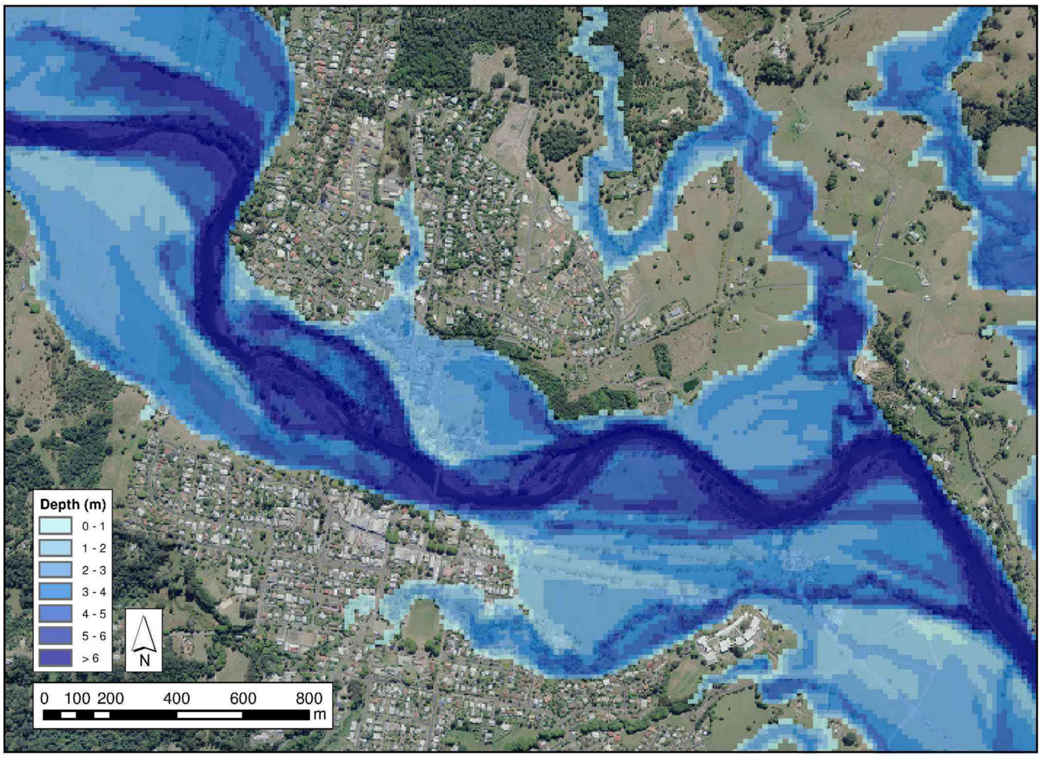
As mentioned in Section 8.2.1 , outputs from unsteady 2D models are 3D in nature, and can therefore be difficult to present in a report. To illustrate this concept, it may be useful to think of the results as a “cube” of data, made up of layers of 2D spatial maps overlaid on each other, with each layer representing a point in time. Time-series information can be obtained by "drilling" through the cube at individual locations, while a spatial snapshot of the results at a given point in time can be obtained from each layer.

Methods used to present model outputs/results typically include:

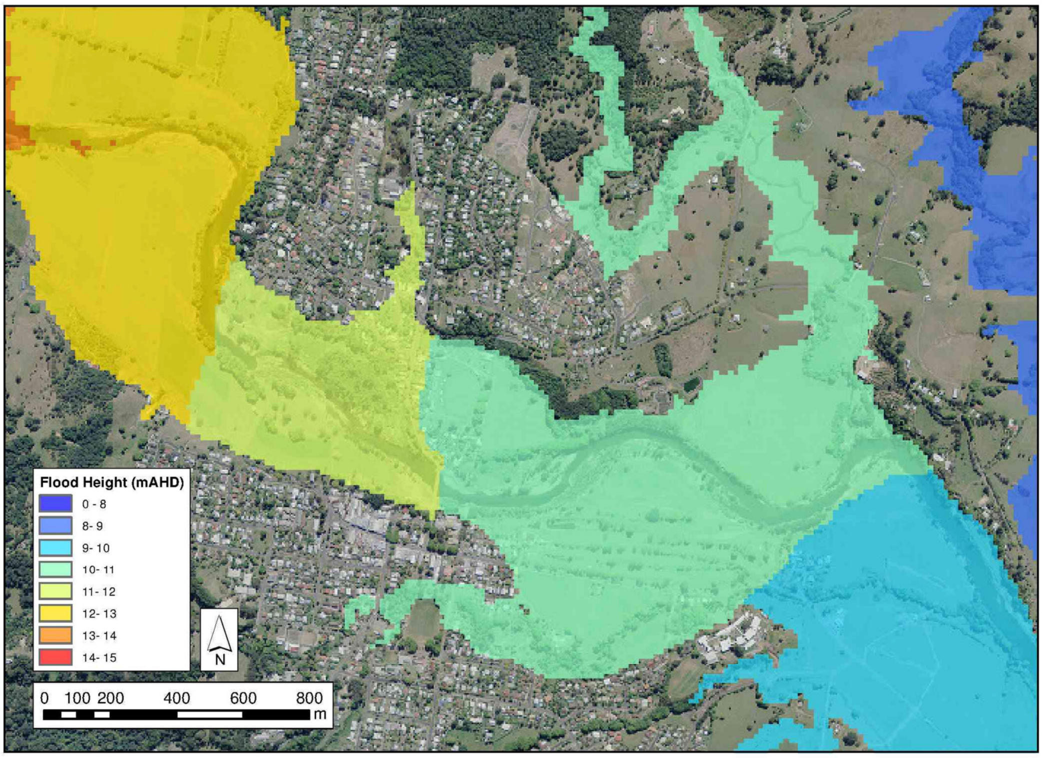
- *Spatial maps (rasters)* – generally used to show variation in peak water surface elevations (stage), depths, speeds, flood hazard categories, impacts of development etc. across the study area;
- *Vector charts* – to indicate the direction and magnitude of flows/velocities, which may be used to characterise the study area in terms of floodways, flowpaths, and storage areas, and/or identify buildings or structures that may be subject to significant flood loadings.
- *Hydrographs* – display the change in water level or flow over time, usually at a location of interest to estimate temporal characteristics such as time-to-flood, rate-of-rise, length of inundation, or to compare with calibration information from stream gauges; and
- *Longitudinal profiles* – indicate the change in peak water elevation along a given flowpath, to compare model results with observed flood levels for example, or to indicate flood levels along a roadway for planning purposes. Longitudinal profiles were traditionally used to present calibration results for 1D models, and the modeller should be aware that some of the information provided by a 2D model (such as super-elevation of the water surface around bends) may be lost using such an approach.

Spatial maps of maximums (such as peak water level, velocity etc) can be obtained from the maximum value at each calculation point in the grid/mesh. This type of output is often very useful and desirable. However, it should be recognised that it does not generally represent a single point in time, particularly if the study area is large or the flow is very unsteady. The maximum in one location may therefore not be coincident with the maximum in another location. For most purposes this will not pose a problem, but it is worth keeping in mind, particularly for calculations of flood hazard (discussed in Section 8.5.3).

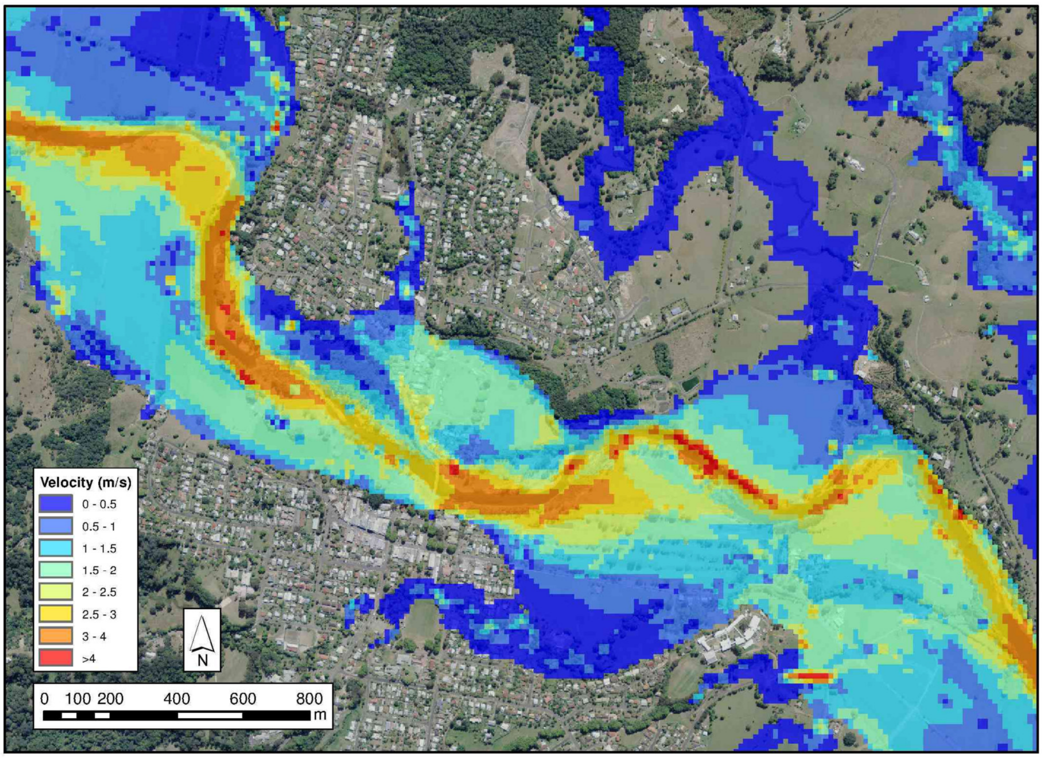
Figure 8-1 depicts examples of 2D modelling results presented using GIS software. An appropriate format for such plots is dependent upon the use and audience of the plot.



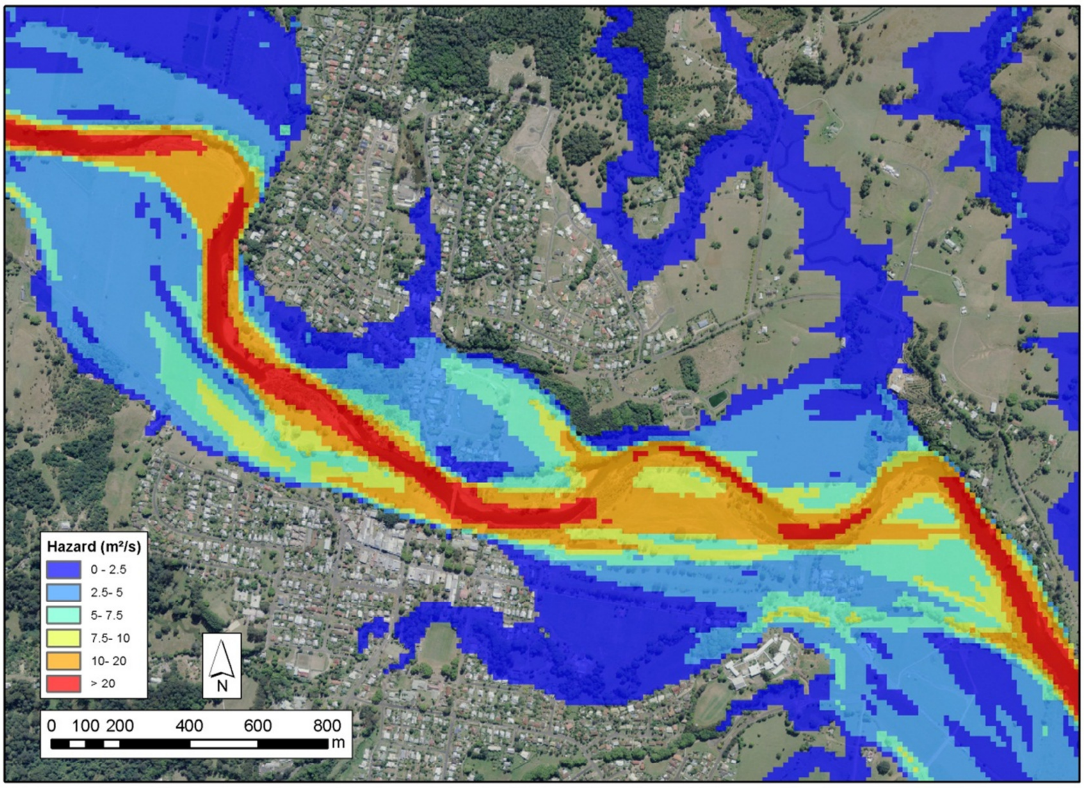
a) Flood Depth



b) Flood Level



c) Velocity



d) Hazard

Figure 8-1 Examples of 2D Modelling Results in GIS

Many of the currently available 2D hydrodynamic modelling software packages either incorporate a GIS-style user interface, or are able to be readily integrated with commercial or open-source GIS packages, which can facilitate the processing and interpretation of results. It may prove particularly useful to the modeller if the GIS interface or package used is capable of managing time-series spatial information of the type produced by unsteady 2D models, and it is worth checking that this functionality exists if a GIS approach is to be adopted.

8.2.5. Animations

It is possible to produce animations from 2D model outputs. Animations can show the change in 2D model outputs over time. Thus, animations can be an excellent way of communicating the full spatial and temporal progression of a flood event. Animations can prove particularly useful when presenting results to clients, government agencies and the community. Animations can be a very effective way of getting community acceptance and input on model calibration. With sophisticated 3D visualisation techniques, animations can produce extraordinary realism, and it may be necessary to keep in mind the uncertainty inherent in the results.

8.3. Checking of Results

The first step that should be applied during the calibration/verification phase is basic checking of the results for obvious errors and model numerical "health." Every new option or model run should have some level of sanity checking to ensure that the results are consistent with what was expected. Gross errors or instabilities identified during this check will often be due to poor schematisation, such that the model is incapable of finding appropriate solutions. Chapter 6 may provide some assistance in resolving these difficulties.

Developing a process for checking that model results are sensible and consistent is a vital quality control measure for the modeller. The modeller needs to satisfy themselves that the model results are reasonable prior to publishing them in a report. The following is a checklist that the modeller should consider when interpreting results:

- *Mass balance* – errors greater than 1% to 2% should generally be investigated, and the cause of the errors identified and rectified where possible;
- *Runoff volumes* – the total runoff as a percentage of rainfall volume should be determined and checked against typical runoff coefficients for similar catchments;
- *Runoff rates* – can be used to check that the runoff rates predicted by the hydrologic model do not significantly diverge from runoff rates predicted by the hydraulic model. If divergence is significant, reason(s) for such should be determined.
- *Continuity* – discharge hydrographs should be obtained at several locations along each flowpath, and at locations upstream and downstream of major flowpath intersections, to check that the continuity and attenuation of flows is reasonable;
- *Stability* – the results should be checked for signs of instability, such as unrealistic jumps or discontinuities in flow behaviour, oscillations (particularly around structures or boundaries), excessive reductions in timestep or iterations required to achieve convergence. Many models will specify criteria based on the Courant number (refer to Chapter 3) that can be

checked to assess model instability;

- *Froude numbers* – Froude numbers should be checked to identify areas of trans-critical and super-critical flow, and the implications of this flow behaviour on the model results considered. In general, model results in areas of trans-critical flow should be used with extreme caution. Flow over embankments, levees and other hydraulic control structures should be roughly checked with suitable hand calculations, such as the broad-crested weir equation;
- *Model startup* – many models do not perform well from a completely "dry" start during the initial wetting stage. The modeller should consider using a suitable "hot-start" condition if such functionality exists, or should exclude results from the very start of the model run from their analysis. This can be particularly important near structures;
- *Structure head losses* – head losses through structures such as bridges, culverts, siphons etc should be checked against suitable hand calculations. More discussion on how to deal with structures is presented in Chapter 10. In particular, consideration should be made of the amount of expansion/contraction losses that are captured by the 2D schematisation, and whether the flow regime is adequately handled by the model; and
- *Steep areas/shallow flow* – it may be difficult to interpolate flow depths where steep shallow flow is occurring, particularly if the flow is not sub-critical. It may be necessary to check results against total energy calculations in such locations.

8.4. Accuracy and Uncertainty of Model Results

The accuracy of the model results is a function of the quality of the data used in the development of the model, such as topographical data, boundary conditions, model schematisation, and the calibration data available. A model that is well-calibrated is not necessarily accurate for other events. Models are typically calibrated using very few historical events, which are often of much smaller AEP than the design floods the model is later used to estimate, and with a lower density of calibration data than is desirable. A well-calibrated model would generally require having as many calibration events as parameters. Typically, only 2 or 3 calibration events are used in practice due to the limited availability of data (refer to Chapter 7).

Modellers need to be particularly careful when modelling events that are much larger or smaller than that to which the model was calibrated. Often there is little choice but these results need to be treated with a high degree of uncertainty.

Calibration results that appear to match observed behaviour "perfectly" should not be assumed to be perfect, since there is a level of uncertainty in the observed data, in the topographic data used to build the model, and in the modelled results. There may be considerable uncertainty in recorded data about the actual magnitude and location, particularly with historical flood marks and stream gauge records, as discussed in Chapter 5 and Chapter 7.

The modeller should be aware that if they were to develop two models with different software packages (or versions of the same package) that used the same input data and parameters they would get different results. Model results are approximations.

Freeboard on flood level estimates is often used as a way to make allowance for the various uncertainties in the estimates, whether derived from models or flood frequency analyses. Gillespie (2005) argues that uncertainty in factors typically included in the freeboard, such as model error, waves, afflux and climate change, may vary between studies or locations. For this reason the freeboard allowance should be based on the best estimate of uncertainty in the factors relevant to the specific study, rather than be a blanket adoption of a standard or default value.

8.5. **Specific Types of Result Analysis**

8.5.1. Impact Assessments

Impacts are typically calculated as the difference in water level, flow or another model result between a base/existing case and a scenario (typically representing a development option) case. The difference is calculated to estimate the effect of a proposed development scenario in the form of a new building, subdivision, levees, bridges, roads, and quarries etc. It is important to understand the model results and what they mean before deciding if impacts are realistic and should be included in the assessment.

Previously, 1D models have been used to define flood level impacts. However, due to the nature of a 1D model, the impacts were constant across a cross-section. A 2D model produces results that can vary spatially. This means that impacts will not be constant but will vary across an area or channel. In some locations, the impacts will be higher or lower than a 1D model of the same situation. The use of a higher resolution spatial grid/mesh may mean that the model produces more highly localised impacts.

It is important to consider whether an impact produced by a model is physically realistic, as it may be falsely produced by the model schematisation. For example, the model may produce results that indicate a slight localised increase in flood levels that lead to a disproportionate increase in flood levels at another location, due to the model discretisation. (see Figure 8-2). The modeller should exercise judgement to determine whether such an impact is physically realistic.

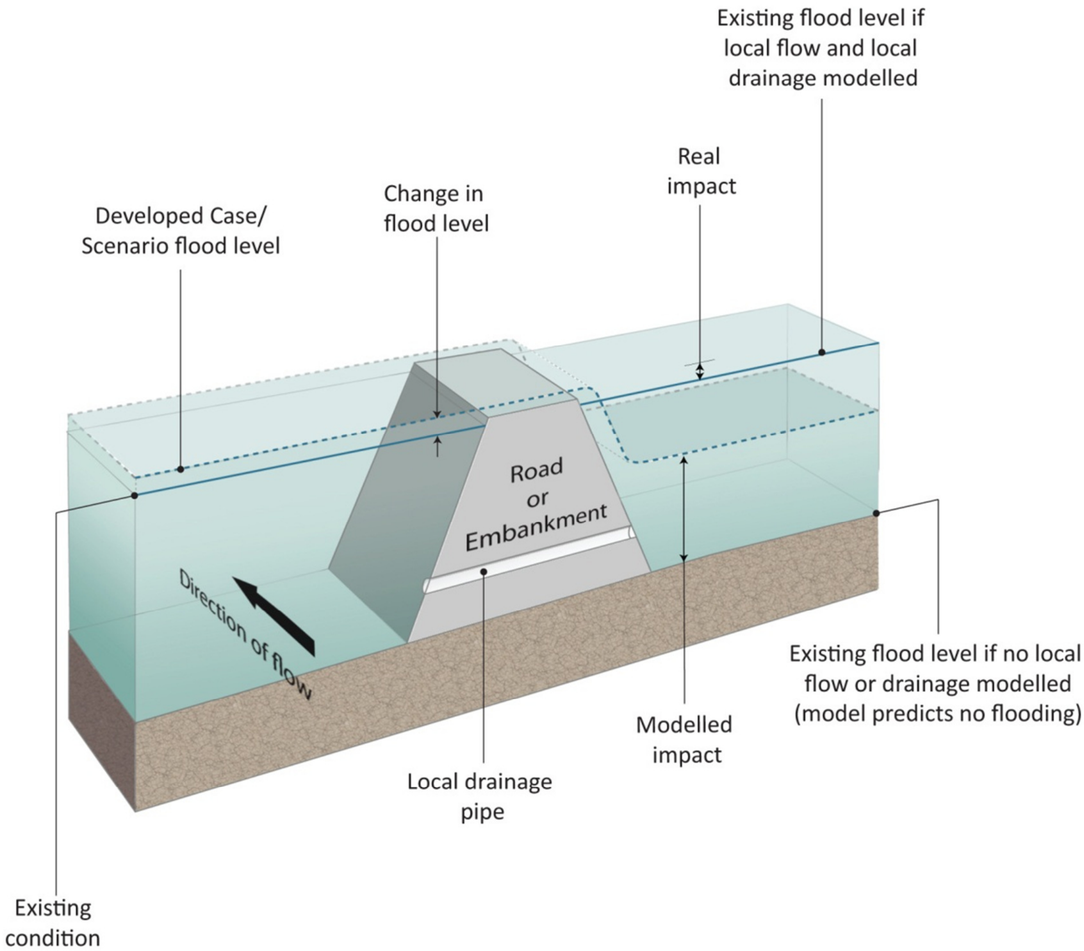


Figure 8-2 Example of Flood Impact Results that are not physically realistic due to coarse model schematisation

Bear in mind the model accuracy, precision and uncertainties when deciding whether an impact is real. At each timestep, models iterate until key variables such as depth and velocity only change by a small amount. Sometimes, slight differences between options or even starting conditions that should have no effect on the result can result in minor impacts in the order of millimetres. These numerical impacts can exceed the magnitude of the expected/real impacts. Typically, results are not reported to the nearest millimetre, and impacts less than 0.01m are not reported, as they are considered to be within the precision of the numerical model and data. However, the unrounded model results should be used to calculate the impact.

Often, when the cumulative impact of development is considered, it is appropriate to report impacts less than 0.01m when considering the contribution to the cumulative impacts. It can be professionally irresponsible in some cases not to look at the cumulative impacts. The reader is referred to the National Manual for more discussion on cumulative impacts.
When reporting impacts consideration should be given to the following:

- scale and extent of the impact,
- accuracy of model,
- accuracy of topography, and
- spurious impacts that aren't real.

As discussed in Section 8.2.2, velocities reported in raster colour figures are in fact the magnitude of the velocity with no directional component. Therefore, when investigating impacts to *velocity* due to a particular scenario, the velocity magnitude result grids cannot simply be subtracted.

8.5.2. Floodplain Management

One of the most common uses of hydrodynamic models is in flood studies and floodplain management studies. Floodplain risk management relates to managing the risks to people and property associated with the occupation of flood prone land. There are detailed manuals available on floodplain management including SCARM, the National Manual and the NSW Floodplain Development Manual. The NSW and other national manuals also have a series of guidelines to assist in the interpretation of technical results. The key uses of hydrodynamic models for floodplain management are to define and inform:

- Flood levels for a range of flood events
- Hazard (as velocity depth product)
- Floodplain Categorisation
- Evacuation planning

8.5.3. Hazard Assessment

An essential step in the floodplain risk management process is to define flood hazard. Flood hazard is a measure of the risk of damage to people and property resulting from a particular flood event. It is typically analysed for one (or more) design flood event(s) of specified AEP. The determination of flood hazard is usually a two-step process. Firstly, preliminary hazard is identified (or mapped) using a hydraulic definition (based on velocity and depth of floodwaters), true hazard can then be refined from this by considering a range of modifying factors. Consideration of factors such as: size of flood; effective warning (and response) time; flood readiness (of the community); rate of rise of floodwaters; duration of flooding; evacuation problems; effective flood access; and type of development, may all cause the assessed true hazard categorisation to be different to the preliminary hazard initially derived from analysis of the depth and velocity of floodwaters alone. As flood hazard is determined from consideration of model results, it can be influenced by the modelling approach, assumptions and parameters used to determine upon which hydraulic behaviour it is based.

Under current practice, development is generally restricted from areas of high flood hazard. It is therefore important that the spatial distribution of flow velocities across the floodplain are modelled appropriately so that areas of high and low hazard can be properly delineated. When either the depth or velocities are considerable, the stability of buildings and/or public safety is threatened. Flow velocity also plays an important role when deciding evacuation routes (especially where crossing floodways), as people and vehicles may be swept away if the velocities are too high.

