Effect of the sub-grid geometry on two-dimensional river flow models

Treball realitzat per:
Ricard González Blanch

Dirigit per:
Ernest Bladé Castellet

Grau en:
Enginyeria Civil

Delft, 16 de juny del 2017

Departament d’Enginyeria Civil i Ambiental
Acknowledgements

Firstly, I would like to show my gratitude to the supervisor of this research project, Ernest Bladé Castellet, for his unconditional help throughout all the stages of this project.

Secondly, I would like to thank my family and friends for their constant support which has kept me motivated during my year as an exchange student.
Abstract

Although one-dimensional flow models have proved to perform adequately when flow is restricted between channel banks, two-dimensional flow modelling has shown to have the upper hand when assessing flows in topographically complex floodplains, where the flow is mostly attributed to be two-dimensional. The high accessibility of topographic data due to recent progresses in remote sensing techniques coupled with great advancements made in computing power have spurred the popularity of two-dimensional hydraulic models. However, the high computation times still remain a drawback suffered by the majority of current two-dimensional flow models.

This study aims to assess the effect of a newly developed modelling algorithm called high resolution sub-grid model on the recently released two-dimensional flow model from HEC-RAS 5.0. This algorithm intends to reduce the aforementioned high computation times of two-dimensional flow models by using larger mesh cell sizes while maintaining accurate results. The latter is achieved by integrating the details of the fine bathymetry from the underlying terrain model into the modelling process.

Furthermore, this work thoroughly analyses the state of the art of two-dimensional hydraulic modelling, heavily focusing on the capabilities and hydrodynamics of the two-dimensional HEC-RAS 5.0 module as well as on the implementation of the sub-grid model in the software. Subsequently, a hypothetical fluvial flooding event caused by a 500 year return period flow hydrograph in the lower part of the Ter river catchment is studied. Simulations are carried out for multiple computational mesh cell sizes with a computation interval that has been appropriately calibrated for each model beforehand.

Finally, the results of the simulations for four relevant variables related to fluvial flooding events are considered. The sub-grid model seems to yield effective flooding extent, maximum water depth and flow velocity results for a mesh cell size of 25x25 m, whereas the 50x50 m model is only recommended for the maximum water depth analysis. These results are obtained in much lower computation times than the 10x10 m model. Specifically, the 25x25 and 50x50 m models do it 29 hours and 31 hours faster than the 10x10 m model respectively. As for the arrival time, the sub-grid model does not offer good predictions, at least for the mesh sizes studied.

Keywords: 2D flow modelling, HEC-RAS, fluvial flood, sub-grid, mesh, computation time
Resumen

A pesar de que los modelos unidimensionales de flujo hayan demostrado ser capaces de rendir adecuadamente cuando el flujo se encuentra encauzado, los modelos bidimensionales de flujo han demostrado ser superiores cuando se trataba de flujos en planicies de inundación con una topografía compleja, donde el flujo tiende a ser mayoritariamente bidimensional. La popularidad de los modelos bidimensionales hidráulicos viene a raíz de la elevada accesibilidad a información topográfica debido al progreso hecho en el campo de la detección remota por satélite junto a los avances tecnológicos realizados en el campo de la computación. De todas formas, los altos tiempos computacionales continúan siendo una de las desventajas comunes entre los modelos bidimensionales de flujo.

Este trabajo tiene como objetivo principal evaluar el efecto de un algoritmo de modelización recientemente desarrollado llamado geometría sub-malla en el nuevo modelo de flujo bidimensional que incluye HEC-RAS 5.0. El algoritmo aspira reducir los altos tiempos computacionales característicos de los modelos bidimensionales de flujo utilizando tamaños de malla más grandes pero conservando resultados rigurosos. Lo último se consigue mediante la integración de los detalles de la batimetría provenientes del modelo del terreno en el proceso de modelización.

Además, el trabajo analiza con gran profundidad el estado del arte de la modelización hidráulica, centrándose en su gran parte en las capacidades y en el esquema hidrodinámico del módulo bidimensional de HEC-RAS 5.0, así como en la implementación explícita de la geometría sub-malla en el software. Posteriormente, se estudia una hipotética inundación fluvial causada por un hidrograma correspondiente a un período de retorno de 500 años en el tramo bajo del río Ter. Se llevan a cabo varias simulaciones para diversos tamaños de malla con el correspondiente tiempo de paso que ha sido debidamente calibrado para cada modelo anteriormente.

Finalmente se consideran los resultados de las simulaciones para cuatro variables relacionadas con inundaciones fluviales. La geometría sub-malla parece dar buenos resultados para la mancha de inundación, calado máximo y velocidad de flujo máxima para un tamaño de malla de 25x25 m, mientras que el uso del modelo de 50x50 m se recomienda exclusivamente para el análisis de calados máximos. Concretamente, los tiempos computacionales de los modelos de 25x25 m y de 50x50 m tardan 29 y 31 horas menos que el modelo de 10x10 m respectivamente. Para el tamaño de mallas estudiado, la geometría sub-malla no logra predecir con precisión el tiempo de llegada del agua.