Retallick and Babister (2008) showed that a number of commercially available software packages underestimated hydraulic hazard on the overbank (up to 50% for the test cases used in the study) The results of the study indicated the potential for significant risk to the community as hydraulic hazard assessments often determine the proximity of development to watercourses.

Syme (2008) and Smith and Wasko (2012) show that the method of schematisation for individual buildings can have a significant localised effect on velocity-depth relationships in and around buildings and other structures. Refer to Chapter 11 for more information.
There is an Australian Rainfall and Runoff revision project devoted to actively revising some of the curves used to define the hazard arising from velocity and depth. Readers are referred to the results and final report from ARR Revision Project 10 for more information.

8.5.4. Floodplain Function

An important and powerful use of 2D models is for classifying the hydraulic function of a floodplain for landuse planning purposes. Velocity and velocity-depth product are the key inputs for this process. It is common practice in Australia to classify the parts of the floodplain which are important for flow conveyance (often called floodways) and those that are important for flood storage. The calculation of floodplain function is a complex task that requires some interpretation by the modeller. Readers are referred to the Victorian Flood Management Strategy and the NSW Floodplain Development Manual.

8.5.5. Evacuation Planning

In an effort to reduce the risk to life of residents and emergency workers, considerably more effort has been put into flood evacuation planning in recent times. 2D hydraulic modelling results can be used to classify those areas that become “low flood islands” and to plan the sequence of evacuation in different size flood events. Readers are referred to NSW flood evacuations guidelines for advice.

8.5.6. Result Filtering

Raw model results should not be unconditionally accepted at face value. They may sometimes require inspection and post-processing to identify and filter out anomalies, based on the judgement of the modeller. However, this should only be undertaken when justified. For example it is usually justified to remove start up noise, but instabilities in the model should be fixed rather than filtered. Details of any filtering that has been undertaken should be fully documented in terms of type, location and reasoning.

8.5.7. Direct Rainfall Issues

The use of direct rainfall on a model grid/mesh raises new challenges in modelling. Further details on the use of direct rainfall and related issues are detailed in Chapter 11. As every 2D element is wet in a direct rainfall model, it can be difficult to interpret results, such as flood extent. The objective of filtering direct rainfall results is to convey the important information. Again, full documentation of the filtering undertaken should always be kept and presented.

8.5.8. Conceptualisation and Schematisation Effects

Model results can often contain results which may be “correct” in that they are consistent with the conceptualisation or schematisation approach adopted, and give a good approximation to general flow behaviour, but may have local flow effects which are not suitable for reporting. Model features at which detailed results may be unreliable are:

- nested boundaries;
- levees;
- structures;
- immediate overbank of defined channel flow;
- study area boundaries;
- fences / walls;
- supercritical flow; and
- flow transition areas.

8.6. References

AS/NZS ISO 31000:2009 Risk management - Principles and guidelines, ISBN 0-7337-9289-8, 20 November 2009.

CSIRO, 2000, Floodplain management in Australia – Best Practice Principles and Guidelines, SCARM Report 73 , CSIRO Publishing

DSE, 1998, Victorian Flood Management Strategy

Engineers Australia, 2010, Australian Rainfall and Runoff revision Project 10- Appropriate safety criteria for people.

NSW Government, 2005, NSW Floodplain Development Manual

NSW Government, 2007, Floodplain Risk Management Guideline - Flood Emergency Response Planning- Classification of Communities

Smith and Wasko (2012), Australian Rainfall and Runoff Revision Projects - PROJECT 15 Two Dimensional Modelling in Urban and Rural floodplains: *Representation of Buildings in 2D numerical Flood Models,* P15/S2/023, Oct 2012

Syme W.J. (2008), Flooding in Urban Areas - 2D Modelling Approaches for Buildings and Fences Engineers Australia, 9th National Conference on Hydraulics in Water Engineering, Darwin Convention Centre, Australia 23-26 September 2008.

Retallick and Babister (2008), Comparison of Two-dimensional modelling approaches used in current practice, *9th National Conference on Hydraulics in Water Engineering*, Darwin Convention Centre, Australia 23-26 September 2008.

9. CHAPTER 9 – 1D/2D AND 2D/2D COMBINATIONS

9.1. Introduction

Combining 1D and 2D model domains is undertaken in order to benefit from the different strengths of both domain types (individual strengths of these domains are discussed further in Chapter 2). 1D domains are generally utilised to define sub-grid scale features, such as creeks and hydraulic structures, or where the 2D domain grid does not accurately define the conveyance. Because of this, the 2D models discussed in this section are usually fixed grid models, as flexible mesh models are generally able to replicate finer details by utilising finer mesh in relevant areas. As such, terminologies used in this chapter only relate to 2D fixed grid models. The following table outlines the typical application of 1D domains and 2D domains in a 1D/2D model. However, it should be noted that this list is by no means prescriptive, as it depends on the modelling system and the specific application.

Table 9-1 Typical Application of 1D and 2D domains

Floodplain Feature	Domain	Comments
In-bank Channel Flow	1D	Typically, a 1D domain is used to define the in-bank channel flow. There are a number of reasons for this: • Data are generally provided in a cross-sectional format, and is therefore suited to the application of 1D domains. • 2D domains would generally require significantly smaller grid cell sizes in order to adequately define in-bank flows, and hence be slower to run. • In-bank flow is generally one-directional in nature (upstream or downstream direction) and so generally, is adequately defined in a 1D domain.
Hydraulic Structures	1D	Hydraulic structures, such as culverts and bridges, are generally defined in the 1D domain of the model. These types of structures are typically governed by empirical based relationships, which can be difficult to reproduce in a 2D model.
Pits & Pipes	1D	Pits and pipes, which represent a naturally separated system from the overland flow component (i.e. underground) are modelled in the 1D domain. These structures are inherently 1D in nature.
Overland Flow	2D	Out-of-bank, overland flow can be difficult to define in a 1D domain, particularly where the flow direction is ill-defined. In these situations, a 2D domain has significant advantages over a more traditional quasi-2D approach (refer

Floodplain Feature	Domain	Comments
		Chapter 2).
Weirs	1D or 2D	The definition of weirs can depend on the modelling software used. Some modelling software are capable of reproducing weir behaviour in the 2D domain of the model, while others effectively implement a 1D relationship across a specified number of grid cells.

9.2. Connection of 1D and 2D

The connection of 1D and 2D model domains is dependent on the modelling software package and the application. There are small differences in approach between software packages. However, it is important to note that currently, in the majority of software packages, only the water level, or water level and flow, is transferred between the two model domains. The velocity (and thus momentum) is not transferred. Some schemes preserve momentum by assuming the velocity is unchanged across the 1D/2D interface, instead of applying the flow as a source (flow with no inertia or direction). A lack of momentum transfer has implications on the schematisation of 1D/2D modelling applications. For example, a connection between 1D and 2D domains should generally not be made in an area where the velocity transfer across this connection is important. If momentum were important then a scheme that preserves momentum across the interface would be required.

The following section outlines some of the methods utilised in the connection of 1D and 2D domains in current modelling software. In each of these sections, the challenges of momentum transfer are highlighted.

9.2.1. Connection of Channel / Creek Flow

There are currently two methods by which channel flow is transferred between a 1D domain and a 2D domain: through a vertical connection or through a horizontal connection. Both of these approaches have advantages and disadvantages, and modellers should be mindful of the limitations when applying these approaches. Currently, most modelling packages only allow the application of one of these connection techniques.

9.2.1.1. Vertical Connection

With a vertical connection, water is transferred between the 1D and 2D domains when the flow exceeds a pre-defined top of banks. Therefore, only flow within the banks and below top of bank is conveyed in the 1D domain of the model, as shown in Figure 9-1.

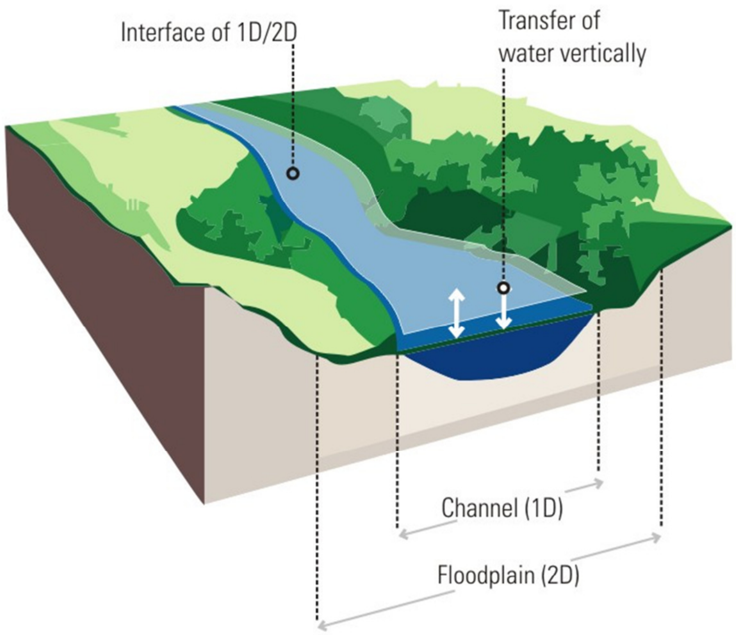


Figure 9-1 Vertical transfer of flow between a 1D and 2D domain

One of the limitations of this approach occurs when there are a number of 2D domain grid cells between the top of banks. For modelling software that connects to just one grid cell above the channel, this results in a large quantity of water transferred into the 2D domain from a large portion of the 1D domain channel underneath. To overcome this limitation, some modelling software is able to connect the 1D domain to all 2D domain grid cells within the top of banks. However, the drawback with this approach is that increasing the number of connections increases the computational run times.

When the water level exceeds the top of banks in the case where a relatively large grid cell size is utilised (i.e. large in comparison to the width of the channel), flow proceeds over a much larger width than the base channel. This size cell may well represent the overbank flow component, but modellers should be mindful that velocities are averaged across the entire grid cell, and therefore may not be representative of the velocities across the top of the channel.

Flow proceeding across the top of the channel in the 2D domain only, interacts with the 1D domain flow via the exchange of water level. If the flow across the top is proceeding in the same direction as flow in the channel, the model roughness in the 2D domain directly over the channel may be lower than the actual channel roughness, because it is actually representative of the top layer of flow, rather than the entire depth of the channel. However, if the flow across the top is proceeding in a different direction to flow in the channel (e.g. perpendicularly), the roughness assumptions are far more complex. Given the absence of momentum transfer between the 1D and 2D domains, providing definitive advice on the variation of roughness values between the 1D and 2D domains across the channel would be misleading. Instead, the modeller is advised to be mindful of the complexities involved in this situation and sensitivity test assumptions made.

9.2.1.2. Horizontal Bank Connection

In a horizontal bank connection, the 2D domain cells directly above the channel are effectively nulled, with the 1D domain of the model defining all flow between the banks. The connection then occurs horizontally along the top of bank, as shown in Figure 9-2.

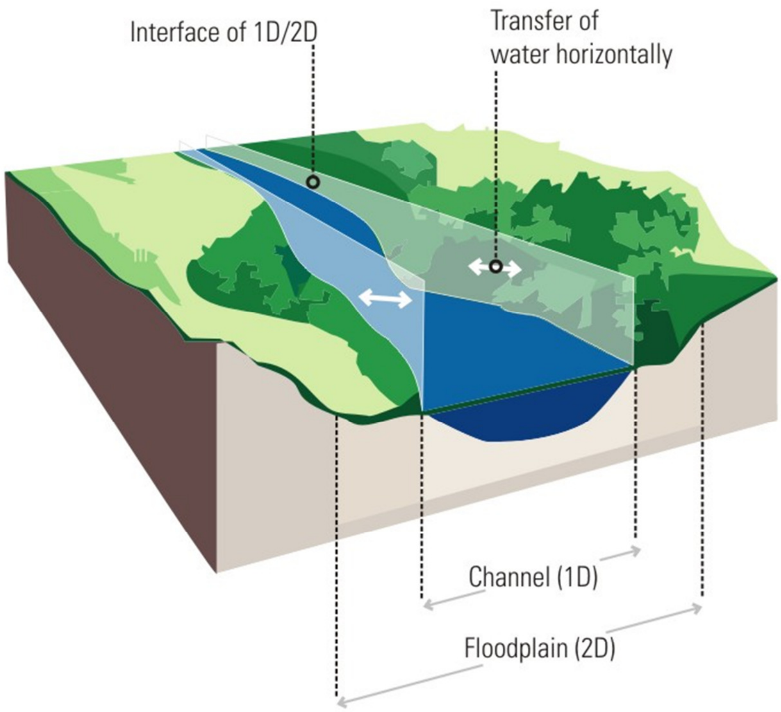


Figure 9-2 Horizontal transfer of flow between a 1D and 2D domain

One of the limitations of this approach occurs when the channel width is not a multiple of the grid width. In some modelling packages, this may result in a loss in volume across the floodplain, as demonstrated in Figure 9-3. This effect becomes more pronounced when the grid cells are large by comparison to the width of the channel. It is therefore important that the 1D channel cross-section width is similar to the 1D/2D boundary width or flow width of nulled 2D cells.

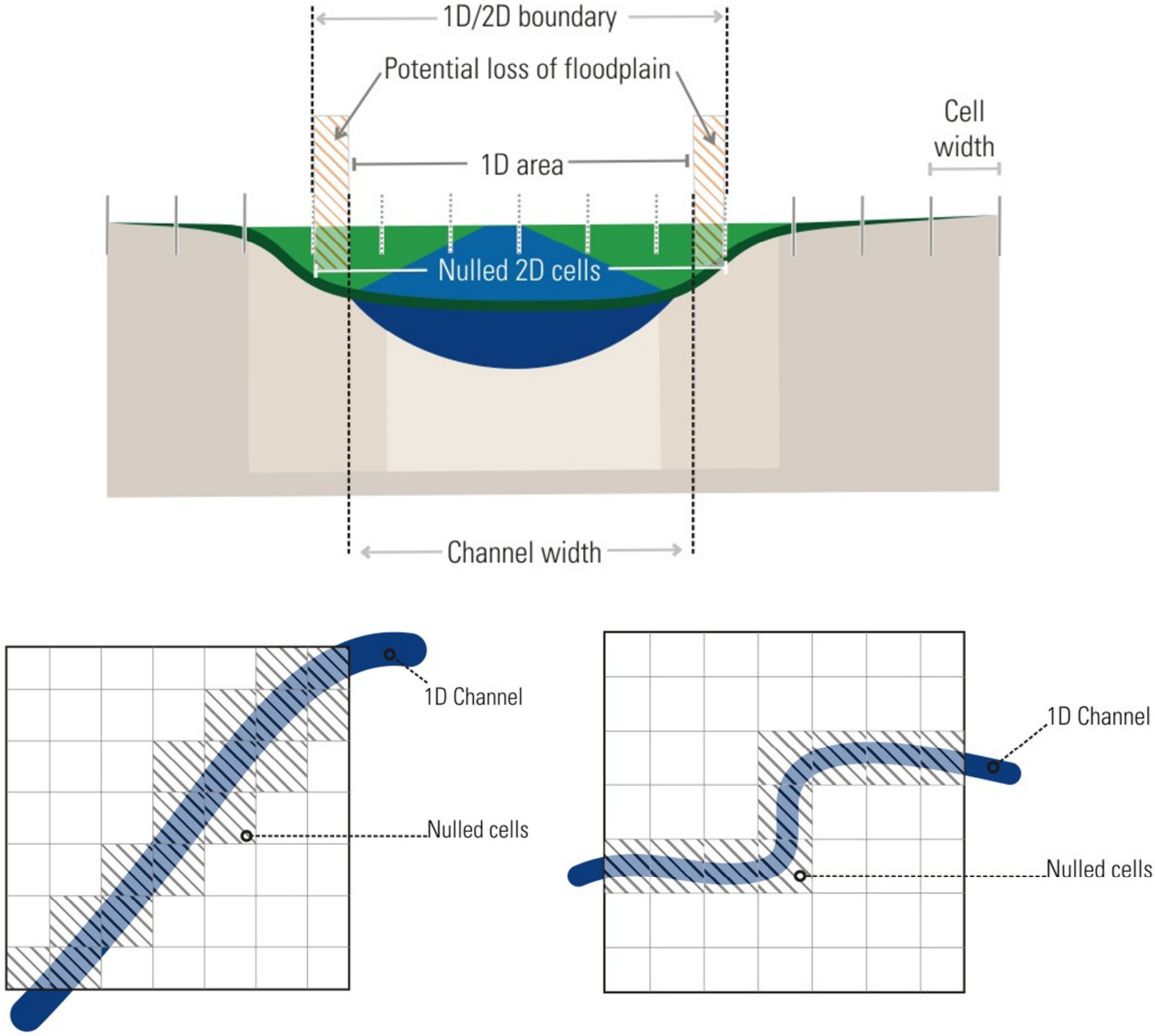


Figure 9-3 Potential loss of floodplain width in a Horizontal 1D/2D Connection

Another limitation of this approach occurs when flow behaviour is two-dimensional across the channel. For example, in a meandering creek system, low flows may be confined to the creek banks and proceed in one direction, while larger flows may proceed in a different direction, as shown in Figure 9-4. This behaviour can be difficult to recreate using the horizontal form of connection if momentum is not transferred once large flows proceed past the top of banks. Schemes with momentum preservation across the 1D/2D interface are capable of reproducing this effect and are preferred if using horizontal 1D/2D connections as illustrated in Figure 9-5.

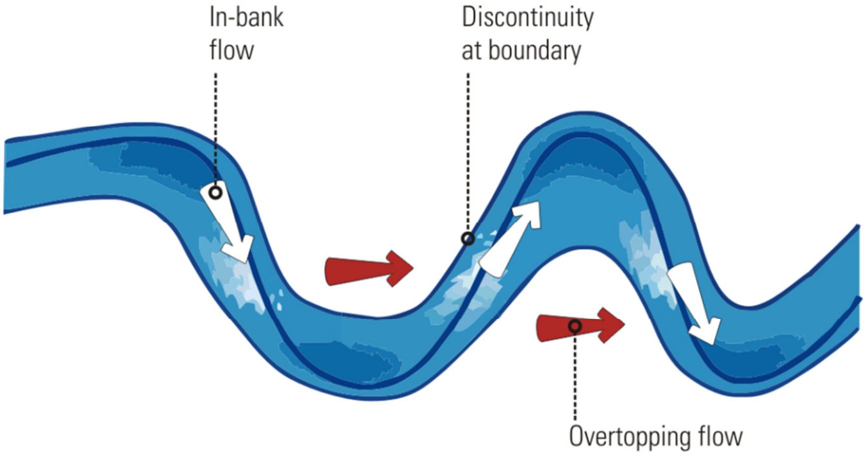


Figure 9-4 Two-Dimensional Flow Across a Channel

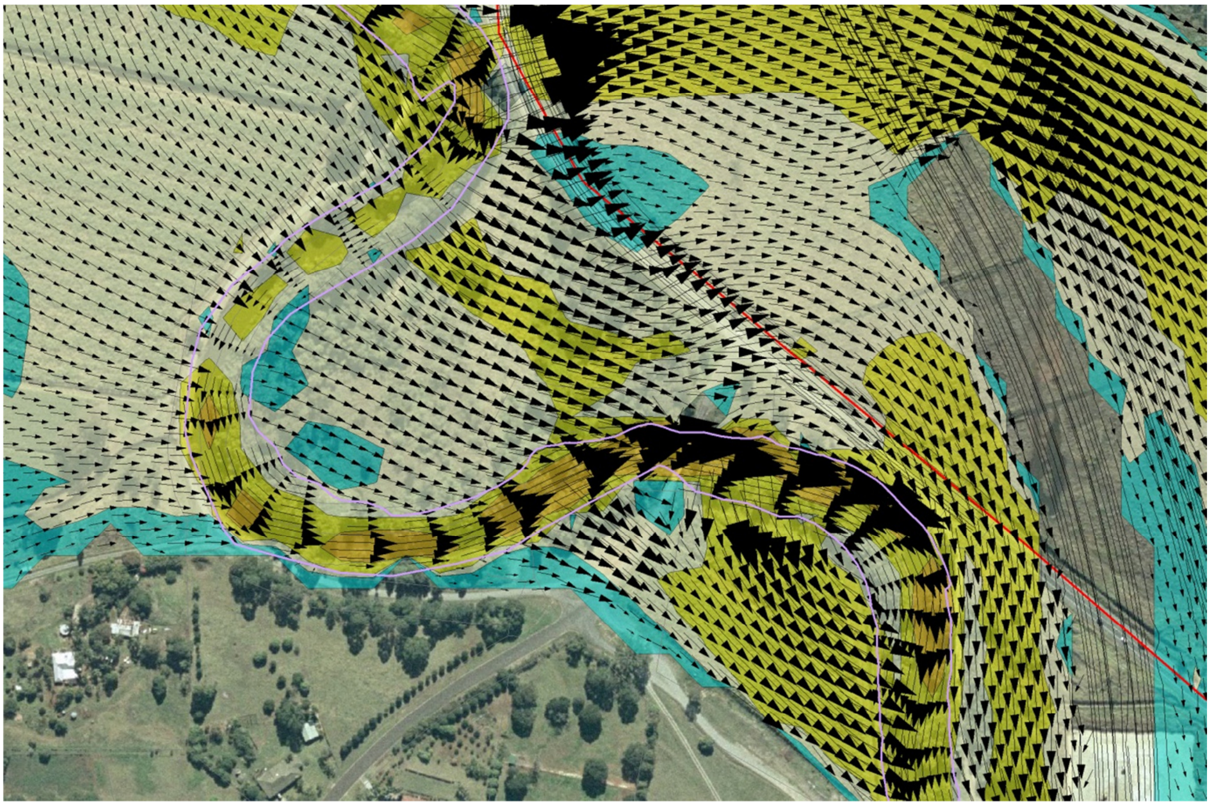


Figure 9-5 Example of 1D/2D Flow Patterns across a Meander using a Horizontal Connection with Momentum Preservation

9.2.1.3. Additional Connection Options

Some modelling packages allow levees or high banks to be incorporated into the 1D/2D connection. For example, a vertical connection typically transfers water once the water level reaches the level of the 2D domain. However, if a levee or high bank is defined in the 1D domain, the water will not be transferred until this levee level is exceeded, as shown in Figure 9-6.

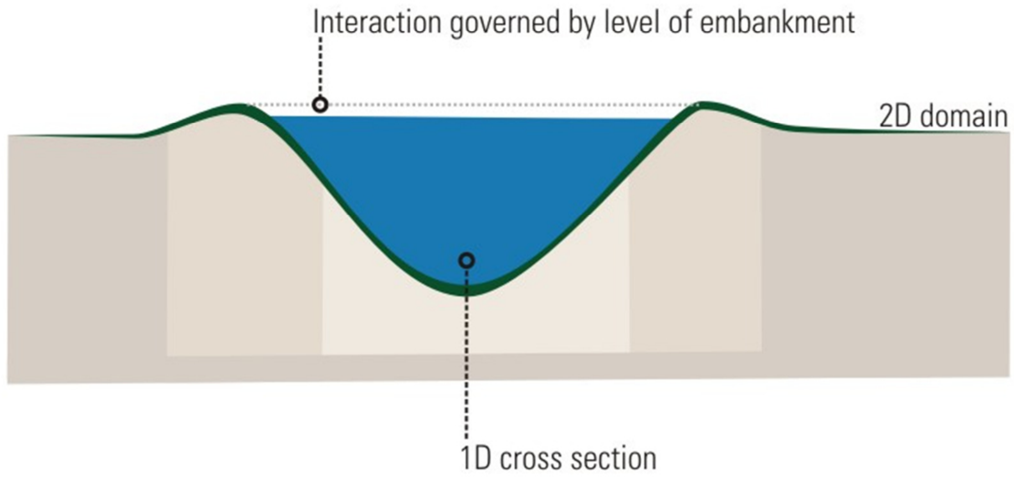


Figure 9-6 Vertical Connection – Levee Option

9.2.1.4. Summary

Table 9-2 provides a brief summary of some of the scenarios discussed above, and how each connection method handles these scenarios. It should be noted that some of the modelling packages employ methods to overcome these limitations, and that both methods have advantages and disadvantages.

Table 9-2 Disadvantages of 1D/2D Connection Types for Various Scenarios

Scenario	Horizontal Connection	Vertical Connection
Grid cell is larger than channel	Potential loss of 2D floodplain volume and conveyance if 1D cross-sections not correctly sized.	Velocity results represent average over cell, rather than the in-bank velocities.
Grid cell is smaller than channel width	Loss of 2D floodplain volume and conveyance become less pronounced with smaller grid cells.	When the horizontal connection is made through a singular grid cell, potential issues arise through the transfer of all the water from a larger channel through a singular smaller grid cell.
Opposing 1D and 2D velocities (such as the meandering creek example)	If no momentum preservation or transfer of velocity component across 1D boundary the horizontal connection can result in the effective discontinuation of velocity	Effective separation of 1D and 2D velocities, allowing opposing velocities. However, no interaction of these opposing velocities.

9.2.2. Connection of Pits & Pipes

The connection of pits and pipes with the 2D domain of the model is generally simpler than channel connections. This is because the pits, by their nature, are effectively a point discharge source.

Most models will connect with a pit network vertically, as shown in Figure 9-7. For discharge from the 1D to the 2D, water builds up within the pit itself, and discharges (potentially through a stage-discharge relationship) once it reaches the street level. Velocity is generally less of an issue in this form of connection and thus the lack of momentum transfer is less of a concern. For water to enter the 1D domain, surface flow within the 2D portion of the model will effectively “drop” into a the 1D pit, potentially through a stage-discharge relationship.

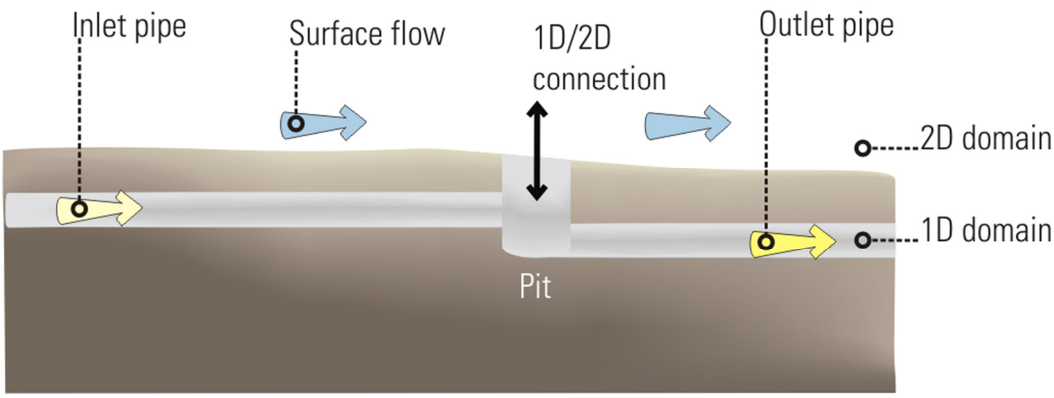


Figure 9-7 Pits and Pipes Connection to 2D Domain

9.2.3. Connection of 1D Hydraulic Structures

Hydraulic structures can be modelled as an element of a 1D domain or as a discrete 1D element within a 2D domain. This section primarily refers to hydraulic structures that are modelled as a discrete 1D element within a 2D domain. Discretely modelled hydraulic structures might include, for example, a series of culverts under a road embankment across a floodplain.

In general, 1D hydraulic structures are governed by empirical relationships, which therefore result in a discontinuity in the fully dynamic equations utilised in most modelling systems. When they are connected directly to the 2D domain, flow is conveyed via an upstream grid cell through the empirically derived hydraulic equation to the downstream cell. As the velocity component of the flow is not conveyed across the 1D/2D connection, the hydraulic structure equation will only be influenced by the upstream and downstream water level. This may lead to an unrealistic representation of afflux at the structure.

One of the challenges for modellers when including discretely defined 1D hydraulic structures within the 2D domain is the definition of contraction and expansion losses. Most empirical hydraulic equations will incorporate the losses in the form of coefficients. When the 2D grid cell is relatively large in comparison to the hydraulic structure, then the losses can be calculated using the empirical relationships of the 1D hydraulic structure. However, when the 2D grid cell becomes relatively small in comparison to the hydraulic structure, some of the losses will be effectively modelled in the 2D domain. In this scenario, it may be necessary to adjust the loss coefficients in the 1D hydraulic structure equations to reduce the losses, otherwise the losses may potentially be duplicated.

As with most applications of hydraulic modelling, the hydraulic behaviour of the structure should always be verified through alternative methods to ensure that it is represented correctly.

9.2.4. Transfer of 1D to 2D, and 2D to 1D

There are a number of occasions when it may be appropriate for a fully 1D domain to transfer to a fully 2D domain. For example, as shown in Figure 9-8:

- Transfer of 2D floodplain flows to a 1D hydraulic structure;
- A 1D creek discharging into a large waterway or lake;
- A steep 1D creek section which discharges into a relatively flat floodplain.

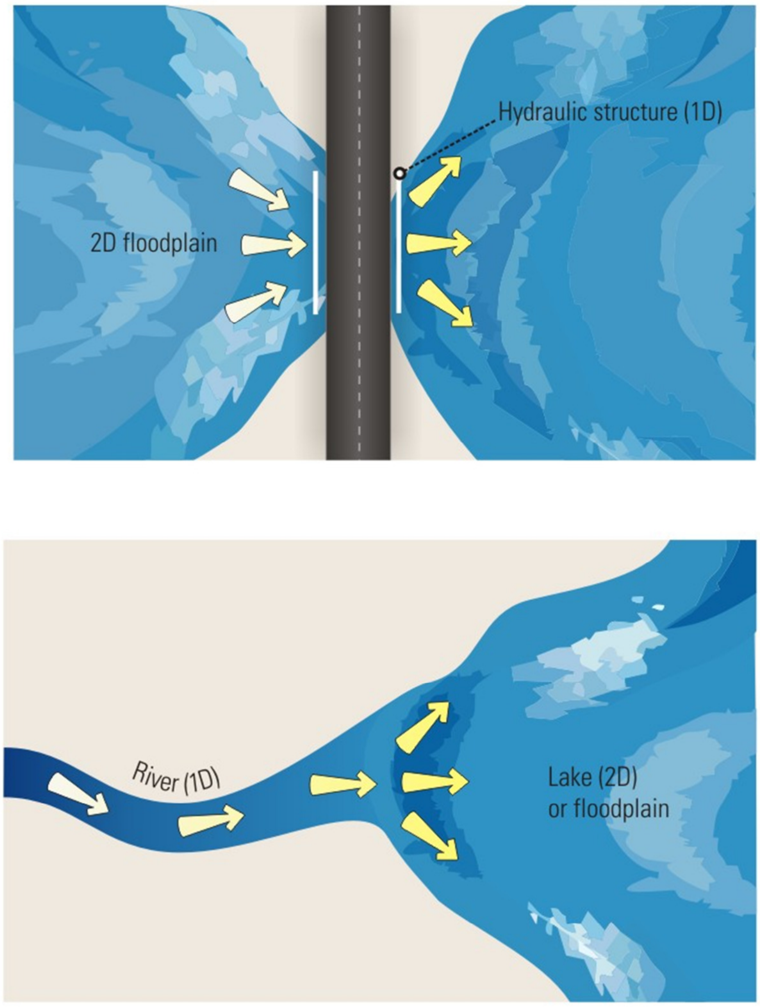


Figure 9-8 Examples of Complete Domain Transfer

One of two approaches is generally adopted for these scenarios:

- The connection between the 1D and the 2D domains is made through an individual grid cell connection. This approach is appropriate when the grid cell size is larger or approximately equal to the width of the channel. When the grid cell is smaller, this can unrealistically restrict conveyance into or out of the 2D domain. For example, a 20 metre wide 1D channel discharging into a singular 5 metre grid cell will provide a constriction that is not realistic.
- The connection between the 1D and the 2D domains is made through a number of 2D grid cells. In this case, the number of cells over which the connection is made will have a similar conveyance to that in the 1D domain. For example, a 20 metre wide 1D channel is connected to four 5 metre grid cells.

It is important to note that with the lack of momentum transfer, this type of connection is limited in its representation of the physical system. For example, if momentum is not transferred or preserved across the 1D/2D connection when a steep section of creek discharges to a lake, unrealistic flow behaviour may result. The flow patterns in Figure 9-9 illustrate the effect of different approaches to modelling a hydraulic structure.

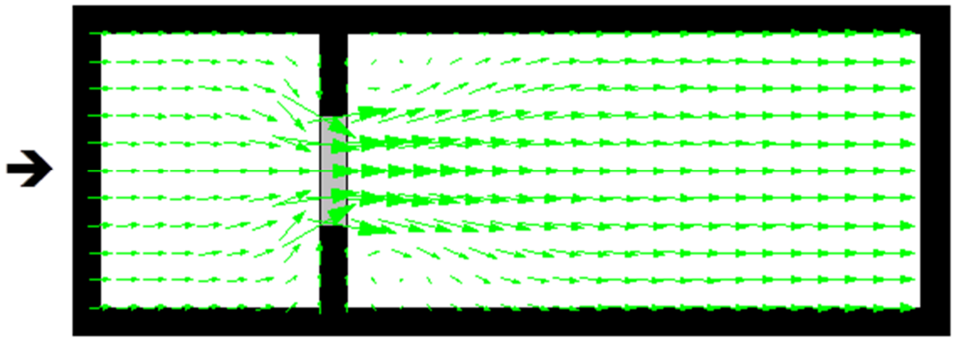
9.3. Parent – Child Grids

Child grids are used to provide a finer resolution over a region of interest that exists within the larger, coarser 2D parent grid. Child grids achieve a better definition of the area interest by utilising a smaller grid cell. For example, a child grid with a 5 metre grid size may be nested in a parent grid with a 50 metre grid size. Child grids are typically utilised to increase the efficiency of models. For example, a rural area is defined in a coarser grid while a township, which may require additional definition, is defined using a child grid.

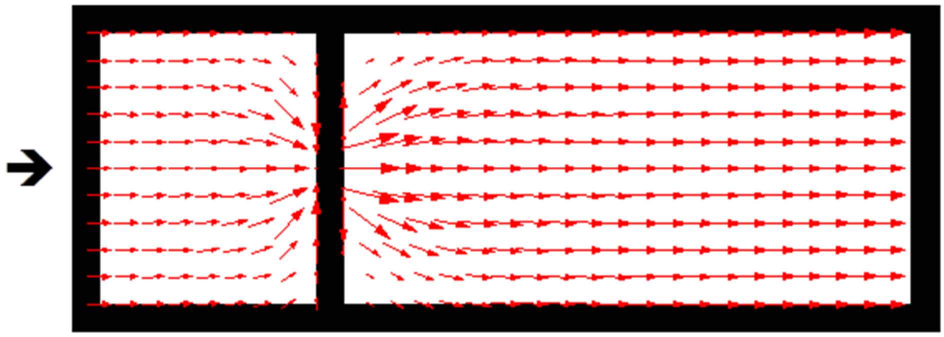
Parent grids and child grids are connected in a similar nature to the 1D/2D connections. Each individual cell along the boundary of the child grid domain conveys flow to the parent grid cell at that boundary. Modelling packages utilise a number of different options in the transfer between grids, with some transferring the momentum while others rely on water level only.

Modelling packages that do not transfer momentum have similar limitations to those discussed in Section 9.2.1. Some modelling packages do attempt to transfer momentum. However, the assumptions made in the transfer between the different sized grid cells can produce challenges themselves.

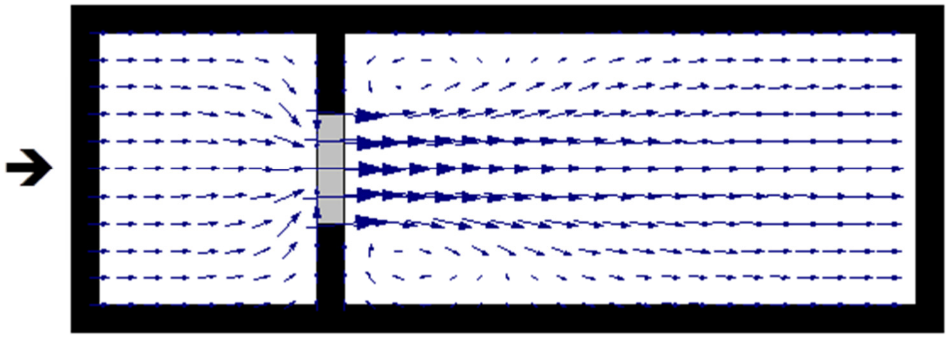
In general, it is recommended that parent-child grid boundaries be placed a sufficient distance from the study area such that any inaccuracies or limitations potentially associated with these boundaries do not affect flooding behaviour in the area of interest. Alternatively, the boundary should be located away from high velocity zones.



Flow patterns using a fully 2D solution



Flow patterns using a 1D element connected to 2D with no momentum preservation



Flow patterns using a 1D element connected to 2D with momentum preservation

Figure 9-9 Different Flow Patterns when Modelling a Submerged Culvert using 2D and 1D/2D Approaches

10. CHAPTER 10 - CURRENT ISSUES

10.1. Introduction

There are a number of areas within the field of 2D modelling practice that are identified as emerging areas, or as current issues requiring special attention. These areas are described separately within this Chapter to provide a quick and direct source of reference. Some of the issues contained within are recommended areas for further research (refer to Chapter 12). These topics are evolving fields and current practice may have evolved past the practice described herein. The reader is referred to the software manuals for how a particular software package incorporates/handles these issues.

10.2. **Modelling Super-Critical Flows in 2D Flood Models**

10.2.1. Background

As described in Chapter 3, the 2D hydrodynamic models used in flooding investigations are generally based on the numerical solution of the two-dimensional shallow water wave equations. These equations describe the depth-averaged conservation of mass and momentum in two horizontal dimensions. The equations and the numerical solution procedures used to solve them are well suited to describing sub-critical flows. They are not, however, as well suited to modelling super-critical flows.

Fortunately, most of the flow situations to which 2D flood models are applied relate predominantly to sub-critical flows. Nevertheless, there can be many situations where the flow can become super-critical. These can include flows down steep grades, and locally accelerated flows at obstructions and structures. Under these conditions, it is important that the model be able to maintain the computation and provide reasonable results throughout the region of super-critical flow. Additionally, there is the tendency for many modellers to "push" the limits of their modelling software and to use it to model flow situations for which it was never intended. It is therefore important that modellers have an understanding of the limitations of 2D flood models when applied to super-critical flows.

10.2.2. Issues

There are three main issues related to modelling super-critical flows in most of the currently available 2D flood models. These are:

- The suitability of the equations of motion forming the basis of the model
- The suitability the numerical method being used to solve the equations
- The need to provide appropriate dissipation in super-critical to sub-critical flow transitions

These issues are discussed in more detail below. This is followed by a general discussion on the suitability of 2D flood models for modelling super-critical flows.

10.2.2.1. Equations of Motion

Three of the key assumptions used in the derivation of the shallow water wave equations are that:

- The flow is two-dimensional (i.e., the effects of variations in the vertical flow velocity profile can be neglected).
- The pressure is hydrostatic (i.e., streamline curvature is small and vertical accelerations can be neglected).
- The flow is nearly horizontal (i.e., the average channel bed slope is small).

Thus, locations where super-critical flow is likely to occur (i.e., at steep slopes and areas of localised accelerations) are the locations where the underlying assumptions to the equations may become invalid. However, in practice, it has been found that the equations can provide good reproduction of flows down quite steep slopes, including dam-break scenarios, provided the rate of change in slope is not too great. Despite this, care should be taken when modelling flows down very steep slopes and in areas where there are rapid changes in bed slope.

In areas where there are rapid changes in bed slope, the assumption of hydrostatic pressure can become invalid. As an example, when water flows over an embankment the initially horizontal flow across the top of the embankment will accelerate downwards, thereby reducing the effects of gravity. A numerical model solving the shallow water wave equations cannot resolve the effects of the downward acceleration, and will still apply the full hydrostatic pressure gradient down the face of the embankment. As a result, 2D flood models can over-estimate flows across embankments, levee banks and weirs where they are modelled directly within the model grid.

As an example, it is noted that McCowan *et al* (2000) found that this over-estimation of flows could result in the surface elevation of the water computed behind a levee bank as being 30% lower than that predicted by standard broad-crested weir formulae. This is the reason why most of the commercially available 2D flood models incorporate structure equations to provide a better estimate of the flows in these situations (refer to Chapter 6 and 9).

10.2.2.2. Numerical Solution Procedures

The main problem with modelling super-critical flows is that the numerical procedure required to obtain solutions to the shallow water wave equations is very different to that required for sub-critical flows.

For sub-critical flows, flow information can propagate upstream as well as downstream. As a result, a boundary condition is required along both the upstream and downstream boundaries of the model. As discussed in Chapter 3, these boundary conditions can be provided in terms of either water levels or velocities.

For super-critical flows, no information can travel upstream and the local flow at any point is entirely dependent upon the upstream flow conditions. As a result, a super-critical flow model requires two upstream boundary conditions, in terms of both a water level and a velocity (given as a speed and direction, or as x and y velocity components), but no downstream boundary condition.

As discussed above, the flow in most 2D flood applications is predominantly sub-critical, with super-critical flows restricted to localised areas in the vicinity of steep slopes and locally accelerated flows at structures and obstructions. Ideally, the numerical solution procedure should be able to adapt from the requirements of sub-critical flow to those of super-critical flow, and vice-versa, as required. In practice, however, the regions of super-critical flow vary in both time and space, and this approach is impractical for most modelling situations.

As an alternative to introducing a true super-critical solution procedure for regions of super-critical flow, most 2D flood models use numerical tricks to allow the sub-critical flow solution procedure to continue computing through any regions of super-critical flow. In this respect, it is noted that the main problems associated with modelling super-critical flows are:

- The solution procedure is "ill-conditioned" (i.e., it is not suitable for super-critical flow and does not have the necessary internal boundary conditions), which would normally lead to instability.
- The high velocities that tend to occur with super-critical flow will exacerbate any pre-existing tendency towards instability in modelling of the convective momentum terms.
- Dissipation needs to be introduced into the model to allow for turbulent losses in hydraulic jumps where the super-critical flow reverts back to sub-critical flow.

As discussed in Chapter 3, the use of time and space-centred (non-dissipative) approximations to the non-linear convective momentum terms in the momentum equations, can lead to unrealistic oscillations and local instabilities in situations where the transport properties of the flow become important. This can occur commonly even with sub-critical flow conditions. With super-critical flows instability is virtually guaranteed.

To overcome problems with modelling the convective momentum terms, some early modellers "switched off" or reduced these terms in the momentum equations as the Froude number increased and super-critical flow conditions were approached. This had the effect of "linearising" the momentum equations, which in turn reduced the numerical effects of the high velocities and helped maintain stability. It also reduced the effect of the "velocity head" associated with the flow, thereby affecting the models ability to model flow separations and circulations. Additionally, simply switching off the convective momentum terms does not, in itself, overcome the problem of ill-conditioning of the solution procedure. As a result, this approach has been generally discontinued.

Another approach is to use simplified forms of the momentum equations for modelling super-critical flows. In this respect, it is noted that the kinematic wave approximation is sometimes used to describe flow conditions in regions of super-critical flow (e.g., Syme, 2001). This approach is reasonable for most flooding applications as super-critical flows are upstream controlled and are normally friction dominated. However, whenever simplified forms of the equations are used, it is important for the user to understand their limitations, and to be aware that care may be required in interpreting the results, particularly in transition areas.

10.2.2.3.The Role of Dissipation

Another early approach used was to introduce numerical dissipation into the solution procedure. For example, the Lax (1954) scheme filtered the velocities used in the main $\partial u / \partial t$ local acceleration term. This effectively introduced an additional $E\left(\partial^2 u / \partial x^2\right)$ "eddy viscosity" type term, with an effective eddy viscosity coefficient of $E = \Delta x^2 / 2\Delta t$. That is, the effective eddy viscosity coefficient becomes a function of the grid size Δx and the ratio of grid size to time step $\Delta x / \Delta t$. It is noted that this numerical eddy viscosity coefficient can be orders of magnitude greater than physically realistic values, and that the numerical dissipation introduced in this way, is frequency dependent, and has its greatest effect on high frequency oscillations. It was found that the high level of numerical dissipation introduced in this manner was adequate to maintain the computation through a region of super-critical flow and, as a result, Abbott (1979) noted that the "Lax scheme" has often been used for modelling hydraulic jumps and super-critical flows.

Following on from this, the main approach used in many of the current 2D models is to introduce numerical dissipation into the numerical solution procedure. This can be done generally by changing the solution procedure throughout the entire model (e.g., Stelling *et al*, 1998), or selectively by changing the solution procedure at locations where the Froude number is high and the flow is super-critical or approaching super-critical conditions (e.g., McCowan et al, 2000). The dissipation can be introduced by filtering the velocities and/or water levels at the old time step (as for the Lax scheme), by "forward weighting" spatial derivatives in time, by up-stream weighting (or "up-winding") spatial derivatives in space, or combinations of these.

The different approaches all have the effect of introducing additional numerical dissipation through "eddy viscosity" type diffusion terms with effective eddy viscosity coefficients in terms of the grid size Δx, the time step Δt, or a combination of both.

One of the more common approaches is to use first order "up-wind" differencing for the $u.\partial u / \partial x$ type "convective momentum" terms. With this approach, it can be shown (e.g., McCowan et al, 2000) that $u.\partial u / \partial x$ becomes equivalent to:

$$u\frac{\partial u}{\partial x} \approx u\frac{\partial u}{\partial x} \quad -\frac{u\Delta x}{2}\frac{\partial^2 u}{\partial x^2} + O\left[\Delta x^2\right]$$

That is, the up-winding introduces a numerical eddy viscosity type diffusion term with an effective eddy viscosity coefficient of $E = u\Delta x/2$. As for the filtering, considered above, it is noted that the numerical eddy viscosity coefficients may be orders of magnitude greater than physically realistic values (Leonard, 1974), and that the numerical dissipation introduced in this way has its greatest effect on high frequency oscillations.

Figure 10-1 shows the results from a test case presented by McCowan et al (2000). The test case considered flow over an initially flat bed, changing to a slope of 1 in 100, and then 1 in 50. The flow velocity over the 1 in 100 slope was about 2.0 m/s, giving a Froude number of just over 0.9. The flow over the steeper 1 in 50 section was super-critical. The figure shows that, without up-winding of the convective momentum term (Normal), the computation goes unstable in the super-critical flow region. By comparison, it can be seen that up-winding the convective momentum terms (Up-winded) allows stable computation to continue throughout the region of super-critical flow.

The main reason the super-critical flow solutions remain stable in the up-winded system is that the numerical eddy viscosity introduced puts a greater emphasis on the diffusion properties of the momentum equation relative to the propagation properties (discussed in Chapter 3). In this respect it is noted that solution of the diffusion properties of the long wave equations requires a single upstream and a single downstream boundary condition. This is consistent with the numerical solution procedures used for sub-critical flows. Thus, provided the numerical diffusion introduced through up-winding is greater than the destabilising effect of the ill-conditioning of the solution procedure for solving the propagation properties, sub-critical flow solution procedures can continue to be used to provide numerical solutions in regions of super-critical flows. This is the basis of the success of the numerical solution of the super-critical flow case presented in Figure 10-1.

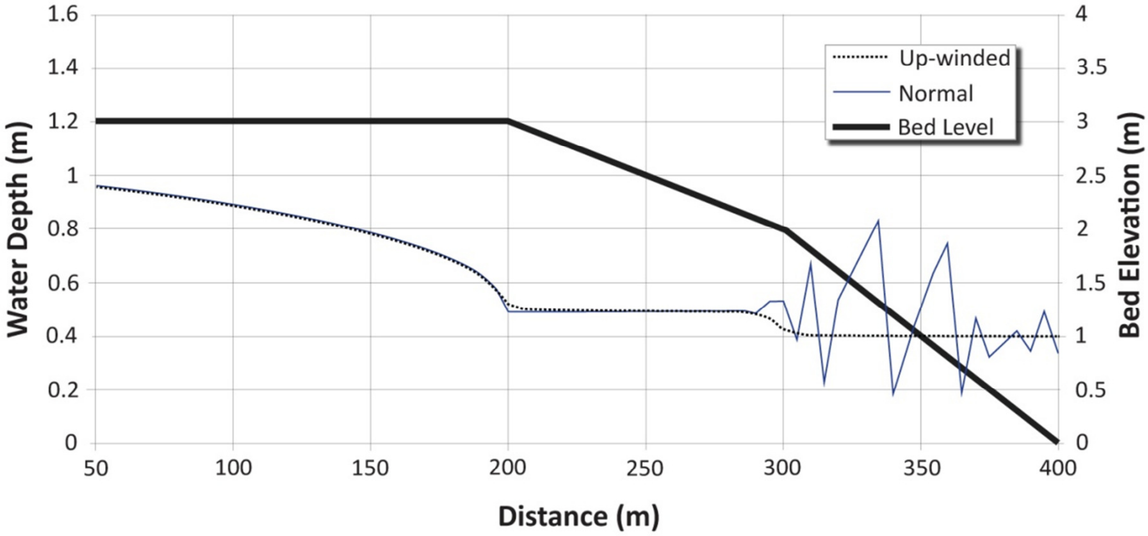


Figure 10-1 The Effect of Up-Winding the Convective Momentum Terms on Super-Critical Flows

In many cases, the numerical diffusion introduced through up-winding of the convective momentum terms is also sufficient to represent the energy dissipation required to maintain a stable numerical solution through transitions from super-critical to sub-critical flow. That is, it also allows us to maintain approximate solutions through a hydraulic jump. An example is presented in Figure 10-2. Here the flow has become super-critical over a levee bank. There is a hydraulic jump at the base of the levee as the flow becomes sub-critical. The flow then transitions smoothly to super-critical again as if flows down a steepening slope.

10.2.3. Recommendations

The most important thing to be remembered in modelling super-critical flows is that the current range of 2D hydrodynamic models do not include rigorous solution procedures for solving super-critical flows. Instead, they use simplified forms of the equations and/or numerical dissipation to maintain the numerical solution through localised regions of super-critical flow. This can provide useful information on the main properties of the flow, but will not be able to provide precise details of the flow, or reliable information of the location and dimensions of hydraulic jumps associated with the transitions back to sub-critical flow. Model users should be aware of the approach used in their model, and of any particular limitations that this may introduce.

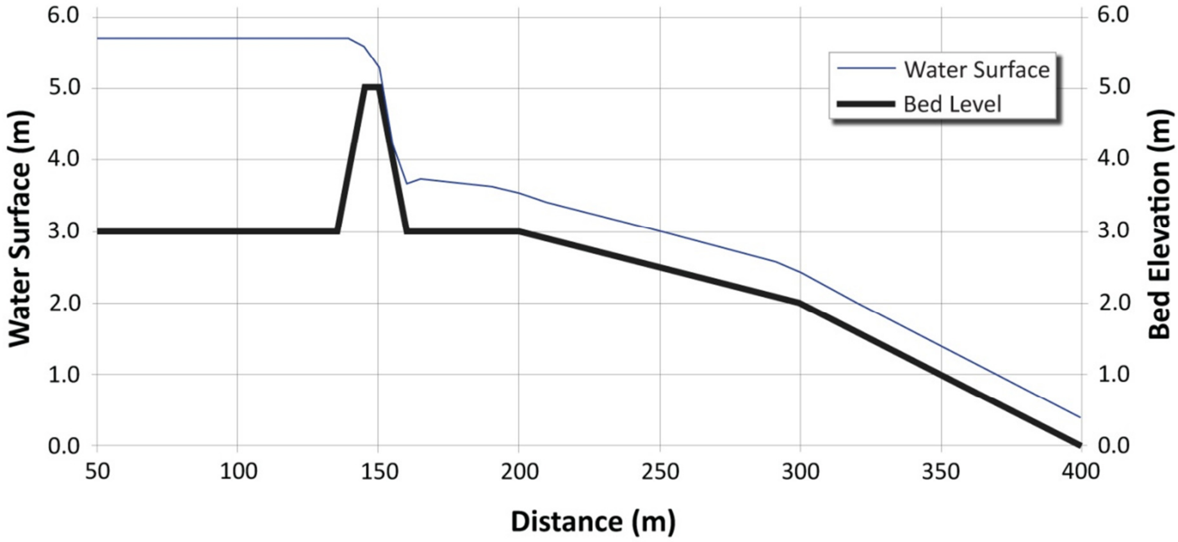


Figure 10-2 Over-topping of a Levee Bank Using the Up-winded Scheme

10.2.4. References

Abbott, M.B. (1979), Computational Hydraulics – Elements of the Theory of Free Surface Flows, Pitman, London.

Lax, P.D. (1954), Weak Solutions of Non-linear Hyperbolic Equations and their Numerical Applications, Comm. Pure and Applied Math.

Leonard, B.P. (1979), A Survey of Finite Differences of Opinion on Numerical Muddling of the Incomprehensible Defective Confusion Equation, In: T.J.R. Hughes (ed.) Finite Element Methods for Convection Dominated Flows, AMSE, New York.

McCowan, A D, Rasmussen, E B and Berg, P. (2000), Improving the Performance of a Two-dimensional Hydraulic Model for Floodplain Applications, Proceedings of the 6th Conference on Hydraulics in Civil Engineering, Hobart, 2001

Stelling, G.S., Kernkamp, H.W.J and Laguzzi, M.M. (1998), Delft Flooding System: A Powerful Tool for Inundation Assessment based upon a Positive Flow Simulation, in V. Babovic and L.C. Larsen (eds) Hydroinformatics '98, A.A. Balkema, Brookfield.

Syme W.J. (2001), TUFLOW – Two & One-dimensional Unsteady FLOW Software for Rivers, Estuaries and Coastal Waters, IEAust Water Panel Seminar and Workshop on 2D Flood Modelling, Sydney.

10.3. **Adaptive Timesteps**

Certain commercially available software packages have the option of using an adaptive timestep. An adaptive timestep allows the modeller to specify the maximum timestep that the model can utilise during simulation. The model is able to use a smaller timestep when needed (e.g. when certain flow characteristics are changing rapidly or when convergence to a solution does not occur). In some cases, the scheme will continue at the reduced timestep for the rest of the simulation or change back to the maximum timestep after a number of intervals.

Adaptive timesteps have a number of advantages and disadvantages that the modeller should consider:

Advantages

- When the flow is relatively uniform a longer timestep can be used
- When rapid changes are occurring, a smaller timestep will be used to capture shocks/abrupt changes through the system
- May lead to significant run time savings, but can also lead to longer run times if forced to use very small timesteps.

Disadvantages

- Actual run time is unknown
- When doing a series of runs whose results need comparison (e.g. impact assessment), adaptive time steps can be problematic as each simulation may run at different timesteps. This makes direct comparison between one run and another difficult as one may better capture the peak flow/flood level etc
- Results will be extracted for the sampling period specified and therefore may be interpolated results
- Any errors (such as volume continuity) that accumulate with time could be very different between runs

10.4. Wetting and Drying in 2D Flood Models

10.4.1. Background

In 2D flood models, the model computational domain is divided up into many thousands of individual computational elements (fixed grid or flexible mesh cells). As a flood flows through the model, elements initially representing dry land must be brought into the computation (i.e., flooded or wetted) as the flood flow reaches them, and taken out of the computation (i.e., dried) as the flood level recedes.

Many of the commonly used 2D flood models have had their origins in 2D estuarine and coastal flow modelling. In most coastal applications, however, the flood and dry algorithms need only cope with the periodic wetting and drying of relatively small areas of almost horizontal inter-tidal flats and have not been designed to describe flooding of initially dry land down a sloping floodplain. In these models, wetting and drying has typically been achieved either by:

- Numerical "tricks" that allow the computation to continue through "dry" grid/mesh cells that may become flooded at some time during the computation, or
- By explicitly changing the internal boundaries of the model and wetting or drying grid/mesh cells, as appropriate throughout the computation.

10.4.2. Numerical Tricks

Numerical tricks that can be used include:

- the use of "slots", or
- the use of porosity.

With the slot approach, each cell has a narrow slot that continues down from the ground surface to a minimum reference level. When the water level is below the ground surface, the flow computation can continue along the slots. This allows the flow computation to continue through what would otherwise be dry grid cells. There is a transition zone near the ground surface such that, when the water surface elevation is higher than the ground level, the computation then continues over the full width of the cell. This approach relies on the slots being sufficiently narrow with respect to the grid/mesh cells that the flow in the slots is negligible relative to the overall flow.

The porosity approach works on a similar principal. The equations of motion are modified to allow for porous flow and all grid/mesh cells that are likely to be flooded in a computation are given a porosity that varies with water level. When the water level is below the ground surface, the porosity is small but finite. This allows the flow computation to continue through what would otherwise be dry grid/mesh cells. As for the slots, there is a transition zone near the ground surface and the porosity becomes unity (i.e., there are no porous flow effects) once the water surface elevation is higher than the ground level. Similar to the slots, this approach relies on the porosity of the "dry" grid/mesh cells being sufficiently small so that the porous flow effects are negligible relative to the overall flow.

Both approaches have similar problems. These can be summarised as follows:

- Significant increase in computational effort, as the computation must be carried out on all floodable grid/mesh cells at every time step.
- Unless the slot widths and porosities are extremely small, there are accuracy problems related to physically unrealistic “leakage” of surface flows to or from the subsurface flows, particularly in areas with steep ground surface gradients.
- Very narrow slot widths and very small porosities can lead to computational instability in the transition zone.

For these reasons, neither slots nor porosity are used as the wetting/drying mechanism in the more commonly used 2D flood models. As a result, the remainder of this section will focus on the more commonly used approach of explicitly wetting and drying grid/mesh cells by changing the internal boundaries of the model as the computation progresses.

10.4.3. Moving Internal Boundaries

This discussion in this section is focussed mainly on fixed grid finite difference models. Finite element and finite volume models have similar issues but the unstructured meshes used can result in additional problems when relatively large partly inundated floodplain elements may need to be simulated.

With moving internal boundaries, the main issues are in determining when to wet (or dry) a particular grid cell, and how to cope with numerical transients (localised instabilities) that may be generated when grid cells are added to (or subtracted from) the computation.

Determining when a grid cell should be flooded (or dried) can be a particular problem with models based on the numerical scheme of Leendertse (1967). This is because the grid points defining the water surface elevations are offset from the grid points containing the bed surface elevations. This can create problems with determining when a grid cell should be flooded or dried. As the Leendertse scheme forms the basis of many estuarine/coastal models (e.g., Stelling, 1984), much of the published work on wetting and drying (see for example Falconer and Chen, 1981, or Stelling et al, 1986) has been concentrated on improving the wetting and drying characteristics of Leendertse-type models. With these types of models (e.g., Syme, 2001), successful wetting and drying can be achieved through wetting/drying checks at cell boundaries, as well as at individual cells.

With models based on the numerical scheme of Abbott *et al* (1973), the water surface elevation and bed surface elevation are defined at the same point. As such, checking need only be carried out at individual grid cells (e.g., McCowan *et al*, 2000).

10.4.4. An Alternative Approach

An alternative approach has been put forward by Stelling *et al* (1998), who presented a new numerical scheme that has been aimed specifically at describing two-dimensional flows in floodplain applications. The Stelling *et al* scheme imposes a condition that water depths remain positive, allowing the computation to proceed even when the water depth is zero (i.e., the grid point is effectively dry). The scheme appears to work well, but includes significant numerical dissipation through (amongst other things) up-winding of the convective terms in the momentum equation. This dissipation helps control any numerical transients that might otherwise occur. With the computation continuing on effectively dry grid points, it could also be expected to increase the computational time significantly relative to the moving internal boundary approach.

10.4.5. Issues

Experience gained in early 2D flood models showed that there could be a number of problems associated with applying the above approach to floodplain studies. These early problems included:

- Numerical transients: These could be generated as computational cells were brought into (or removed from) the computation. In serious cases these could lead to general instability. Numerical transients were sometimes countered by increasing the bed-friction during the wetting and drying process.
- Numerical wetting induced mass errors: In some early schemes, an initially dry grid point was assumed to have a user specified initial depth in order for the computation to be maintained across recently flooded grid cells. Even with a relatively small initial depth of say 0.10m, this could lead to significant mass errors over a wide expanse of floodplain.
- Sticking grid points: With high velocity floodplain flows some numerical schemes will occasionally compute extremely small or even negative water depths. In some cases the bed-friction associated with very small water depths could become so high as to preclude significant flow into or out of such a grid point, irrespective of the water level in adjacent grid points. In these cases, increased friction to damp out numerical transients would exacerbate any tendency for grid cells to stick.
- Retarded propagation of flood wave fronts: Problems with inappropriate wetting and drying algorithms could result in the front of a flood wave propagating more slowly than it should. This would be exacerbated by any problems with “sticking” grid points.

The results of an example simulation have been used to illustrate the potential problems with the last two of these points. The simulation involved water spilling out over a wide, initially dry horizontal floodplain, and then flowing down a steepening slope. The grid size was 5m, the time step was 1s and the slope gradients were 1:100 and 1:50. The discharge at the upstream boundary was held constant at $1m^2/s$ per metre width of floodplain. The bed roughness was equivalent to a Manning’s ‘n’ of 0.03 over the floodplain.

The results presented in Figure 10-3 and Figure 10-4 show profiles of water depths at 30-second intervals after water first began to spill out over the floodplain. The results in Figure 10-3 were obtained using an early wet/dry algorithm which had problems with sticking and excessive friction during the wetting process. This has resulted in an artificial build-up of the water surface behind the wave front where depth of the flow become more than twice that of the equilibrium depth for uniform flow in this region.

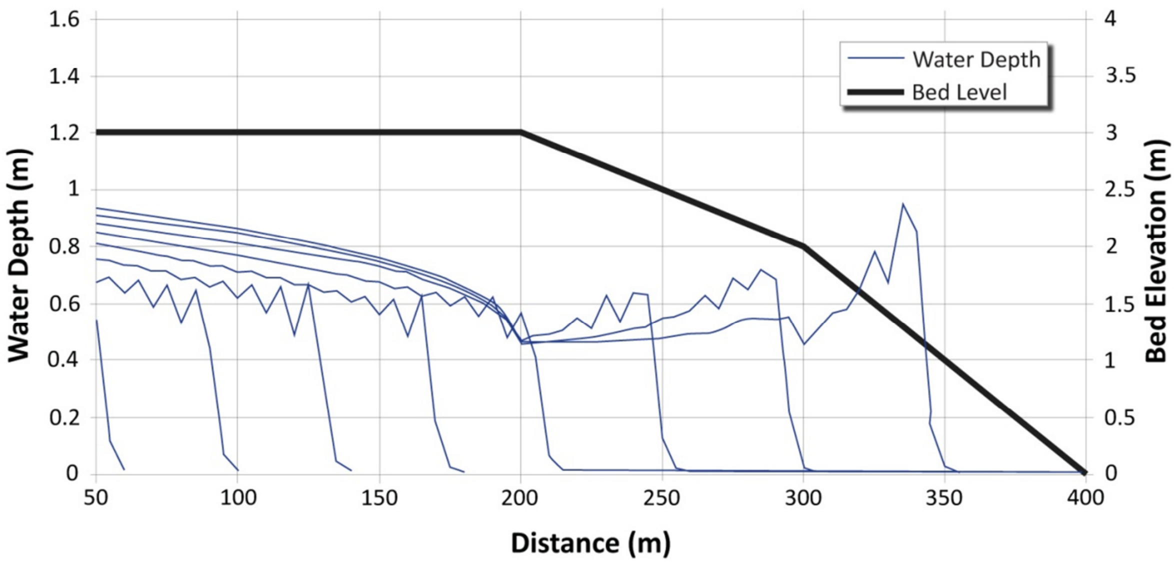


Figure 10-3 Water Depth Profiles Computed Using an Early Wet/Dry Algorithm

The results in Figure 10-4 were obtained with an improved wet/dry algorithm. They show the flood wave propagating more smoothly out over the floodplain with water depths approaching the equilibrium depth for uniform flow down each of the slopes. The front now propagates approximately 20% faster than with the original scheme, and there is no artificial build up of water behind the front.

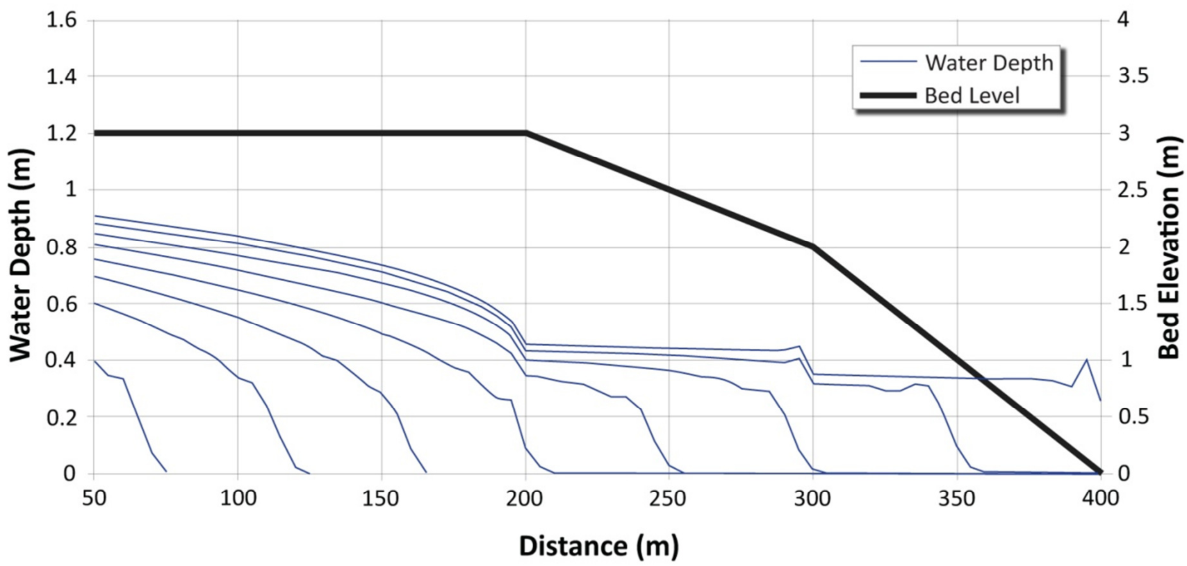


Figure 10-4 Water Depth Profiles Computed Using an Improved Wet/Dry Algorithm

10.4.6. Recommendations

The 2D hydrodynamic models available for floodplain modelling use a range of different approaches to wetting and drying different parts of the computational domain. Model users should acquaint themselves with the actual wetting and drying technique used in their particular model and should make themselves aware of any potential problems associated with this technique. Users should also familiarise themselves with the purpose of any user definable input parameters and ensure that they are set to appropriate values for their particular application.

10.4.7. References

Abbott, M.B., (1979). Computational Hydraulics, Elements of the Theory of Free Surface Flows, Pitman, London.

Abbott, M.B. and Rasmussen, C.H. (1977). On the Numerical Modelling of Rapid Contractions and Expansions in Models that are Two-dimensional in Plan, Proc. 17th Congress IAHR, Baden-Baden.

Abbott, M.B., Damsgaard, A. and Rodenhuis, G.S. (1973). System 21, Jupiter, A Design System for Two-Dimensional Nearly Horizontal Flows, Journal of Hydraulic Research, Vol. 1.

Abbott, M.B., McCowan, A.D. and Warren, I.R. (1981). Numerical Modelling of Free-Surface Flows that are Two-Dimensional in Plan, in Transport Models for Inland and Coastal Waters, H.B. Fischer editor, Academic Press, New York.

AUSTROADS, 1994 Waterway Design: A Guide to the Hydraulic Design of Bridges, Culverts and Floodways, Technical Editor, David Flavell, Sydney

Barton, C.L. (2001) Flow Through an Abrupt Constriction – 2D Hydrodynamic Model Performance and Influence of Spatial Resolution, Thesis submitted as partial fulfilment for Master of Engineering Science, Environmental Engineering, Griffith University

Bishop, W.A., McCowan, A.D., Sutherland, R.J. and Watkinson, R.J. (1995). Application of Two-Dimensional Numerical Models to Urban Flood Studies, 2nd Int'l Symp. on Stormwater Management, Melbourne.

Collins, N.I., Callaghan, D.P. and McCowan, A.D. (1999). Zero Flooding Impact Assessment – The Need for Full Two-Dimensional Analysis, 8th Int'l Conf. on Urban Storm Drainage, Sydney.

Falconer, R.A. and Chen, Y. (1981). An Improved Representation of Flooding and Drying and Wind Stress Effects in a 2D Tidal Numerical Model, Proc. Instn of Civil Engrs, Part 2.

King I, 2004, RMA2 – A Two Dimensional Finite Element Model for Flow in Estuaries and Streams Version 7.4, Resource Modelling Associates, Sydney, Australia

10.5. **Valid Parameter Ranges/Envelopes**

This Section provides guidance on ranges of parameter values that are currently regarded by the industry as acceptable. Background discussion and detail on these parameters are provided in previous chapters, including: Chapter 5 on Data, Chapter 6 on Model Schematisation and Chapter 7 on Calibration and Uncertainty.

10.5.1. Roughness

Overview

In a 1D model roughness is typically assigned according to the cross-sections or along particular branches. For 2D models, roughness is generally specified as a spatially varying grid/mesh over the 2D model domain. It is important to note that the loss processes embedded in hydraulic roughness parameters for 1D and 2D models, whilst closely related, are not exactly the same.

In a 1D model the roughness parameter can account for:

- Friction losses associated with the bed material of a channel/floodplain,
- Drag losses associated with vegetation or other obstructions in the channel/floodplain,
- Form losses due to turbulence in a channel/floodplain due to channel geometry,
- Variations in geometry and associated form losses between cross-sections,
- Bend losses in a channel.

In a 2D model, some of the above losses are to some degree accounted for by the numerical scheme. For example, some aspects of bend losses due to change in directional momentum are explicitly modelled in a full 2D solution. Similarly, part of the form loss due to variations in geometry will be explicitly modelled in 2D scheme, depending on the grid/mesh resolution. Whilst the 2D roughness parameter nominally represents friction loss due to the ground surface material in each grid/mesh element, in practice there are still many sub-element loss processes that are not explicitly described, such as vegetation resistance (trees, shrubs), physical obstructions (fences, cars, poles etc) and local variability in topography. In effect, roughness parameters in a 2D domain also need to account for some losses in addition to the bed frictional losses, but less so than a 1D domain applied over the same area. In general, increasing grid/mesh resolution in the 2D domain can result in more of the additional losses being accounted for within that domain and less needing to be compensated for within the roughness parameter.

In urban areas, the way in which buildings are represented in the model has a significant bearing on the specification of roughness. In areas where buildings are explicitly represented as obstructions in the model topography, roughness in the surrounding areas should only account for the nature of the land-use (such as grass, paved, or vegetated areas). Alternatively, where buildings or other major obstructions are not explicitly modelled, the impact of these features on losses can be incorporated into the roughness parameter, using a significantly higher value than would otherwise be the case. Chapter 10.11 contains further discussion on incorporating buildings, fences and other urban features within the 2D domain.

Developing Hydraulic Roughness

Applicable ranges for hydraulic roughness in 1D models have been well established and defined in numerous references over the last 50+ years, such as Chow *et al.* (1988). Values that represent average conditions within and between cross-sections are applied either at the cross-section or along a part or all of branch.

Roughness for 2D models is generally specified as a map and based on land-use information that can be derived from aerial photography, satellite images, planning zone maps or field observations. Different areas can be digitised into land-use polygons representing zones of similar loss characteristics (e.g., vegetation or impervious surface type). This is typically conducted in a GIS environment and then transferred to the required format for a specific model package.

Roughness maps have also been generated from auto image or LiDAR processing in some areas, however this is not a commonly adopted technique at present.

2D roughness is generally parameterised in terms of Manning's 'n', or similar related parameterisation of bed friction. Typical ranges of 2D roughness parameters for various land-use types are listed in Table 10-1.

Table 10-1 Valid Manning's 'n' Ranges for Different Land Use Types

Land Use Type	Manning's 'n'
Residential areas – high density	0.2 – 0.5
Residential areas – low density	0.1 – 0.2
Industrial/commercial	0.2 – 0.5
Open pervious areas, minimal vegetation (grassed)	0.03 – 0.05
Open pervious areas, moderate vegetation (shrubs)	0.05 – 0.07
Open pervious areas, thick vegetation (trees)	0.07 – 0.12
Waterways/channels – minimal vegetation	0.02 - 0.04
Waterways/channels – vegetated	0.04 – 0.1
Concrete lined channels	0.015 – 0.02
Paved roads/car park/driveways	0.02 – 0.03
Lakes (no emergent vegetation)	0.015 – 0.35
Wetlands (emergent vegetation)	0.05 – 0.08
Estuaries/Oceans	0.02 – 0.04

10.5.2. 2D Grid/Mesh Resolution

The resolution of a 2D model grid/mesh determines the scale of physical features and flow behaviour that can be modelled for a given study area. Selection of an appropriate resolution is generally driven by a combination of the following factors:

- The scale of topographic and/or flow phenomena to be modelled
- The desired level of detail to be achieved in the model outputs
- The length of event time and consequent run time
- The size of the area of interest

Details of the model schematisation process including resolution aspects are described in Chapter 6. Chapter 7 also highlights the importance of grid/mesh resolutions in achieving manageable run times to maximise calibration outcomes.

Table 10-2 provides guidance on levels of model resolution that may be appropriate in certain typical situations.

Table 10-2 Typical Grid/Mesh Resolutions

Modelling Case	Typical 2D Element Resolution
Flow in a channel	In order to adequately resolve flow in a channel it is desirable to provide at least 5 grid/mesh elements laterally across the channel
Urban overland flow	Most urban flood models employ grid/mesh resolutions of 2 to 5 m with up to 10 m acceptable in some cases.
Rural floodplain flow	Rural floodplain models typically employ grid/mesh resolutions of between 10 m and 50 m (although resolutions up to 200 have been used) depending on the size of the area to be analysed, the characteristics/dimensions of the floodplain and the desired level of output detail.
Lakes or estuaries	These situations often include areas of open water were less detail is required than along the water body boundary. Such situations are well suited to a flexible mesh rather than a grid-based model as the mesh is able to incorporate a change of resolution across the model domain. Element resolutions for these models can span the full range as described above depending on project requirements.
Flow over an embankment	Embankments effectively operate as weirs in the floodplain context. Many 2D modelling packages have automatic or manually activated corrections that compensate for the error in head loss typically associated with modelling broad-crested weir flow with a shallow-wave 2D scheme. For practical purposes, a single 2D element is generally adequate to represent the impact of a levee, road or railway embankment. The resolution of these elements is generally not a significant limitation on the schematisation of most model domains.

10.5.3. Timestep

The computational timestep is a key determinant of numerical model stability and accuracy. In a practical sense it dictates the rate at which flow behaviour is computed within a numerical scheme relative to the actual physics of the flow itself. If this rate is too high, the model will either be unable to accurately resolve the propagation of the flood wave, resulting in errors, or the model may simply become unstable and crash.

There are two important aspects to consider when selecting a computational time-step in a numerical model.

- Shallow water wave celerity – this is the speed at which a small water wave will propagate through shallow water (a typical example of this is the speed of ripples travelling away from a splash in a pond). This is a function of depth: $c = \sqrt{gh}$
- The maximum advective speed within the model, this is the maximum rate at which water physically moves through the model.

A common way to determine an appropriate computational timestep is using a parameter called the Courant Number. The Courant Number represents the propagation of information in the model relative to the element resolution and timestep as follows.

$$Cr = \frac{\left(v + \sqrt{gd}\right) \times \Delta t}{\Delta x}$$

Where v is velocity (m/s), g is gravity, d is depth, Δt is the timestep and Δx is the grid/mesh dimension.

In order to ensure stability and accuracy within a 2D model, explicit schemes require a Courant number of 1 or less. Semi-implicit schemes, such as those used for fixed grid finite difference solutions are able to run at Cr conditions greater than one but, are unlikely to produce stable and reliable results for Cr values greater than 10. Where depths are shallow and velocities high, the maximum advective speed can be substituted in place of the shallow wave celerity to calculate the Courant Number. This is also discussed in Chapter 6.

Whilst the Courant Condition is a useful guide to setting an appropriate timestep, there are other rules of thumb that can be useful, such as:

- Choose an initial timestep of ½ the grid size, this effectively assumes that the maximum celerity in the model is 2m/s, which is reasonable for most flooding situations.
- Halve the present timestep and check if there are any differences in the results. If there are no significant changes between the model runs then the larger timestep should be appropriate to use (assuming results are stable etc).

10.6. Turbulence and Eddy Viscosity

10.6.1. Introduction

Turbulence is a three dimensional phenomenon and responsible for a significant amount of energy losses within rivers. The eddy viscosity term or turbulence exchange coefficients are used in two-dimensional depth averaged numerical modelling in order to characterise energy losses caused by turbulence effects at a sub-grid scale (Barton, 2001). In 2D hydraulic modelling, the eddy viscosity term can be expressed as a constant parameter or as a function of local flow properties (and will therefore vary at different locations within the model domain).

10.6.2. Background

Turbulence is an intricate three dimensional, time dependant phenomenon. As the equations which describe turbulence cannot be solved directly for practical applications, simplifications are often employed to account for the turbulence and associated momentum transfer and energy loss in numerical models. A number of different turbulence or "eddy viscosity" models are used to estimate turbulence effects. These models range from simple zero-equation models to more complex n-equation models. Turbulence models from the simpler end of the range are typically used for flood modelling purposes. Turbulence models typically found in current numerical software are as follows (in order of simple to complex):

- No turbulence model – some commercially available 2D numerical model software do not incorporate a specific eddy viscosity term in their momentum equations and thereby implicitly neglect any momentum transfer due to turbulence.
- Zero-equation turbulence models (i.e. constant eddy viscosity term) :
    - Prandtl mixing length model, and
    - Constant eddy viscosity model: In a detailed review of different types of turbulence models, Rodi (1984) describes the constant eddy viscosity as a simplification for representing turbulent shear stress.
- One-equation turbulence model:
    - The Smagorinsky turbulence formulation: Despite its relative simplicity, the Smagorinsky formulation is currently the most complex eddy viscosity model used in flood modelling applications.
- Two-equation turbulence model
    - The *k-ε* model (Launder and Spalding, 1974): This model is not currently used in commercial flood modelling applications but is used extensively in coastal and oceanographic modelling, particularly in 3D modelling.

The zero-equation models simply allow a constant eddy viscosity value to be set by the modeller to account for turbulence. The Smagorinsky formulation has the advantage of dynamically determining the eddy viscosity value for each grid/mesh element based on the element size and velocity gradient (King, 2004) (effectively a mixing length approach). In this way it is able to vary throughout the model both temporally and spatially depending upon local flow conditions. It effectively considers local generation of turbulence due to current shear. The k-є model considers the generation, transport and decay of turbulence. It is more complex than the Smagorinsky formulation, but for most floodplain applications its detail is unwarranted.

10.6.3. Impacts of Eddy Viscosity Terms in Flood Models

If the model incorporates a constant eddy viscosity term, the larger the constant the more momentum is transferred to the floodplain. This results in a lowering and flattening of the peak velocity in the channel and increasing overbank velocities. Figure 10-5 (Retallick and Babister, 2008) illustrates advantages and disadvantages of the constant eddy viscosity model versus the Smagorinsky Formulation by comparing the model outputs with results computed from the Turbulent Mixing Length Hypothesis (TMLH) (Werner and Lambert, 2007). The TMLH was developed specifically to account for the momentum transfer in compound channels and as such provides a good basis for comparison. Adjustment of the constant eddy viscosity results in significant variations in the velocity distribution and magnitude compared to relatively small variations when the coefficient for the Smagorinsky Formulation is varied. This is due to the fact that the first order Smagorinsky model relates the stresses directly to the explicitly predicted volume-averaged flow variables. As such the model is better suited in a predictive capacity where calibration data is not available. However, where such data is available, the constant eddy viscosity allows the modeller to modify the coefficients so as to replicate measured data. Conversely this becomes an issue when no calibration data is available and can result in significantly erroneous predictions of flow variables depending on what value the coefficient is assigned.

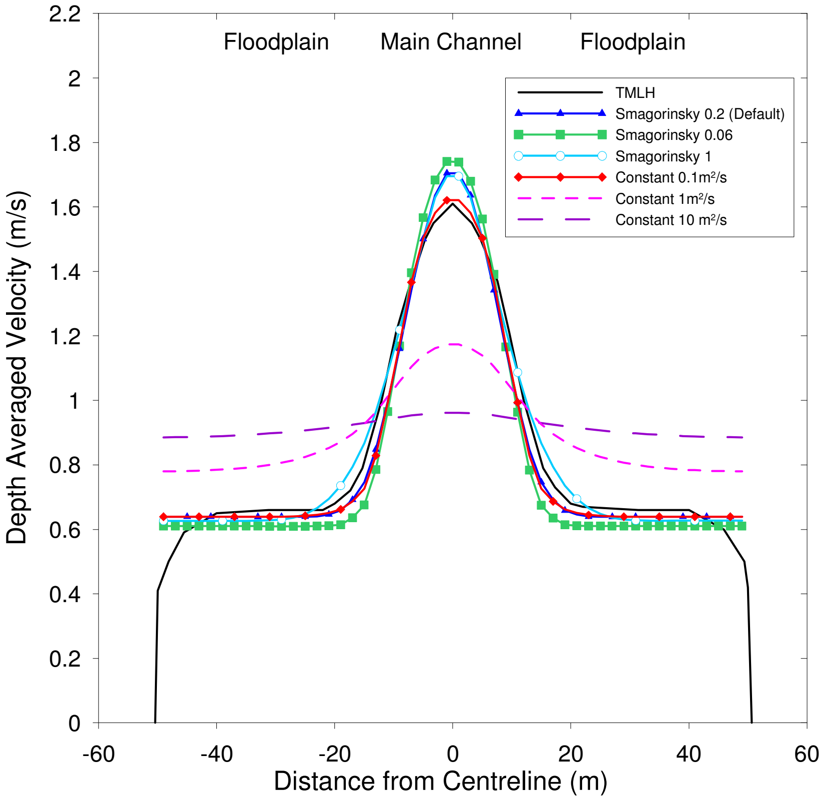


Figure 10-5 Comparison of Velocity Profiles - Smagorinsky versus Constant Eddy Viscosity (courtesy of Retallick and Babister, 2008)

Two-dimensional models that do not include eddy viscosity formulations do not transfer momentum across the waterway correctly. This may result in the incorrect predictions of main channel and floodplain conveyance, which can be of great importance to model calibration. Ignoring the effects of momentum transfer in some compound channel flow situations can result in other parameters (e.g. roughness) being altered to compensate for the lack of momentum transfer due to turbulent shear stress. The resulting calibrated model may only be produce reliable flows for those events with similar flow magnitudes to that of the calibration event. For example, if momentum transfer is important and is ignored, then calibrating a model against an in-channel flow event and applying it to the simulation of an overbank flow situation would result in an overestimate of main channel flow and a corresponding underestimate of overbank flows. This should be of particular concern to flood modellers when determining the site of potential floodplain development. This is also of importance when investigating hydraulic hazard (calculated as the product or some function of velocity and depth), the safety of people and determining where development should occur. Figure 10-6 shows the hydraulic hazard across a channel predicted by models using different eddy viscosity formulations compared to field measurements. Furthermore, neglecting the momentum transfer characteristic of compound channels can overestimate the overall conveyance capacity of a channel by up to 25% (Martin and Myers, 1991).

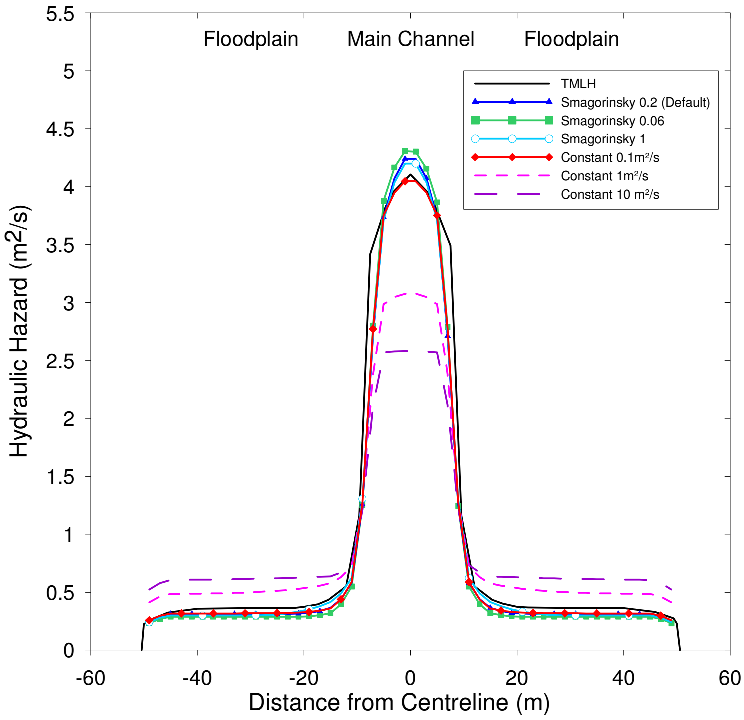


Figure 10-6 Comparison of Hydraulic Hazard - Smagorinsky versus Constant Eddy Viscosity (courtesy of Retallick and Babister, 2008)

It is recommended that further testing in the manner of Retallick and Babister (2008) (who investigated the ability of a selection of currently available commercial software to predict the velocity and hydraulic hazard distribution of a test channel compared to the theoretical distributions) be conducted on all currently used commercial models.

10.6.4. Issues

There are a number of issues associated with the potential momentum transfer between the channel and overbank regions in 2D hydraulic modelling. Good modelling practice requires the:

- Determination of when accurate representation of momentum transfers is required
- Knowledge of different approaches adopted by different model software to account for this process (e.g. which (if any) eddy viscosity formulation is used by the model software and is this approach appropriate for the purpose of the model?)
- Linking between the 1D and 2D domain is usually based on continuity only and ignores momentum transfer (although some model software can include momentum transfers between 1D and 2D domains – refer to User Manuals). However, if accurate representation of momentum transfers between channel and overbank areas is required, this should be done in a full 2D model and 1D/2D linking should not be used in the critical areas requiring momentum transfer.
- 2D models consider vector components while 1D assumes direction
- Momentum transfer may affect model calibration. As the effects of momentum transfer vary with flood level, ignoring the momentum transfer mechanism can limit the range of flows to which a calibration is applicable.

10.6.5. Practical Advice

In situations where the modeller believes the representation of momentum transfer to be important, the choice of model is critical. The modeller should give special consideration to the eddy viscosity formulation used by the numerical model software and the use of a model with a more complex eddy viscosity formulation. The modeller needs to be aware that some problems are inherently 3D, and a 2D depth-averaged model (and particularly depth averaged velocities) may provide a poor representation of the flow behaviour. 1D models are further limited by their assumption of parameter uniformity throughout a subsection.

One example where the eddy viscosity term may be particularly important is when modelling a smooth channel with rough overbank, such as an older style concrete-lined channel in urban areas. However, momentum transfer is further complicated in many Australian rivers where the extremely rough riparian zone will affect momentum transfer.

Sensitivity testing should be conducted if the modeller believes that eddy viscosity is important to the situation they are modelling, with calibration to field measurement where possible.

The following general rules apply to the relevance of eddy viscosity to the model:

- when grid/mesh resolution >> depth, eddy viscosity not as important as friction is dominant in determining flow distributions.
- when grid/mesh resolution ≥ depth, eddy viscosity begins to become important and the use of constant eddy viscosity as a minimum is appropriate.
- when grid/mesh resolution ≤ depth, sub-grid scale processes become increasingly important. If, under these circumstances, a reliable description of flow distribution at grid scale is required, then a more reliable method of estimating the eddy viscosity coefficients will be required (e.g. Smagorinsky formulation)

Use of the Smagorinsky formulation approximation is recommended when the model grid/mesh resolution is less than, or approximately the same, as the depth. A constant eddy viscosity value is typically recommended when the grid/mesh resolution is much larger than the depth or when other terms in the momentum equation are expected to dominate (e.g. high bed resistance). A more complex formulation such as k-є may be more appropriate in some applications, particularly 3D situations. Users are referred to the model manuals for further guidance as the guidance provided here is of a generic nature.

10.6.6. References

Barton, C.L. (2001) *Flow Through an Abrupt Constriction – 2D Hydrodynamic Model Performance and Influence of Spatial Resolution*, Thesis submitted as partial fulfilment for Master of Engineering Science, Environmental Engineering, Griffith University

Martin, L.A. and Myers, W.R.C. (1991) *Measurement of overbank flow in a compound river channel*. Water, Maritime and Energy; Proceedings of the Institution of Civil Engineers, 91**,** pp. 645-657

Retallick M, and Babister M, (2008) Comparison of two-dimensional modelling approaches used in current practice to estimate hydraulic hazard, Proceedings of the 9th National Conference on Hydraulics in Water Engineering, Darwin Australia 23-26 September 2008

Rodi W, 1984, *Turbulence Models and their Application in Hydraulics- State of the Art Review* (2nd Edition), International Association for Hydraulic Research, Netherlands, 116p.

Werner, M.G.F. and Lambert, M.F. (2007) Comparison of modelling approaches used in practical flood extent modelling, *Journal of Hydraulic Research*, Vol 45 No. 2, pp. 202-215

10.7. Very Shallow or Deep Flows in Two Dimensional Modelling

A two dimensional model of free surface shallow flow solves the depth-averaged governing equations by depth-averaging the mean equations for three-dimensional turbulent flows. This averaging contains an inherent assumption in relation to the extent of formation of the turbulent boundary layer within each grid cell. Consequently the relative flow depth to grid or element width ratio has an important influence on the lateral momentum transfer in the case of deep flows and vertical momentum transfer in the case of shallow flows.

Abbott and Basco (1989) defined the so-called "Abbott Number" which expresses the ratio of the water depth to the grid spacing.

$$Ab = \frac{h}{D_s}$$

Where Ab is the Abbott Number, h is the water depth, and D_s is the grid spacing.

The Abbott number is defined as the relative influence of the sub grid diffusive effects that proceed at the length scale of the fluid depth to the diffusive effect in the horizontal plane. In other words, the Abbott number can be regarded as the ratio of the horizontal and vertical eddy. In theory, the Abbott number should always be less than 1. However in practise, this limit can be and is often exceeded requiring a pragmatic and cautionary approach when reviewing results.

The effect of the diffusive terms in the vertical will be more pronounced where the bed roughness is low such as in deep fast flowing rivers. Care should be taken to review results where the river depth is significantly greater than the element width. For example, fine-scale resolution modelling of bridge piers in a deep river where the element sizes are smaller than the piers and smaller than the depth may produce unrealistic flow patterns with streamlines failing to reform and energy losses overestimated. Adjustment of the eddy viscosity coefficients or formulation may help in producing improved results. Benchmarking to desktop analyses and calibration where possible should be carried out.

10.7.1. References

Abbott, M.B. and Basco, D.R. (1989), *Computational Fluid Dynamics – An Introduction for Engineers*, Longman Scientific & Technical, Singapore

10.8. Numerical Precision

The numerical precision used by software is usually either single or double precision. Single precision uses 4-byte real numbers or around seven significant figures. Double precision uses 8-byte reals or around 13 significant figures. Depending on the compiler, single precision usually runs faster than double precision (e.g. at current speeds, about 30% faster). Double precision will also consume significantly more RAM than single precision.

If there is an option to use single or double precision, some guidelines are:

1) If near identical results are obtained using single and double precision, then the additional accuracy offered by double precision is not beneficial. In this case use the single precision build as it is likely to run faster and consume less RAM.

2) If different results are obtained, or improved convergence/lower mass error occurs when using double precision, the double precision version should be used.

Double precision is most likely necessary for schemes that solve for water level (rather than depth) where the following occurs:

1) The model is highly elevated (i.e. the ground elevations are >1,000m).

2) Direct rainfall simulations as the amount of rainfall falling on a 2D grid/mesh element in one timestep can be very small relative to the elevation of the element. For example, in single precision an elevation of 10m plus a rainfall of 0.000005m in one timestep may be calculated as 10.00000 or 10.00001 causing a 100% mass loss or 100% mass gain.

10.9. **Sub-Grid features**

10.9.1. Background

Nearly every model domain will contain important features that cannot be represented at the chosen model resolution due to their smaller relative size. These are called "sub-grid" features. Two typical types of sub-grid features are:

- Conveyance features, such as small drains, channels and gutters that are much narrower than the grid size. These features may provide hydraulic links between parts of the terrain and can be important during the initial wetting and final drainage phases. They are often critical in defining the hydraulic behaviour of low flow.
- Localised obstructions or blockages, such as small structures, fences, gates, vegetation and cars.

The modeller needs to decide if sub-grid features can be ignored, or treated in some coarser way, and how this will affect the model results. In some cases, the presence of the sub-grid features will not affect peak flood levels due to their small size. However, they may impact upon the way in which floodwaters enter and leave an area and if they are not represented in the model, the model is not able to reproduce this behaviour. Again, this may not affect the peak flood levels or major flood behaviour, but it may cause the community to lose faith in the model and results if the initial wetting and final drainage behaviour predicted by the model is different from what they have seen in reality.

10.9.2. Solutions

Flexible mesh models are sometimes able to mitigate the problem of sub-grid features due to the modeller's ability to adapt model resolution, which is not possible with fixed grid models. The modeller can also consider other solutions as outlined below.

10.9.2.1. Sub-Grid Conveyance Features

Some modelling software allows the modeller to account for sub-grid conveyance features by placing a small 1D network within the 2D domain. Chapter 6, 9 and 10.10 provide detailed guidance on incorporating 1D domains within 2D domains.

Some modelling software allow 2D elements to have a state in which they are only partially wet, which allows limited conductivity between elements before they are fully wet. In many cases, it is necessary to document that drainage times are unrealistic once the water level drops below a certain level and to accept that water will become trapped in artificial depression storages.

10.9.2.2. Blockage Features

Sub-grid blockage features can be difficult to include. The simplest approach is to add additional roughness to account for the obstruction, which works well when the obstructions cover a large area but is less satisfactory when the obstruction is localised such as the flow between buildings, fences and gates. The problem with trying to account for blockage of flow area with roughness is that it does not often work well across a range of depths. Some modelling software allow for the partial blockage of the connection between elements. This approach allows a more the realistic schematisation of the obstruction. This can be a very useful way to simulate the obstruction component of bridge piers, or gaps between buildings. To model an obstruction over a range of the depths, the percentage obstruction needs to vary with depth. The modeller will need to be careful with the interpretation of model velocity in this situation as the reported velocity might be either the average velocity over the full width of the 2D element or actual velocity in the non-obstructed section. Further guidance is provided in Chapter 10.11.

10.10. **Structures**

10.10.1. Background

Representing hydraulic structures such as bridges and banks of culverts is one of the more challenging aspects a modeller faces. The flow patterns through a structure are complex and three-dimensional (3D) in nature, therefore necessitating major assumptions when applying 1D or 2D solution schemes.

Dynamic 1D domains usually substitute the momentum equation with equation(s) representing the flow passing through the structure based on the upstream and downstream water levels. This is reasonably straightforward for some structures such as broad-crested weirs. For more complex structures such as a bridge with embankments and piers, the modeller relies on judgement as to the energy losses that occur due to the much more complex 3D flow patterns. The estimation of the energy losses may be automated by the modelling software or derived from publications that provide suitable guidance. Either way, it is not uncommon for the modeller to either unwisely rely on the software or to have difficulty when deriving and applying loss coefficients.

2D domains pose an additional level of complexity when modelling structures, because 2D schemes inherently model a proportion of the structure's energy losses such as that from the expansion of flow downstream. Therefore, to apply the same energy losses as would be applied when using a 1D scheme is fundamentally wrong. Essentially, energy losses applied to a 2D scheme by the modeller at a structure need to represent the losses from fine-scale features that the 2D element resolution cannot represent adequately, such as piers and eddy formations at the vena-contracta. In addition, any energy losses in the vertical dimension (i.e. 3D effects) would require additional energy losses to be applied by the modeller.

The dilemma for the modeller is how much additional energy losses should be applied when using a 2D scheme as there are no guidelines available as there are for 1D schemes.

There is also an added complexity when 1D elements are embedded in a 2D scheme. Once again the modeller is faced with the dilemma of how much of the energy losses the 2D scheme is automatically modelling, and therefore, by how much should the energy losses applied by the modeller to the 1D element be reduced to compensate for those losses inherently represented by the 2D domain.

10.10.2. Issues

Key issues the 2D modeller needs to be aware of when representing structures are:

- Appropriate energy losses across the structure and the vertical variation in these losses (eg. as the bridge deck becomes submerged).
- Correct representation of the structure flow area.
- Duplication of energy losses if embedding a 1D element to represent the structure.
- Stability of the scheme (structures are a common source of instabilities).
- Calibration of parameters (if suitable data are available).

- Benchmarking against other approaches.
- Blockage considerations.

10.10.3. Structure Flow Area

For a 1D scheme it is relatively straightforward to have the correct flow area as the structures shape/dimensions are reasonably accurately defined by a cross-section or in the case of culverts, the dimensions.

For 2D schemes, especially if the grid/mesh resolution is coarse, cross-checks should be made as to whether the correct flow area is being reproduced at different stages of the flood.

Once the structure obvert (soffit) is surcharged, it is very important that the flow area remains fixed, otherwise the correct velocity will not be calculated. This latter point is particularly important as the structure losses are proportional to the velocity squared, therefore, an inaccurate velocity can results in a highly inaccurate energy loss.

If the structure is overtopped, allowance also needs to be made for modelling the flow over the top of the structure. This may be a feature of the 2D scheme or modelled as a 1D weir element or similar.

For banks of culverts modelled using 2D elements, there is the additional need to represent the increased wetted perimeter. 2D schemes usually only apply a wetted perimeter equal to the horizontal width of the element (ie. no side friction). Therefore the software needs to be able to increase the wetted perimeter at 2D culvert structures, otherwise the modeller needs to make some manual adjustment such as artificially increasing Manning's n.

Flexible mesh solvers offer the ability to provide a finer resolution of elements through hydraulic structures and, therefore, provide a more accurate depiction of the flow width, especially where sloping abutments are present. However, the mesh needs to be fine enough to replicate the changes in flow width. For example, if only one element represents the sloping section of the abutment, this element will be either wet or dry and will be a poor representation. Several elements are needed that progressively become wet or active as the flood rises. Care should be taken in reviewing the results where the element width becomes smaller than the depth (see Section 10.7).

Structures usually represent a constriction in the flow patterns causing a contraction then expansion of the flow lines, thereby losing energy in the process. If the structure flow area is greater than the approach and departure flow areas (whether 1D or 2D), consideration needs to made as to whether a structure should be modelled.

10.10.4. Energy Losses

A simple and worthwhile check is to:

- Measure the afflux across the structure (upstream "still" water to downstream "still" water).

- If possible/necessary, reduce the afflux by the estimated energy loss due to bed friction using the Manning's equation (this is usually only a small, if not inconsequential, amount of the afflux).
- Take the average velocity through/in the structure and calculate the dynamic head (v2/2g).
- Divide the afflux by the dynamic head. This is a quick, indicative calculation as to the amount of energy loss (as a proportion of the dynamic head in the structure). In most cases the value should be less than 1.5. If it is greater than 1.5 then this could be an indicator that inappropriate/incorrect parameters have been specified. It can also be an indicator that there are significant other losses such as severe flow expansion downstream (Syme 2006). Surcharging against the bridge deck will also cause increased energy losses.

Indicative energy loss coefficients and affluxes can be derived from publications such as Hydraulics of Bridge Waterways (FHA, 1973) and AUSTROADS (1996). Compare the afflux in the 2D model with those derived from publications such as these.

Another common approach is to use software such as HEC-RAS or other 1D software to construct a 1D model of the structure and compare affluxes.

A close match may not result between different approaches due to the various assumptions in the different methods. However, there should be some level of consistency between the results. Markedly different results would indicate a problem that may need to be further examined by further analysis and investigation into the uncertainties of each method used.

Of importance is that the software allows for the 2D model afflux to be adjusted, primarily through the application of additional energy losses being manually specified by the modeller. This is particularly relevant for fine-scale features such as bridge piers and effects in the vertical such as surcharging against the bridge deck.

Increasing Manning's n at the 2D elements is generally not recommended, but may be acceptable if the structure is located outside the area of interest, and the structure energy losses, as a proportion of $v^2/2g$, are reasonably constant in the vertical (e.g. pier losses only meaning that the bridge deck is not surcharged). The derived Manning's n value should be clearly justifiable through desktop calculations.

10.10.5. Embedding 1D Elements to Represent Structures

Where the 2D grid/mesh resolution is too coarse, or the software capabilities are inadequate, a 1D element that is dynamically linked to the surrounding 2D solution should be used to represent the structure.

Of particular note is that the modeller needs to be careful not to duplicate energy losses. If the 1D element is several or more 2D elements wide, the 2D flow patterns that develop downstream of the structure as the water exits the 1D element and expands within the 2D domain will to a certain extent model the expansion (outlet) losses. Therefore there is a need to reduce the loss coefficient(s) applied to the 1D element to compensate for the duplicated energy losses, especially at the structure outlet (Syme, 2001).

There are no definitive guidelines on how to adjust/reduce the 1D loss coefficients. However, the same principle as discussed above for cross-checking affluxes using alternative methods should be used as and when appropriate.

As a general guide, 1D structures that are small relative to the adjoining 2D elements require little or no adjustment to their entrance and exit losses (e.g. a 1.2m pipe discharging into 10m wide 2D elements) as there will be little change in the downstream 2D velocity patterns due to the coarseness of the 2D elements relative to the 1D element. However, 1D structures that are large (one or more 2D elements in flow width such as a 30m culvert bank discharging onto 10m 2D elements) are likely to require a downward adjustment to their exit losses so as to not duplicate the expansion losses. In this case, the 2D scheme should be showing significant changes in the downstream velocity patterns as the water jets out from the 1D element and expands downstream. Most inlet losses are incurred as the water expands inside the structure after the vena-contracta, and as this effect is not being modelled by the 2D domain, there, is usually no need to adjust the 1D elements inlet loss coefficient other than the conventional adjustments related to factoring in the effect of the approach velocity.

10.10.6. Calibration of Parameters

If calibration data exists at the structure, these data should be used to help set/confirm any additional energy losses applied by the modeller to 2D elements, or any adjustment in the 1D loss coefficients if embedding a 1D element.

An example is provided in Syme (2006), which presents the findings from an exhaustive hydraulic modelling investigation that involved the calibration of 2D schemes and a 1:30 scale physical model to three flood events at a bridge crossing. The outcome from this investigation highlighted the need for good calibration data at structures to validate models of all persuasion.

10.10.7. Stability Issues

Instabilities can readily occur at or near structures, especially if the schematisation is poor, or incorrect/inaccurate data are entered into the model. The process for resolving the instability will vary between software and techniques, therefore it is strongly recommended that expert advice is sought from the software providers or from more experienced modellers.

10.10.8. Blockage

Blockage is a common source of exacerbated flooding and may need to be modelled. Generally there will be a number of ways to represent blockage of a structure, but in most cases the end result is to reduce the flow area/dimensions of the structure. For further guidance on this refer to ARR Project 11 (Engineers Australia, 2009).

10.10.9. References

AUSTROADS, 1994, Waterway Design: A Guide to the Hydraulic Design of Bridges, Culverts and Floodways, Technical Editor, David Flavell, Sydney

Engineers Australia, November 2009, Australian Rainfall and Runoff Revision Project 11: Blockage of hydraulic structures, Stage 1 report

FHA (1973) U.S. Department of Transportation, Federal Highway Administration (US FHA 1973) Hydraulics of Bridge Waterways Hydraulic Design Series No. 1, Second Edition.

Syme, W.J. (2001) Modelling of Bends and Hydraulic Structures in a Two-Dimensional Scheme Conference on Hydraulics in Civil Engineering, Hobart, November 2001.

Syme, W.J. (2006) Bruce Highway Eudlo Creek Hydraulic Investigations - A Turning Point 30th Annual Conference of the Association of State Floodplain Managers, June 11–16, 2006, Albuquerque, New Mexico.

10.11. **Buildings, Fences and Other Urban Features**

10.11.1. Introduction

As 2D models of increasing resolution are applied to urban areas it is increasingly the case that results are being examined in greater detail. Specific areas where planners are looking at results are in the areas around and between houses and buildings and also where flow interacts with fences and other sub-grid urban features.

Although there is little published research which is able to demonstrate how different methodologies for modelling such features in 2D models compare to gauged data, it is certainly the case that many practitioners have experience with producing 2D results that take into account different property types (on piles/stilts versus slab on ground for example) and also fence/flood interaction.

Given that there is a growing demand for results which take into account these features, this section will outline some methodologies for dealing with these. It should be noted however that more research is required in this area before anything prescriptive can be detailed. As such, suggested methodologies detailed herein are just that, and it is acknowledged that no verification of results against observed or physical model data has been carried out.

10.11.2. Background

Floodwaters flowing through urban areas can follow a complex path as the water negotiates buildings, fences and other obstructions. These obstructions dissipate energy by forcing the water to change its direction and speed. Historically, 1D and 2D models have represented this energy dissipation by either increasing the bed friction parameter (eg. Manning's 'n') where buildings and fences lie within the 1D cross-section or 2D domain. Another approach is to block out sections of the 1D cross-section, or remove/deactivate 2D elements, although this latter approach will not include the storage effects of water entering a building or underground carpark.

As 2D models have become finer in their discretisation of urban areas, software continues to improve, and hardware processing times reduce, the modeller needs to be aware of the different approaches for representing buildings and fences, and the sensitivity of model results (depth, velocity) to the various approaches.

The representation of these features is especially important for analysis of flooding in urban areas, where sometimes relatively small features can have a significant impact. These features are also of interest in rural regions, but the scale of the flooding issues in these regions is larger and the hydraulic analysis therefore is usually simpler. However, in urban areas, the effects of these features are important and more detailed assessment is also usually required. In many cases, relatively small changes in water level may divert flows significantly and these can affect flood damages and impacts.

This section deals firstly with the consideration of buildings, where several different approaches may be appropriate, and then considers fences, which are a specific concern.

10.11.3. Blocking Out Model Elements

Blocking out model elements occurs when the model grid/mesh elements are prevented from receiving model flow. This can occur in one of two ways:

- Raising the elevation of the elements, or
- Nulling (removing) the elements.

Changing Model Elevations

Blocking out of 2D elements by raising the 2D grid/mesh elevation to represent buildings may be appropriate where the building is designed or protected so that water cannot enter it, as shown in Figure 10.6. This is an appropriate representation provided the grid/mesh element sizes are sufficiently fine to adequately represent the flow patterns around the building (see further discussion in Section 10.11.10).

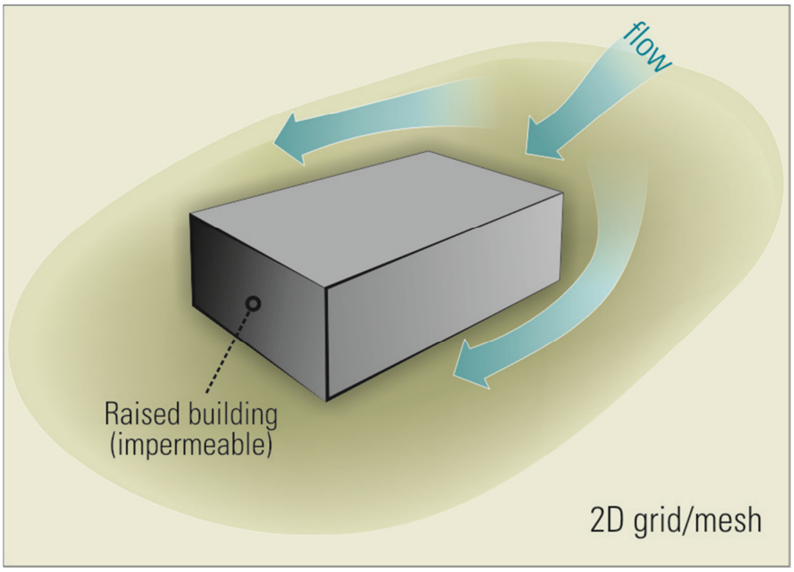


Figure 10-7 Model Elements Fully Raised (Blocked for all Flow Magnitudes)

However, as most buildings allow water to enter, thereby contributing to the floodplain storage and attenuation of the flood wave, this approach is not always appropriate. Also, commercial buildings often have underground parking, thereby contributing further to the flood storage. In the case of the latter, the 2D elements would need to be lowered so as to correctly represent the storage of the underground carpark.

Buildings are often constructed on an earth pad. In this case, if the building floor remains flood free, it can be appropriate to raise the 2D grid/mesh elements (provided they are sufficiently fine) to the height at which flow will commence (e.g. pad or floor level) This is shown in Figure 10-8. Modelling of the building structure itself above this elevation can be undertaken using one of the other methods discussed in Sections 10.11.4 to 10.11.7.

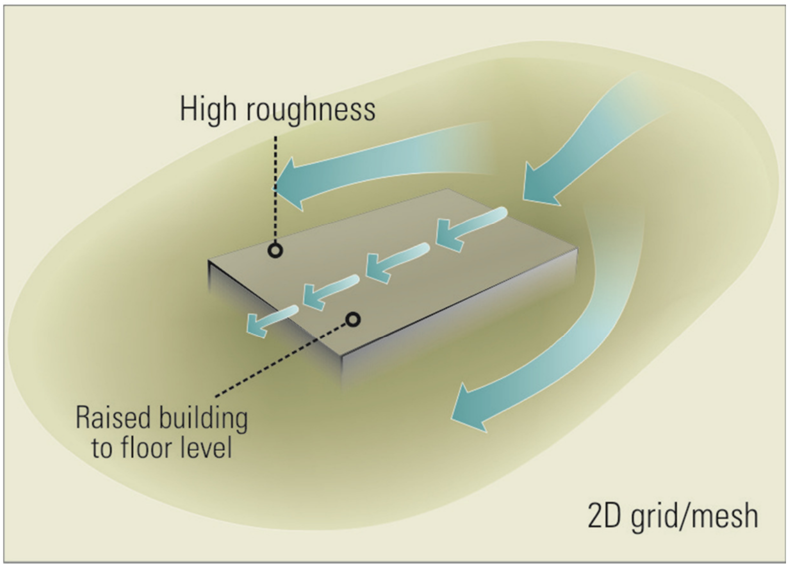


Figure 10-8 Model Elements Raised to Building Floor Level

Nulling (Removing) Cells

Another means of preventing flow in a particular area is to null (remove) the grid/mesh elements over that area, as shown in Figure 10-9. However, there are a number of issues associated with nulling 2D elements:

- Nulling elements results in "holes" in the 2D model results, and the flood level inside a building will need to be interpolated from surrounding flood levels (assuming that water can enter the building). This can be an inconvenience when interrogating a 3D water level surface to assign flood levels to buildings for a flood damages assessment, or for setting minimum floor levels for building planning controls.
- It is important to avoid nulling elements in combination with the direct rainfall method due to the potential complications of the rain volume being influenced by the number of active grid/mesh elements. This is discussed further in Section 11.

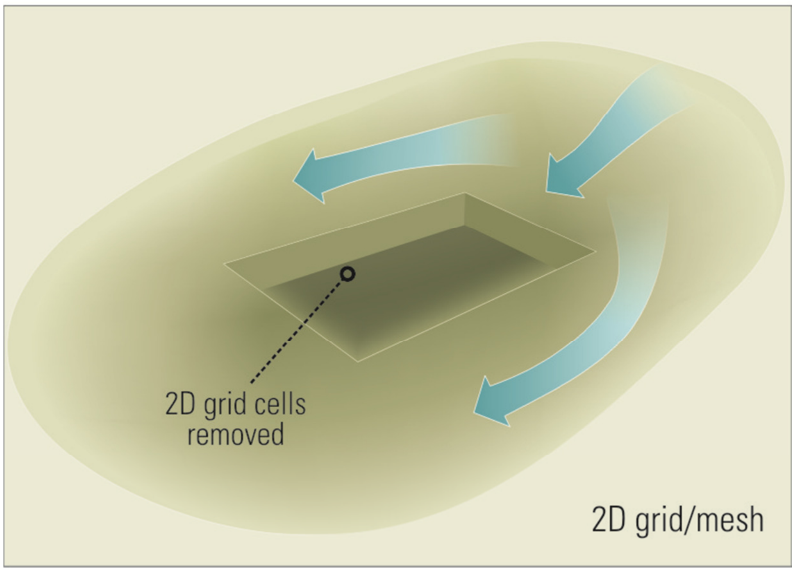


Figure 10-9 Model Elements Nulled

General recommendations are:

- Buildings are not represented by blocked out elements (either raised elevation or nulled) unless they can be demonstrated to exclude water inflows.
- If the building is sited on a pad, then the element ground elevations may need to be set to the pad or building floor level. If the underlying DEM does not include buildings, then the element elevations may need to be raised. If the DEM includes buildings, the DEM elevations are usually representative of the building roof, and the model's element elevations will need to be lowered.
- The representation of the areas surrounding the buildings (e.g. gardens and fences), will need careful consideration, since these will generally not exclude flood water.

10.11.4. Increased Roughness

Increasing the bed resistance parameter is a commonly used method for representing the increased energy dissipation of water flowing through and around buildings, gardens and fences. This is especially applicable where there will be flow around the buildings and the detailed representation of the flow around and into individual buildings is not required. It is shown diagrammatically in Figure 10-10 and is often favoured over blocking out features such as buildings as it includes the storage effects of the building being inundated, and is relatively easy to apply. The Manning's 'n' (or similar) parameter is also universally offered by all 2D software and is commonly used to represent the impact of buildings on flow behaviour. Figure 10-6 shows output for a model where higher roughness was used for houses and gardens and a much lower roughness for roads.

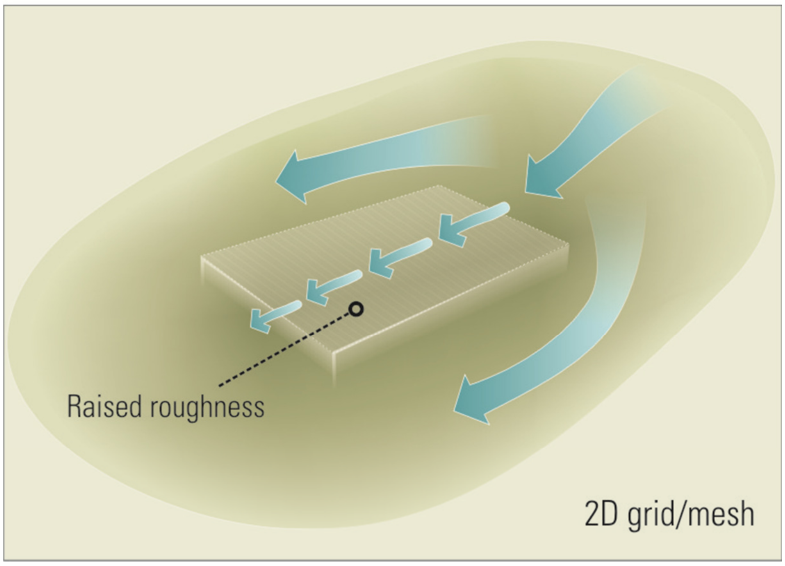


Figure 10-10 Model Roughness Value Increased

The value of Manning's 'n' needs to be established and spatially varied depending on the element resolution. The Manning's 'n' can represent a combination of a number of buildings lumped together, or can be used to represent individual buildings, which can convey flood water, depending on the size of the building or the grid/mesh resolution of the model. For example, a lower Manning's n value can be used for houses and gardens (combined as one n value), compared with a higher value for commercial properties on the basis that residential areas are less “dense” than commercial areas.

Varying the Manning’s n value also works well when the resolution of the 2D elements is coarse. For example, a fixed grid model using 10m square elements will struggle to adequately represent the flow between buildings if the buildings have been blocked out, whereas varying the Manning’s ‘n’ value is somewhat less sensitive to this effect.

There are no definitive suggestions on recommended values for Manning’s ‘n’ to represent different urban features. As a rough guide, Table 10-3 provides indicative ‘n’ values previously used within the Australian industry.

Table 10-3 Guide to Mannings ‘n’ Values for Different Urban Features

Category	Manning’s n
Roads, buildings, gardens, fences combined	0.06 to 0.1
Buildings, gardens, fences combined	0.08 to 0.3
Buildings (within flooded areas)	0.1 to 0.5
Buildings (outside flooded areas in direct rainfall models)	0.015 to 0.02
Gardens	0.03 to 0.1
Fences	Variable, noted below
Roads	0.015 to 0.025

For the modeller it is also important to note:

- The Manning’s ‘n’ value for an urban area may change with element resolution. For very coarse elements, an n value may represent the combination of roads, buildings, gardens and fences, while for very fine elements, different n values can be specified for each of these features.
- For direct rainfall modelling (see Section 11), ‘n’ values may vary for buildings depending on whether they lie within flood prone areas or not. Outside flooded areas, a very low n value may be used for buildings to represent the fast runoff from the building’s roof, while inside flooded areas a high value is used to represent the much higher resistance to flow as water flows around and through the building. Model elevations should be set to building pad or floor level.
- For reasons such as in the previous point, having the ability to vary Manning’s ‘n’ with depth can be very useful in representing the different flow regimes that can occur within urban areas.

10.11.5. Increased Energy Dissipation

An alternative to using an increased Manning’s ‘n’ value, is to specify form or energy loss coefficients to represent the fine-scale energy dissipation within and around the building. This is arguably more correct in that the energy losses due to buildings and other urban obstructions are mostly due to the water contracting and expanding as it flows through, around and over an obstruction. Whilst a 2D scheme models some of these energy losses such as the expansion of water downstream of the building, the fine scale losses that are not well represented, need to be included, hence the need for additional energy dissipation.

As for the increased Manning's 'n' option, no guidelines are as yet available for appropriate energy loss value(s). To use this approach the software needs to offer the ability for the modeller to add additional energy losses as a percentage of the dynamic head ($v^2/2g$), or some other similar approach.

10.11.6. Representing Buildings as Exterior Walls Only

A possible approach for representing buildings is to raise the sides of elements along the building's exterior walls. The walls will deflect the water, and provided that there is a break in the wall, the water enters the building to represent the storage effects. A consistent approach would need to be adopted as to whether the upstream or downstream sides of the building are open to let the water in. The example shown in Figure 10-11 has the opening on the downstream side of the building.

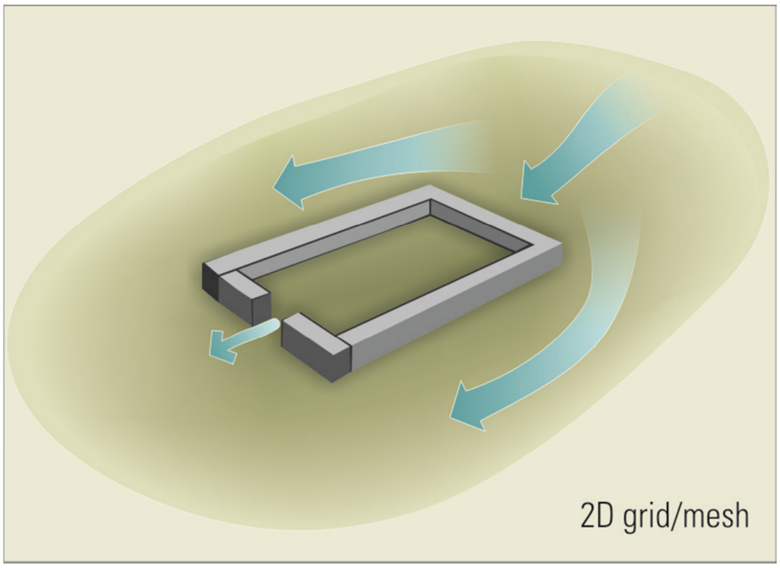


Figure 10-11 Representation of Buildings as exterior walls only

As with the approach of blocking out elements, a sufficiently fine element resolution is required to ensure adequate representation of the flow patterns around the building walls. This approach also overcomes the problem of having a "hole" in the water surface as the buildings still become flooded.

Of note is that if the building is open at the upstream side, the water level inside the building will be representative of the upstream energy head. If open on the downstream side a lower water level, representative of the downstream water level, will result. Therefore the location of this opening needs to be considered carefully and placed appropriately.

For the software to be able to offer this option, the element sides need to be able to modified in height independently of the centre element elevation, and the elevated element sides act as a thin (essentially zero thickness) barrier to flow.

10.11.7. Representing Buildings as "Porous" Elements

Another approach to model buildings is to specify the element sides as being "porous" or partially blocked. The 2D element sides within the building outline are constricted to represent the blockage of interior and exterior walls, and other obstructions. This is shown in 10-12.

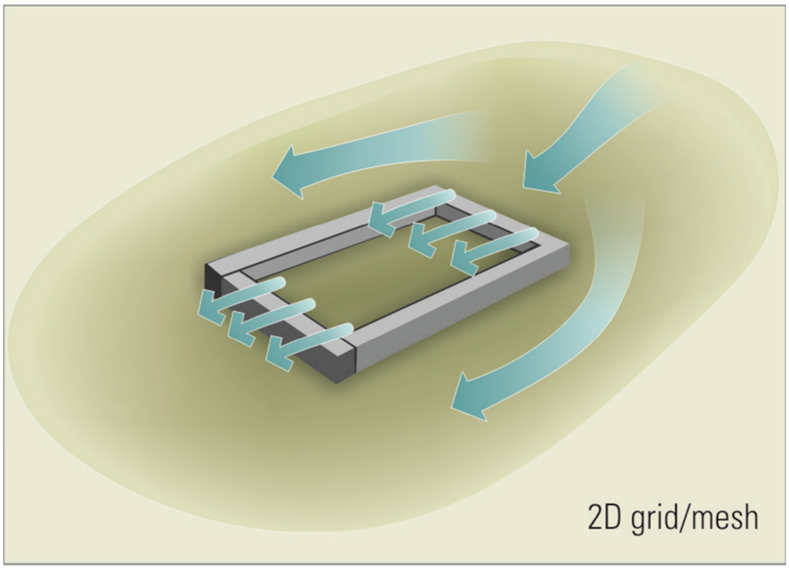


Figure 10-12 Representation of Buildings as Porous Walls

The advantage of this option is that the velocities through the building will be higher (due to the constricted element sides), which will produce a higher, and arguably more appropriate measure of the flood hazard and risk to life.he software would need to have the capability of adjusting the flow widths of the element sides to model partial blockages, without reducing the element's storage.

10.11.8. Fences and Other "Thin" Obstructions

Fences can cause significant blockages to floodwaters and they have the added complication of tending to collapse during a flood. The risk of collapse depends on a range of factors such as the flow velocity, the amount and type of debris build-up and the type and condition of the fence. This means that there may be a significant constriction to flow until it is dramatically reduced at one point in time. These complex conditions are difficult to model. They may also be partially open (eg. a picket or chain wire fence), and may also become blocked with debris. If the floodwaters are sufficiently high they will be overtopped and may act like a weir until the fence is damaged or destroyed.

Because of the relatively low and weak condition of fences, they have the most significant impacts on shallow slow moving flow and flat terrain. In these cases, fences may be a key controlling factor in flood behaviour.

Blocking out whole elements is not a good option for fences unless the element size is very small or long (flexible mesh elements). Increasing Manning's 'n' is also not recommended although it may be the only option available to the modeller depending on the software used.

To represent fences, the following software features are recommended:

- The ability to raise the element side elevations to the height of the fence. This effectively models the fence as a thin, or zero thickness obstruction (ie. does not affect storage).
- Automatic switching with upstream controlled weir flow across the element side if overtopping occurs.
- Partial blockage of the element side below the top of the fence to model partially open fences.
- The ability to apply additional energy losses that are likely to occur.
- If required, the ability to collapse element sides based on time, water depth or water level difference triggers.

10.11.9. Effect of Viscosity (Sub-Grid Scale Turbulence) Term

The eddy viscosity term (see Section 10.6) becomes particularly relevant with increasingly finer element resolution (the term is proportional to the inverse of the element length squared, therefore, the smaller the element, the greater the influence). The term also only has any influence where there is a change in the velocity direction and/or magnitude, such as flow around buildings, into and out of structures, and at bends. It also can have an influence where there are marked changes in roughness such as between a stream's waterway and its (highly vegetated) much rougher banks. It can therefore be an influential parameter, and solution schemes that do not include this term may be unsuitable.

It is not recommended to use viscosity coefficients outside of the ranges recommended by the software and in the literature as this may distort the results.

10.11.10. Effect of Fixed Grid Orientation and Element Resolution

Fixed grid 2D models (i.e. those that use a grid of square elements) may block or artificially choke narrow flowpaths when the orientation of the flowpaths between buildings and fences are oblique to the 2D grid orientation. Flexible mesh elements have more flexibility in this regard, however, adequate mesh resolution is still required.

This issue is of particular relevance if using:

- the blocking out elements to remove buildings;
- the raising of element side elevations to represent the building walls; and
- if fences are being represented as thin barriers by raising the element sides.

General suggestions are:

- If a flowpath (eg. between buildings) is a critical one, preferably three or more fixed grid cells should be active across the flow path (ie. perpendicular to the flow direction). For variable meshes, the use of two or more elements is desirable.
- If the flowpath is minor (ie. the buildings are located in a backwater or the percentage of flow carried along the flowpath is insignificant) a coarser resolution may suffice. Note that if there

are numerous minor flowpaths, then the cumulative effect of the flowpaths should be considered when assessing their relevance.

11. CHAPTER 11- DIRECT RAINFALL

11.1. Introduction

The application of rainfall directly onto the 2D domain is known as the direct rainfall approach or 'rainfall on the grid'. It attempts to remove, at least in part, the need for a separate hydrological modelling package.

Currently, direct rainfall is a relatively new technique. While this technique entered into mainstream commercial 2D modelling over the last 10 years, it has increased in popularity in recent years.

Only a limited amount of research has been undertaken in this area. Most studies undertaken to date (such as Caddis *et al.* (2008), Rehman *et al.* (2003), Clark et al (2008) & Rehman *et al.* (2007)) have provided comparisons between traditional hydrological models and the direct rainfall method. However, there has been very little research into the comparison of the direct rainfall method with gauged catchments. This is partially due to the limited number of gauged catchments in Australia, in particular in urban environments, where this approach is currently being utilised extensively.

By comparison, traditional hydrological models have been used extensively in Australia and have been verified against a number of gauged catchments. They have been shown to reproduce catchment flows in a number of different catchment conditions in Australia, provided the appropriate parameters are selected.

It is difficult to determine the accuracy of direct rainfall models through the comparison to traditional models alone. Rehman *et al.* (2007) demonstrated that there is as much difference in discharge time series between two different traditional hydrological models, as there is between direct rainfall and traditional hydrological models. This highlights some of the issues associated with selection of appropriate parameters for traditional hydrological models (as well as direct rainfall models) when no gauge data is available.

Until further research is undertaken, thorough checking of direct rainfall models should be undertaken. This section of the report outlines the key issues associated with the direct rainfall approach, including model setup techniques, viewing of results and verification. It provides some suggestions and recommendations, although these should be considered with the knowledge that available research in this area is limited.

Throughout this section, three key terms are utilised:

- Direct Rainfall – referring to the application of rainfall directly onto a 2D domain
- Traditional hydrological models – referring to lumped rainfall-runoff models, which are more traditionally utilised for hydrological modelling.
- Traditional hydraulic models – referring to models where the inflows are derived from traditional hydrological models, rather than from direct rainfall.

11.2. Concepts

11.2.1. Background

In traditional flood modelling, separate hydrological and hydraulic models are constructed. The hydrological model converts the rainfall within a sub-catchment into a peak flow hydrograph. This flow hydrograph is then applied to the hydraulic model, which estimates flood behaviour across the study area.

In the direct rainfall approach, the hydrological model is either partially or completely removed from the process. The hydrological routing is undertaken in the 2D domain, rather than in a lumped hydrological package.

11.2.2. How it Works

Direct rainfall works through the application of rainfall directly onto the 2D domain. The rainfall depth in a particular timestep is applied to an individual grid/mesh cell, and the 2D model utilises its internal hydraulic calculations to determine the runoff from this grid/mesh cell.

A simplified conceptualisation of this process is to visual each grid/mesh cell as a sub-catchment, similar to sub-catchments in traditional hydrological modelling package (refer Figure 11-1). In a similar manner to traditional hydrological modelling, runoff from an individual grid/mesh cell will be dependent on:

- Grid/mesh cell area
- Rainfall depth
- Grid/mesh cell roughness
- Slope between neighbouring grid/mesh cells
- Rainfall losses

The key difference is that in a 2D model the runoff can proceed in four directions, depending on the slope and water level in neighbouring cells. In addition, not all rainfall may be converted to runoff, as the 2D grid/mesh cell may provide storage.

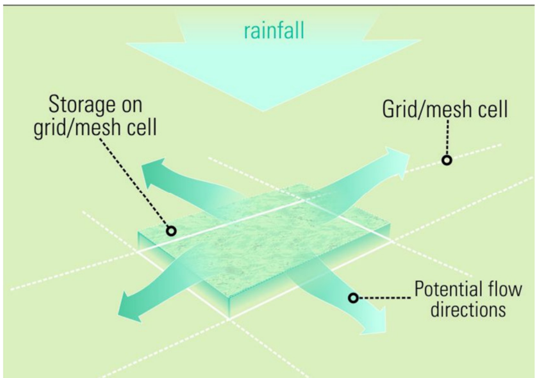


Figure 11-1 Conceptualisation of Direct Rainfall

Routing of the runoff through the larger catchment is then undertaken using the hydraulics within the modelling package. This is equivalent to lags or channel routing in traditional hydrological modelling packages. The key factors influencing the timing and magnitude of flow arriving at the catchment outlet include:

- Roughness of grid/mesh cells along the flowpath
- Available storage areas along the flowpath
- Slope of the 2D terrain along the flowpath
- Obstructions & hydraulic structures along the flowpath.

11.2.3. Primary Reasons for using the Technique

This approach can potentially be utilised for any flow analysis. Currently, the general reasons for using this technique are:

- Flat terrain and catchment
- Cross catchment flows
- Detailed urban studies
- Simplification of modelling process

There are both advantages and disadvantages of using the direct rainfall method, discussed as follows.

11.2.3.1. Advantages of the Direct Rainfall

There are a number of advantages associated with the use of the direct rainfall approach:

- Where a 2D model can be constructed to cover the whole catchment, use of the direct rainfall approach can negate the need to develop and calibrate a separate hydrological model, thus reducing overall model setup time.
- Assumptions on catchment outlet locations are not required. When a traditional hydrological model is utilised, an assumption is required on where the application of catchment outflows are made to the hydraulic model.
- Assumptions on catchment delineation are not required. Flow movement is determined by 2D model topography and hydraulic principles, rather than on the subcatchment discretisation, which is sometimes based on best judgement and can be difficult to define in flat terrains.
- Cross catchment flow is facilitated in the model. In flat catchments, flow can cross a catchment boundary during higher rainfall events. This can be difficult to represent in a traditional hydrological model.
- Overland flow is incorporated directly. Overland flow models in traditional hydrological packages require a significant number of small sub-catchments, to provide sufficient flow information to be applied to a hydraulic model.

11.2.3.2. Disadvantages of Direct Rainfall

There are a number of disadvantages associated with the use of the direct rainfall approach:

- Direct rainfall is a new technique, with limited calibration or verification to gauged data. Caution and detailed checking is needed in the application of this approach.
- Potential significant increase in hydraulic model run times. Hydrological models on their own generate peak flows significantly faster than direct rainfall models (although this may be compensated for by the fact that traditional hydrologic models take longer to build).
- Requires digital terrain information. Depending on the accuracy of the results required, there may be a need for extensive survey data, such as aerial survey data.
- Insufficient resolution of smaller flowpaths may impact upon timing. Routing of the rainfall applied over the 2D model domain occurs according to the representation of the flowpaths by the 2D model. Higher in the catchment, these flowpaths become smaller and it is likely that they will not be as well-represented by the 2D model as they may exist on a sub-grid scale. This may affect timing of runoff routing. Methods for overcoming this are discussed throughout this chapter and in Chapter 10.
- The shallow flows generated in the direct rainfall approach may be outside the typical range where Mannings ‘n’ roughness parameters are utilised. Some of the challenges and potential solutions are discussed in Section 11.3.1.

11.2.4. Methods of Application

Currently, there are primarily two methods by which direct rainfall is applied:

1. Applying direct rainfall over a portion of the catchment, with flow hydrographs for the remainder described in a traditional hydrological package.
2. Applying direct rainfall to the 2D model over the entire catchment (the 2D model must therefore cover the entire catchment).

The first approach can utilise the benefits of both traditional hydrological models and the direct rainfall approach. By limiting the size of the 2D domain, model run times can be reduced, but flows over the 2D domain do not need to be defined separately. While it still requires two separate models, it can reduce the sometimes large assumptions associated with the application of flows within the 2D domain, particularly in complex urban areas and/or flat terrains.

The second approach requires only one model, with the traditional hydrological model not needed. However, the 2D model does need to cover the entire catchment area. Run times will also be longer than if a separate hydrological model provided flows.

11.3. Model Setup

11.3.1. Model Roughness

Details on the application of model roughness in 2D models are provided in Chapter 5. Details that particularly apply to the direct rainfall method are discussed in this section. One of two approaches for model roughness is generally adopted when using the direct rainfall method:

- Constant roughness, or
- Depth-varying roughness

Constant roughness models do not take into account changes to roughness with changes in flow depth (refer Chapter 5). The direct rainfall approach introduces shallow flows across the entire 2D domain, and in doing so, magnifies the impact of depth-variations in roughness for shallow flows.

A depth-constant roughness parameter may under-estimate or over-estimate the effective roughness depending on the surface type, and hence result in a faster or slower routing of catchment runoff. Research comparing direct rainfall models using constant roughness with traditional hydrological models (e.g. Rehman *et al.* (2007), Caddis *et al.* (2008), Clark *et al.* (2008)) under different catchment conditions has shown that the direct rainfall approach generally appears to result in longer runoff times. For some of these case studies, this may be partly attributed to the impact of hydraulic controls such as bridges and culverts along the routing flowpath, which are absent in a traditional hydraulic model. However, for relatively simple terrain and roughness conditions, Rehman *et al.* (2003) showed that the direct rainfall approach still resulted in slightly longer runoff times than for traditional hydrological modelling, as shown in Figure 11-2

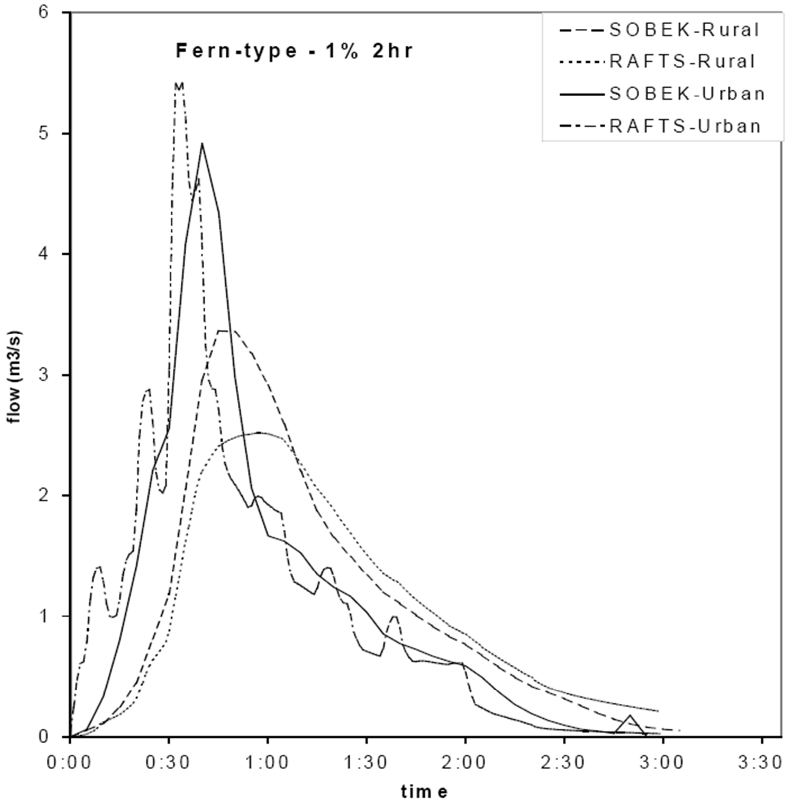


Figure 11-2 Comparison of Direct Rainfall (SOBEK) with Traditional Hydrology (RAFTS) (courtesy Rehman *et al.* (2003))

An alternative approach is the utilisation of a depth-varying roughness parameter (refer Chapter 5). Generally, catchment-wide 2D models currently lump roughness parameters based on land use and other features. Urban 2D models, for instance, generally adopt a singular roughness

value to describe a residential area, rather than describe individual elements such as fences, grassed areas, paved areas and other features. This lumping of roughness may make it difficult to create a meaningful roughness function with depth. Additional delineation of features such as buildings as different roughness elements or blockages may overcome some of these issues.
Some recent research on depth varying roughness with Direct Rainfall (such as Caddis *et al.* (2008)) suggests that the roughness function should be increased for different surface types for shallow flows, similar to typical applications of depth varying roughness. However, Caddis *et al.* (2008) found that lowering roughness values for shallow flows for lumped areas (such as residential and commercial areas defined with a single roughness) provided a better match to a traditional hydrological model (Figure 11-3). It is important to highlight that lowering of the Manning's 'n' values for shallow flows is contrary to roughness theory and observation (discussed in the first part of this Section). However, the work by Caddis *et al.* (2008) was undertaken for a single catchment, and characteristics specific to that catchment may have resulted in a lowered 'n' value over lumped areas being applicable. It is not recommended that readers apply this finding without additional investigation.

The challenge with the depth-varying roughness approach, as per Chapter 5, is in determining an appropriate relationship between the roughness parameter and depth. This is further confounded by some of the issues raised above with Direct Rainfall.

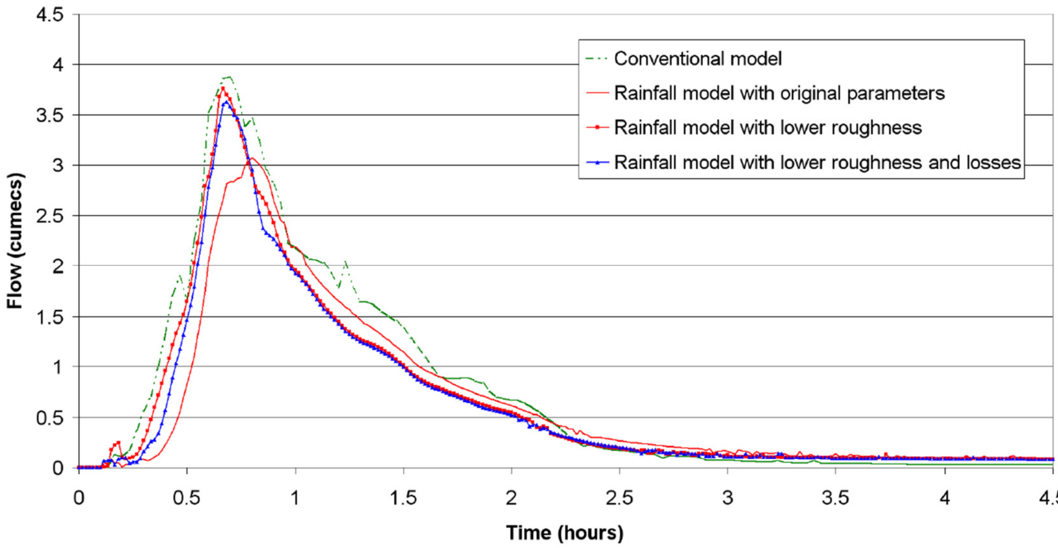


Figure 11-3 Comparison of Flows from Traditional Hydrological model with Direct Rainfall modelling (courtesy of Caddis *et al.* (2008))

The following provides some recommendations on the application of both roughness parameter methods. They can potentially be applied by an experienced modeller if a reasonable calibration/ verification is not achieved.

- The constant roughness method may under- or over-estimate the effective roughness for catchment runoff where shallow flow dominates. This may result in an early peak or a delayed peak in the flow hydrograph respectively. Therefore, applying a different roughness to the main "floodplain" areas or flowpaths when compared to the "catchment runoff" areas may be warranted. An iterative approach may be needed to identify areas where shallow flow dominates.
- The modeller can develop a "library" of roughness parameters for either the depth-varying or constant roughness parameters, until more detailed research is available. For example,

where gauged catchment data is available for a particular study, record the calibrated roughness parameter values associated with various landuses and these could be used a guide for new studies.

11.3.2. Losses

Not all rainfall falling on the catchment will end up as flow at the catchment outlet. In the process of rainfall becoming runoff and flow, catchment water losses occur due to a number of processes, including interception, infiltration, evaporation, transpiration and storages. Some of these loss processes predominately occur while the rain falls (e.g. interception) and some occur anytime in the hydrological cycle (e.g. transpiration, which can occur for many months after the rain has ceased to fall). The actual amount of loss due to these processes is dependent upon many factors, including vegetation type, soil type, initial saturation levels, exposed water surface and so on.

This section focuses on the way in which these losses are incorporated into direct rainfall models. The following broad methods are outlined in this section:

- Rainfall Loss Models
- 2D Loss Models
- Integrated Groundwater Models

Some of the challenges discussed in the following sections related to losses are not strictly a 2D model issue, but rather a result of scaling. Traditional hydrological models consider catchments lumped together and parameters used define the overall catchment as a whole. Frequently, any errors or assumptions in the traditional models are effectively averaged out, as the internal catchment flows were not reported.

By comparison, the direct rainfall method is effectively delineating the catchment into a number of grid/mesh cells. Assumptions on losses are required spatially. However, information may not be available to adequately define these parameters spatially and at a grid/mesh cell level. Catchment filters are one option to remove the results within small sub-catchments. These are discussed in Section 11.5.1.

11.3.2.1.Rainfall Loss Models

Rainfall loss models incorporate losses due to interception, evaporation, infiltration, initial storage and other processes by removing a portion of the rainfall applied to the model. That is, rainfall loss models can only apply losses while the rain falls. These models can use a variety of methods to remove these losses, such as:

- Initial and continuing loss
- Constant loss rate
- Loss as a constant fraction of rainfall
- Infiltration equations

While each one of these loss methods was originally developed for traditional hydrological modelling, they have been applied more recently to the 2D direct rainfall method. Details on these loss methods when used in traditional hydrologic modelling are contained in ARR (1987).

One of the loss processes incorporated in rainfall loss models is the storage loss. This is the loss of catchment water that occurs close to the start of the event where catchment storages (both small and large) start to fill. The terrain of the 2D model will incorporate some aspects of catchment storages. The number and type of these storages will be dependent on the resolution of the 2D domain, together with the resolution of the underlying survey data. A smaller grid/mesh cell will result in greater definition of the terrain (assuming the underlying survey data is of sufficient accuracy), and hence a greater number the localised small storages will be represented in the 2D domain. Thus, rainfall loss due to storage will only need to account for those storage losses occurring at the sub-grid scale. This means that the initial loss of the initial and continuing loss model should be lower in a direct rainfall model when compared with a traditional hydrological model.

Caddis *et al.* (2008) found that direct rainfall results compared better to traditional hydrological model results if the roughness values <u>and</u> losses were lower than the traditional values. However, based on the results presented in Caddis *et al.* (2008), the impact that losses had on the flow hydrograph were overshadowed by the impacts that roughness had on the flow hydrograph.

11.3.2.2.2D Loss Models

2D loss models remove water from the 2D domain, rather than from the rainfall being applied to the 2D domain. This means that runoff is exposed to potential losses while it is being routed through the 2D domain, rather than only during the initial application to the 2D domain (as discussed in the previous section).

The main advantage of the 2D loss model is that it can potentially provide a better representation of the physical system, and fully utilise the benefits of 2D runoff routing. The nature of the 2D model allows evapotranspiration and infiltration to be defined based on the area of inundation, rather than defining these based on the depth of rainfall.

The main disadvantage of a 2D loss model is that it may require significantly more information than a rainfall loss model. This information includes spatial definition of parameters for infiltration equations based on soil parameters, together with estimation of evapotranspiration. By comparison, a rainfall loss model generally lumps these processes into a loss value (with the exception of infiltration equations). The additional information required for the 2D loss model does not necessarily translate into a more accurate model. Estimation of these parameters may not always be straightforward, as information (such as soil parameters, vegetation etc) may not always be available, or spatially well-defined. The assumptions behind these parameters may be no more accurate than the lumped loss values in the rainfall loss models. Distributed hydrological models face similar challenges.

11.3.2.3. Integrated Groundwater Models

An integrated groundwater model effectively extends the loss methods to the next level of detail beyond a 2D loss model. It involves linking a groundwater model to the 2D surface water model. These types of models have a two way connection, allowing water from the 2D surface water model to enter the groundwater model (i.e. infiltration) and allowing water from the groundwater model to enter the 2D surface water model (i.e. baseflow).

Both a rainfall loss model and a 2D loss model assume that any water that has entered the groundwater system has effectively been removed from the surface runoff, whereas the integrated groundwater model is not limited by this assumption. However, this assumption is generally not critical for most flooding applications, as the groundwater processes are substantially slower than the surface water component. It does become important in long-term simulations.

11.3.2.4. Pre-Wetting of 2D Models

As noted above, direct rainfall models incorporate some element of depression storages within the model. The use of raw DTM data (such as ALS) can in some cases also lead to an over-estimation of depression storage and underestimation of hydraulic conductivity. This problem is particularly pronounced in catchments subject to small magnitude, short burst duration design rainfall where the rainfall volume is relatively low.

In order to increase the hydraulic conductivity, a technique known as “pre-wetting”, can be used. Pre-wetting is the application of a rainfall burst to a DTM with the purpose being to fill unconnected depressions in the DTM. Once the runoff excess has drained, water level results of the final time step are then used as the model topography in subsequent runs.

Gray and Ball (2009) applied this methodology to improve the calibration of flows against a gauged catchment. However, it is noted that this was applied to a single catchment only, and therefore it is difficult to draw conclusive findings from this study alone without further research.

11.3.2.5. Kerb and Gutter Representation

Stamping a preferred flow path into a model grid/mesh (at the location of the physical kerb/gutter system) may produce more realistic model results, particularly with respect to smaller flood events that are of similar magnitude to the design capacity of the kerb and gutter. Stamping of the kerb/gutter alignment begins by digitising the kerb and gutter interval in a GIS environment. This interval is then used to select the model grid/mesh elements that it overlays in such a way that a connected flow path is selected (i.e. element linkage is orthogonal). These selected elements may then be lowered relative to the remaining grid/mesh. The methods' usefulness has been confirmed in only one tested application (Gray and Ball, 2009), and should be regarded as a suggestion only. Modellers will need to assess the applicability of this method to their own situation.

11.3.3. Direct Rainfall in Urban Areas

Buildings can be represented within 2D models using a number of methods, which are described in detail in Chapter 10. One method is to “null” or block out the footprint of the building so that water is unable to enter the 2D model cells that represent the building. This approach should not be used in conjunction with direct rainfall unless the software package is able to account for the runoff from the nulled cells in another way. Direct rainfall applies rainfall to every grid/mesh cell within the 2D domain. Removal or nulling of grid/mesh cells will result in a loss of runoff volume. This may become significant if there are a large number of buildings in the catchment represented as nulled cells within the 2D model. Some software packages have methods of accounting for this potential loss in volume (for example, by applying the equivalent rainfall volume over the active cells). Although, this may result in rainfall being spread unrealistically across the catchment, with relatively less applied in densely developed areas (more building footprints nulled) and more applied in less developed areas. Thus, it is recommended that the combination of direct rainfall and nulled model grid/mesh cells be avoided if possible. If not possible, the modeller should ensure that the volume and spatial distribution of the rainfall across the catchment is appropriate.

11.3.3.1. Response Time

In an urban environment, roofs of buildings are generally connected to either the stormwater system or street gutter via the roof drainage. Depending on the reliability of the roof drainage and the stormwater system, this can result in runoff from a roof arriving at the catchment outlet more rapidly than the runoff from the rest of a property.

Traditional hydrological models overcome this through lumping of all of the catchment parameters, and only providing data at the catchment outlet. In the direct rainfall approach, the replication of this phenomenon can be difficult. It is generally not possibly to model the roof drainage, unless modelling is undertaken on the micro scale.

The importance of this behaviour will be dependent on the analysis being undertaken. If infrequent (large) events are being analysed, and the stormwater infrastructure has a capacity for only frequent events, then the issue may not be as critical. However, if this behaviour is affecting results, one of two options is recommended. It is important to note that neither of these approaches has undergone significant testing. It is recommended that results be checked for sensibility and that sensitivity testing is undertaken. More research is required into this area.

1. Averaging Model Roughness in Catchment

If stormwater infrastructure is not incorporated within the 2D model, the roughness across the entire catchment (including buildings) may be adjusted to average out the difference in timing between the overland flow and the flow through the stormwater infrastructure. The resulting flow hydrograph will be an averaged hydrograph of the two, rather than the characteristic twin peak discharge behaviour of the stormwater infrastructure and the overland flow. Again, this approach is the only option if detailed information on the stormwater infrastructure is not available, or if stormwater is not being modelled in detail within the catchment. It should be undertaken by lowering the roughness in the upper catchment areas, not in the flowpaths. It is important to understand that this is an approximate technique, and should be used with caution.

2. Incorporating Stormwater Infrastructure within the Model

Incorporating stormwater infrastructure into the model will ensure that the characteristic twin peak discharge will be present in model predictions. However, the roof areas as represented in the 2D model still will not replicate realistic behaviour. If the grid/mesh cells are raised to represent the roofs, then one solution may be to lower the roughness on the roof area in order to shorten the runoff time from the roof. However, this approach may cause instabilities in some models. In addition, the runoff from the roof will still need to proceed overland before reaching the street drainage, which may cause a relative delay in runoff timing compared to the physical system.

Another option is to lower the roughness either for the property or the building. It is recommended that this only be undertaken for the areas outside the major flowpaths and floodplain.

11.3.3.2. Model Stability

Another method of incorporating buildings into 2D models is to raise the grid/mesh cells to represent a building (refer to Chapter 10). However, when used in combination with the direct rainfall approach, this method can result in model stability issues in some software packages. This is due to the steep gradients between the cells representing the roofs and the cells representing the ground. These model stability issues can potentially result in longer model run times, erroneous results or volume errors (volume errors are discussed in more detail in Section 11.4.3). In general, these issues may be overcome by limiting the height by which the grid/mesh cells are raised. The grid/mesh cells should be raised a sufficient height to replicate the obstruction of flows, but not to create model instabilities.

Some modelling packages allow the application of the building runoff as a point source at a separate location or as a representative “downpipe”. In these situations, the roof area is effectively nulled for the computations.

Instabilities associated with raising grid/mesh cells in combination with direct rainfall can be avoided if buildings are modelled using increased roughness parameters.

11.3.4. Grid/mesh Cell Size

Grid/mesh cell size is an important parameter in undertaking any 2D modelling. As discussed in Chapter 6, grid/mesh cell sizes need to effectively represent the relevant flowpaths such that flow behaviour in the area of interest is replicated.

In 2D models using direct rainfall, flowpaths throughout the entire study area need to be modelled such that the rainfall applied to each cell is appropriately routed. In the upper catchment, there is a risk that a coarser grid/mesh will not replicate the smaller flowpaths in this area, and hence result in unrealistic flow extents and timings. Alternatively, a finer grid/mesh may produce a better representation of the actual flowpaths and sub-grid features but will increase run times, and therefore may not be feasible. It may be possible to use 1D elements to define some of these sub-grid features, but depending upon the number, perhaps not all.

Noticeable errors in the upper catchment can be reduced with the application of a catchment area cutoff, which is discussed in Section 11.5.1.

For very shallow flows in the upper catchments, there are also small sub-grid features that will affect the conveyance of flow. While to some degree these can be incorporated as a roughness parameter (such as the depth-varying parameters discussed in Section 11.3.1), they may have an effect on the flow behaviour.

11.4. **Calibration & Verification**

Currently, the direct rainfall approach is relatively new to the industry. There has been very little research into the direct rainfall approach, and only limited published calibration of the approach to a gauged catchment. This is not to say that the direct rainfall approach is any less accurate than traditional hydrological modelling. However, traditional modelling packages generally have a greater level of acceptance given their longer period of exposure in the industry and the fact that they have been used successfully in the calibration of a far larger selection of gauged catchments.

It is therefore particularly important that a calibration and/or verification be undertaken for models using the direct rainfall approach. In undertaking a calibration and/or verification of a direct rainfall model, there are only a limited number of parameters to alter:

- Roughness values defined within the catchment
- Losses assumed
- Grid/ mesh resolution

As recommended in Chapter 7, calibration using streamflow data should always be undertaken where this data is available. Unfortunately in most cases, streamflow data will not be available and without such data it is recommended that the model be verified using all of the following:

- Comparison with Alternative Models
- Indirect Calibration (where information is available)
- Volume Checks

11.4.1. Comparison with Alternative Models

This method of verification compares the results of the direct rainfall model with a traditional hydrological model. A traditional hydrological model is constructed for a few sample sub-catchments within the study area, and the flows from this model are compared with the flows in the direct rainfall model.

The sample sub-catchments should be representative of the different hydrological characteristics of the study area (such as slope, land-use etc). The outlet of sub-catchments should also be located such that it provides a good match with a measured outflow location in

the 2D direct rainfall model.
Sample sub-catchments are preferred to modelling of the entire catchment as:

- Flows at the outlet of the overall catchment are likely to be affected by hydraulic controls and other mainstream features. A significant amount of detail would be required in the traditional hydrological model to define these features.
- The construction of a detailed traditional hydrological model for the entire catchment would negate some of the time-saving benefits of undertaking the direct rainfall approach.

Comparisons of the results of the traditional hydrological model and the 2D direct rainfall model should look at the complete flow hydrograph, including the shape, the volume and the value and timing of the peak flow. The flow hydrographs of the models may not necessarily match. Both sets of models will rely on a number of underlying assumptions and it is recommended that the modeller choose a traditional hydrological model that they are familiar with.

An example of this type of verification was described in Swan & Thomson (2011), for three studies in NSW and one in Victoria. They compared both the timing of the flows as well as the peak flows and total volume of the storm from both a traditional hydrologic model and a 2D direct rainfall model. Some examples of these comparisons are provided in Figure 11-3. It is expected that these are the types of comparisons that would be required as a part of this verification technique.

The modeller should consider the following when comparing the results:

- Is the sub-catchment adequately described in the traditional hydrological model?
- Does the sub-catchment fall within the underlying assumptions of the traditional hydrological model?
- Are the underlying parameters of the hydrological model reasonable?

If the modeller is satisfied with the representation of the traditional hydrological model and the flow hydrographs are not providing a satisfactory match, consideration will need to be given to changes to the 2D direct rainfall model. A detailed description of calibration processes is provided in Chapter 7.

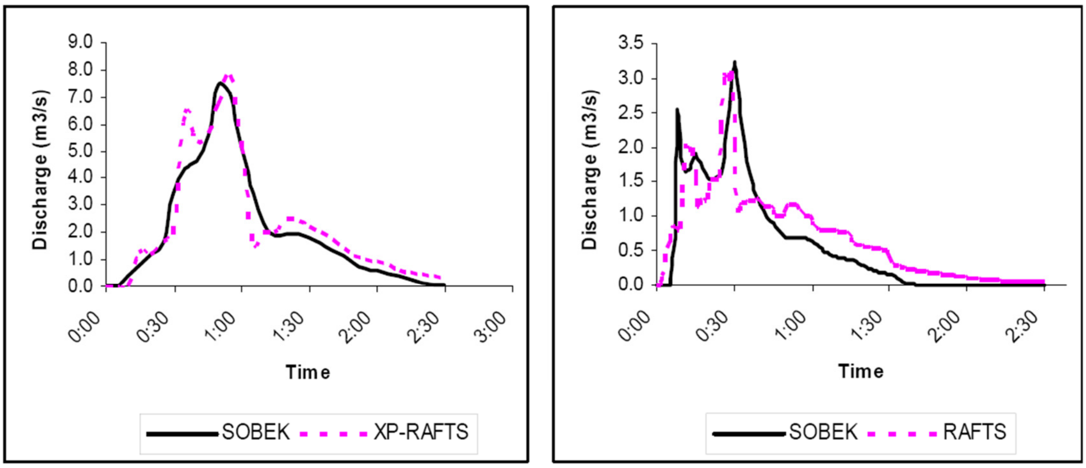


Figure 11-4 Comparison of Flows from Traditional Hydrological model with Direct Rainfall modelling (courtesy of Swan & Thomson (2011))

11.4.2. Indirect Calibration

An indirect calibration is undertaken through the comparison of peak water level information or other observations, and inferring the calibration of the estimated flows. Many traditional hydrological models are calibrated via a similar approach. This provides some confidence in the results produced.

A detailed discussion on the calibration of 2D hydraulic models is provided in Chapter 7. This process should be undertaken in conjunction with the verification with alternative models described in Section 11.4.1.

11.4.3. Volume Checks

Volume checks are a method of error checking the direct rainfall approach. The volume of water leaving the model through the downstream boundary should be equal to the amount of water that was applied (through direct rainfall and/or external boundaries), less losses and storages within the model.

Equation 11-1 Volume Balance $$V_o = V_i + V_B + V_R - V_L - V_S + E$$

V_o - Volume at the outlet
V_i – initial volume specified in model (such as lake or initial depth in channel)
V_B – Volume applied through external boundaries
V_R - Volume of rainfall
V_L - Volume of losses (rainfall losses or 2D grid/mesh losses)
V_S – Volume in storages
E – Error

Most 2D hydraulic models will produce some volume errors, particularly when the direct rainfall approach is utilised. If the error term in Equation 11-1 becomes a significant proportion of the volume at the outlet of the model, then this suggests that there may be numerical issues with the 2D model. Generally, errors within the range 0% to 5% of V_o would be considered acceptable. However, this is dependent upon several factors including modelling application and standard of results required (refer Chapter 4). The following troubleshooting points may assist in identifying the reason for an error in volume:

- The first check should be to review the schematisation of the model, to ensure that model schematisation is not causing model instabilities or volume errors. This might include schematisation of buildings and 1D/2D connections. If there are no significant issues with the schematisation, then the numerical parameters within the model should be reviewed.
- Each modelling package has different numerical parameters that can be adjusted to potentially reduce volume errors.
- Similarly, some modelling packages can potentially encounter problems with wetting and drying, or wetting & drying on steep slopes. The application of direct rainfall results in water on all the grid/mesh cells in the 2D domain, compared with more traditional 2D models where water was limited to the floodplain and flowpath areas. As such, wetting and drying

issues can become more pronounced in direct rainfall models. Similarly, wetting and drying on steep slopes can become more pronounced in direct rainfall models, where this issue may have been limited in traditional 2D models.

- For very shallow flows, some models assume that overland flow is not activated until the water depth exceeds a certain depth. This may potentially result in an effective loss of volume from the system. To overcome this, consideration may be given to modifying rainfall losses.
- Volume errors may also become more pronounced over longer duration storms. Therefore, volume checks should be undertaken across the full range of durations for a study area.
- If the volume errors remain high following adjustment of the numerical parameters and checking of the model schematisation, it might be that the model is not appropriate for the direct rainfall approach, or not appropriate for the particular catchment being modelled. It may therefore be necessary to utilise either a different 2D model or a traditional hydrological model.

11.5. Results

The use of direct rainfall can have an impact upon model results. The following sections outline some of these impacts and potential solutions.

11.5.1. Filtering of Results

The very nature of the direct rainfall approach results in the entire 2D domain effectively being inundated. This can lead to difficulties in interpreting results, as a decision needs to be made as to what portion of the 2D domain is representative of catchment runoff and what is representative of flooding.

There are a number of methodologies that may be used to help distinguish between catchment runoff and flooding, including:

- Depth filter – Constant
- Depth filter – Proportional to Peak Rainfall Depth
- Flow cutoff filter
- Velocity-depth filter
- Catchment area cutoff
- Small pond filter

This does not represent an exhaustive list of the possible filtering techniques, nor has there been research into the most suitable methods. This report does not specifically recommend any of these options, as each one has its own advantages and disadvantages. Further research may reveal that some approaches are more suitable than others.

11.5.1.1. Depth Filter – Constant

A constant depth filter removes any flood level results that are below a certain depth. For example, any results below a depth of 0.10m are removed. The exact depth chosen may depend on the particular catchment analysed and the overall objectives of the study being undertaken. For example, if overland flow is important, then a smaller depth filter might be applied. This approach is generally easy to apply, with the most challenging decision being the actual choice of filter depth.

The limitations of this approach include:

- Potentially removing areas that are flood affected
- Potentially removing key features of the flow behaviour, such as shallow overtopping flow across a weir or high velocity/low depth flow down the length of the road. This can result in the apparent discontinuous flow behaviour in areas where flow and extent is in fact continuous.

11.5.1.2. Depth Filter – Proportional to Rainfall

This approach is similar to the constant depth filter, with the depth filter factored based on the design rainfall.

11.5.1.3. Velocity-Depth Filter

A velocity-depth filter may overcome the limitations mentioned in relation to the depth filter, such as removing high velocity / low depth regions of flood behaviour. There is no known research into the application of velocity-depth filters with direct rainfall. The following are some suggestions for its application:

- Choose a velocity-depth filter based on the requirements of the study and the results that need to be viewed. For example, a v*d filter of 0.05 will result in areas of 1 metre depth and a velocity of 0.05m/s being included. However, it will exclude areas where the depth is 0.5m and velocities are 0.05m/s.
- Preliminary investigations of this method have suggested that this may potentially produce good results in steeper areas. However, in flatter areas where slow moving storage is more typical, this method tends to filter out areas that may be required due to low velocities.
- Based on preliminary investigations, it is perhaps most appropriate to utilise a combination of a velocity-depth filter with a depth filter, where the output location needs to satisfy both filters before being removed. For example, a minimum v*d of 0.05 could be utilised together with a minimum depth filter of 0.3m. Using the previous example, areas with a depth of 0.5m and velocity of 0.05m/s would be included due to the depth filter threshold not being satisfied. Similarly, areas with a depth of 0.05m and a velocity of 1m/s would also be included due to the velocity-depth threshold.

11.5.1.4. Flow Cutoff Filter

A flow cutoff filter removes results in areas where the flow is below a certain threshold. The flow threshold selected is dependent on the objectives of the study.
There are two potential approaches with this methodology:

- **Individual grid/mesh cell filtering.** Under this approach, a flow threshold is set for each individual cell. Note that this approach is effectively the same as the velocity-depth filter (as the flow is effectively $v*d*cellsize^2$).
- **Flowpath filtering.** Under this approach, whole flowpaths are analysed and are filtered according to the chosen threshold. However, the measurement of flow spatially within the 2D model is challenging, and may become a manual process.

11.5.1.5. Catchment Area Cutoff

A catchment area cutoff seeks to remove areas of inundation in the upper portions of a catchment, where flow behaviour may not be important. In addition, it has the potential to remove areas where spatial assumptions on losses and roughness parameter/s may cause unrealistic flow behaviour. In the upper catchment, flows will generally be confined to smaller flowpaths. Depending upon the schematisation of the model, the relative size of the grid/mesh cell may increase in comparison with the flowpaths from the lower to the higher catchment. . Therefore, flowpaths may not be as well defined by the 2D grid/mesh in the upper catchment.

Under this methodology, there is a need to define sub-catchments in the upper portions of flowpaths within the study area, which may remove some of the benefits of the direct rainfall approach. For example, for an individual flowpath, results in the upstream 1ha of the catchment might be removed. This would require a 1ha catchment area to be defined for each of the flowpaths in the study area. The determination of the catchment area could be based on a number of factors. One option would be to base the size of the excluded catchment area on the grid/mesh cell size applied and the appropriateness of this grid/mesh cell size in defining the flowpaths. There is potential for relevant backwater areas to be removed in using this method and this would need to be assessed on a case by case basis.

This approach would also need to be combined with another form of filtering applicable to the downstream areas, as it effectively only filters the upper catchment results.

11.5.1.6. Small Pond Filter

This method aims to filter small discrete ‘ponds’ of inundation. These ponds are commonly caused by localised depressions within the grid/mesh elevations. Ponds are filtered according to a size threshold. For example, it may be appropriate to remove ‘ponds’ of inundation that have an area less than the equivalent area of 4 cells.

It is likely that this approach would need to be combined with another form of filtering.

11.5.1.7.Summary

Currently, appropriate filtering of results is most likely to be achieved by a combination of options presented here. It is recommended that each combination should include some form of depth filter.

11.5.2. Flood Mapping

Under the above filtering methodologies, additional adjustments will be required to produce sensible results. These adjustments should be undertaken based on engineering judgement, and may include deletion of small pockets of inundation, inclusion of areas of shallow flow with high velocities, filling in of buildings (if they are modelled as blocked) for flood extents, and so on. Further details on flood mapping is provided in Chapter 8.

11.5.3. Overland Flow versus Mainstream Flooding

The definition of overland flow when compared to mainstream flooding is not exact. From the perspective of 2D modelling, the separation of these flows is generally not an important issue. However, a number of policies and controls by government organisations are reliant upon the knowledge of whether an area is affected by overland flow or mainstream flooding. Sometimes an area can be affected by overland flooding and then affected by mainstream flooding later.

One person's overland flow is another person's mainstream flooding.

One of the challenges of the direct rainfall method is that all flowpaths within a catchment are defined. This can create challenges when preparing model results for end-users, who may require a distinction between overland flow and mainstream flooding. Currently, there is no solution to this challenge. However, it is an important issue that should always be raised with the end-user.

11.5.4. Sensitivity Testing

A detailed discussion on sensitivity testing is presented in Chapter 7. When utilising the direct rainfall method, sensitivity testing should specifically include the following key parameters:

- Rainfall losses
- Model roughness (and depth-varying roughness, where applicable)
- Representation of the roof to catchment outlet drainage system
- Variations in rainfall (e.g. spatial application of rainfall, proportioning of historical rainfall etc)

These sensitivity assessments on the direct rainfall method should focus on the flow generation, rather than the peak water levels or depths

11.6. References

Caddis B M, Jempson M A, Ball J E and Syme W J (2008). *Incorporating Hydrology into 2D Hydraulic Models – The Direct Rainfall Approach*, 9th National Conference on Hydraulics in Water Engineering, Engineers Australia, Darwin, 23-26 September.

Clark K M, Ball J E and Babister M K (2008). *Can Fixed Grid 2D Hydraulic Models be used as Hydrologic Models?*, Water Down Under 2008 Conference, Adelaide

Gray S and Ball J, 2009, Coupled One and two dimensional modelling in urban catchments - reducing uncertainty in flood estimation,32nd Hydrology and Water Resources Symposium Newcastle

Rehman H, Zollinger M and Collings G (2003). *Hydrological vs Hydraulic Routing – Possibilities with Two-Dimensional Hydraulic Modelling*, 28th International Hydrology and Water Resources Symposium, November, Wollongong.

Rehman H, Thomson R & Watterson E (2007). *Hydrology in Urban Areas – 'Rainfall on the Grid' Versus Traditional Methodologies*, 47th NSW Floodplain Management Authorities Conference, Gunnedah.

Swan R & Thomson R (2011). *Direct Rainfall – Verifying the Technique Across Two States*, 34th IAHR World Congress, 26 June – 1 July, Brisbane, Australia.

Syme W J (2008). *Flooding in Urban Areas – 2D Modelling Approaches for Buildings & Fences*, 9th National Conference on Hydraulics in Water Engineering, Engineers Australia, Darwin, 23-26 September.

12. CHAPTER 12 – RECOMMENDATIONS

The following issues were considered by the authors to be areas of 2D hydraulic modelling practice in which further research is required. This list is by no means exhaustive and should not restrict future research in the field of 2D hydraulic modelling.

12.1. Direct Rainfall

The use of direct rainfall on a hydraulic model as a substitute for a hydrologic model is a relatively new concept, which is discussed in detail in Chapter 11. The use of direct rainfall was initially used on projects as an addition to the hydrologic model to allow the rainfall within the study area (which was historically only a small part of the catchment) to be included. Direct rainfall was also used for areas where the upstream inflow was predominately sheet flow. This practice has now evolved to the modelling of complete catchments (without a hydrologic model). Despite the fact that the direct rainfall approach is currently used in practice, the approach must still be regarded as being "under development". This document attempts to highlight some of the potential issues and uncertainties associated with its use (refer to Chapter 11). It is important to note that only limited testing of this approach has been undertaken, and very little of that has been on gauged catchments. The aim of this section of the report is to provide a summary of issues that have been identified during testing to assist in focussing further research. Issues include:

- Suitability of the rainfall excess model,
- Artificial depression storage leading to shallow depths and reduction in the event, volume
- Suitability of conventional roughness equations at very shallow depths,
- Delayed response time compared to hydrologic model,
- Calibration with gauged catchments, and
- Filtering of results.

12.2. Momentum Transfer

The representation of momentum transfer between the main channel and floodplain differs between commercially available models. Some models include the transfer of momentum between the channel and floodplain, and some do not. The main issue identified as requiring research is to identify when models that don't account for momentum transfer should not be used. This is likely to affect floodplain velocity, hazard and where development controls would be placed. This research needs to be able to provide advice for compound and meandering channels.

12.3. Test Catchments

A recommendation of both this ARR revision project and other ARR revision projects is the need to establish a test catchment which can be used to test new models and methods in both hydraulics and hydrology. The test catchment must have a detailed test data set. At the time of writing, no catchment in Australia meets the intensive data requirements set out below for a suitable test data set:

- pluviograph records,
- topographic data (channel cross sections, ALS data etc)
- gauged catchment,
- calibration data, and
- information regarding roughness values.

Test catchments will provide a vehicle for testing a range of modelling approaches such as representation of buildings in the floodplain, direct rainfall methods, and assessment of the threshold conditions under which urban pipe/street drainage components become important. For 2D models this will allow the benchmarking between models and their parameters. It will allow the testing of new models and schematisation. Test catchments will be part of the final phase of the ARR revision projects.

APPENDIX A

APPENDIX A: 2D HYRAUDLICS MODELS CURRENTLY USED IN AUSTRALIA

This section provides a brief history of hydrodynamic modelling in Australia.

1980s River modelling characterised by the use of steady state 1D models such as HEC-2 (now called HECRAS) and quasi-2D CELLS model

1990s River modelling characterised by the use of MIKE11 link and branch model with alternatives including RUBICON, and ESTRY

2000s 2D modelling becomes mainstream with 1D River and drainage models being linked

RMA and Mike21 have been used in various forms on specialist problems since the 1980s.

In the mid to late 1990s RMA and Mike 21 started to be applied not only for estuarine problems but also to flood problems. By 2005 most 2D modelling in Australia is carried out using fixed grid models (Mike21, SOBEK and TUFLOW) with RMA more commonly used for flooding of estuaries. Flexible grid finite volume models are currently available under the TUFLOW and MIKE brands as well as ANUGA. These models have mainly been used in estuarine and ocean situations as they have no or limited capacity to include structures and as a general rule are more computationally intensive.

The table below provides a snap shot of where 2D modelling is currently at in Australia. These features are likely to change and the user should contact the software supplier or consult the software manuals to determine what features are available in the version they are using.

Name	Supplier	Grid/ Unstructured	Parent/Child grids	2D weirs and structures	1D rivers	1D engine	1D structures	1D drainage	1D drainage engine	pit loss	Direct Rainfall	Evaporation
MIKE FLOOD	DHI	Grid Triangular & Quadrangular	Yes (fixed 3 -1 ratio in Fixed Grid)	Yes	Yes	MIKE11	Yes	Yes	MIKE URBAN	Yes	Yes	Yes
TUFLOW	BMT-WBM	Grid	Yes (with variable domain and rotation)	Yes	Yes	Estry (other options available)	Yes	Yes	?? (XP-SWMM option available)	Yes	Yes	Yes
SOBEK	Deltares	Grid (unstructured option also available)	Yes (variable ratio)	No	Yes	Yes	Yes	Yes	Yes	No	Yes	Yes
RMA2	Resource Management Associates	Triangular/rectangular elements	N/A as the FE mesh can have different resolution in different regions.	?	Yes	RMA2 – which can undertake 1D and 2D solutions within the same finite element mesh	Yes	No		No	Yes	Yes
ANUGA	Public domain	Triangular elements	N/A	No	No	No	Under development	limited		No	Yes	?
TUFLOW FINITE VOLUME	BMT-WBM	Triangular elements	N/A	No	No	No	Under development	No	No	No	Yes	?