Palabras clave: modelos bidimensionales de flujo, HEC-RAS, inundación fluvial, geometría sub-malla, malla, tiempo computacional
# Table of contents

Acknowledgements ........................................................................................................... i

Abstract .............................................................................................................................. ii

Resumen ................................................................................................................................ iii

Table of contents ................................................................................................................ iv

List of tables ....................................................................................................................... vii

List of figures ...................................................................................................................... viii

1. Introduction ...................................................................................................................... 1
   1.1 Objectives ................................................................................................................... 2
   1.2 Limitations .................................................................................................................. 3

2. Hydraulic modelling ........................................................................................................ 4
   2.1 Types of models ......................................................................................................... 4
   2.2 Software for computational models ......................................................................... 5
   2.3 Two-dimensional models ......................................................................................... 7

3. HEC-RAS 5.0 .................................................................................................................. 9
   3.1 Overview of features ................................................................................................ 9
   3.2 Two-dimensional flow capabilities ....................................................................... 9
   3.3 Two-dimensional modelling limitations .................................................................. 12
   3.4 Computational mesh properties ......................................................................... 13
   3.5 Sub-grid model ....................................................................................................... 14
   3.6 Two-dimensional unsteady flow hydrodynamics .................................................... 22
      3.6.1 Assumptions ....................................................................................................... 22
      3.6.2 Continuity equation ......................................................................................... 22
      3.6.3 Momentum equation ....................................................................................... 23
      3.6.4 Diffusion-wave approximation ....................................................................... 23
      3.6.5 Sub-grid model numerical algorithm ............................................................... 24
      3.6.6 Boundary conditions ....................................................................................... 26
   3.7 Numerical methods ................................................................................................. 27
   3.8 Computational mesh cell size and computation interval ....................................... 27

4. Pre-processing required information for simulation of the lower Ter ....................... 30
4.1 Study area .....................................................................................................................30
4.2 Terrain model ..............................................................................................................32
4.3 Boundary conditions .................................................................................................33
4.4 Hydrograph ..................................................................................................................33
4.5 Land cover classification and roughness map .............................................................35
5. Two-dimensional model development ........................................................................39
  5.1 Importing terrain model and computational mesh generation ....................................39
  5.2 Boundary conditions .................................................................................................43
  5.3 Land cover layer ........................................................................................................44
  5.4 Two-dimensional unsteady flow simulation parameters ............................................45
6. Results ..........................................................................................................................47
  6.1 Computation interval calibration ..............................................................................48
    6.1.1 Initial computation interval assumption for all mesh cell sizes .......................48
    6.1.2 Mesh cell size of 100x100 m .................................................................................51
      6.1.2.1 Appropriate computation interval for a 100x100 m mesh cell size ..........51
      6.1.2.2 Stability of the 100x100 m mesh cell size model ........................................61
    6.1.3 Mesh cell size of 50x50 m .....................................................................................63
      6.1.3.1 Appropriate computation interval for a 50x50 m mesh cell size ..........63
      6.1.3.2 Stability of the 50x50 m mesh cell size model ............................................66
    6.1.4 Mesh cell size of 25x25 m .....................................................................................67
      6.1.4.1 Appropriate computation interval for a 25x25 m mesh cell size ..........67
      6.1.4.2 Stability of the 25x25 m mesh cell size model ............................................70
    6.1.5 Mesh cell size of 10x10 m .....................................................................................70
      6.1.5.1 Appropriate computation interval for a 10x10 m mesh cell size ..........70
      6.1.5.2 Stability of the 10x10 m mesh cell size model ............................................74
  6.2 Maximum water depth, maximum flow velocity and arrival time map results ...........75
  6.3 Sub-grid model comparison analysis .......................................................................79
    6.3.1 Flooding extent ....................................................................................................79
    6.3.2 Maximum water depth .......................................................................................83
      6.3.2.1 Maximum water depth along profile lines ..................................................83
      6.3.2.2 Maximum water depth difference map .......................................................86
Effect of the sub-grid geometry on two-dimensional river flow models

Ricard González Blanch

6.3.3 Maximum flow velocity ........................................................................... 90
6.3.4 Arrival time ............................................................................................... 91

7. Conclusions ..................................................................................................... 96

7.1 HEC-RAS 5.0 2D module .............................................................................. 96
7.2 Development of the models ........................................................................... 96
7.3 Simulation of the models ............................................................................... 96
7.4 Sub-grid model results .................................................................................. 98

References .......................................................................................................... 100
Appendix............................................................................................................... 103
List of tables

Table 1. Evolution of HEC-RAS features for different versions ........................................ 9
Table 2. Hydrograph values for a 500 years return period flood in the lower Ter basins ..........................................................35
Table 3. Computation times for different mesh cell sizes with a 30 minute computation interval ........................................................................48
Table 4. Computation times for initial computation interval simulation for a 100x100 m mesh cell size ........................................................................................................51
Table 5. Computation times for initial computation interval simulations for a 50x50 m mesh cell size ........................................................................................................................................63
Table 6. Computation times for extra computation interval simulations for a 50x50 m mesh cell size ........................................................................................................................................65
Table 7. Computation times for initial computation interval simulations for a 25x25 m mesh cell size ........................................................................................................................................67
Table 8. Computation times for extra computation interval simulations for a 25x25 m mesh cell size ........................................................................................................................................67
Table 9. Computation times for initial computation interval simulation for a 10x10 m mesh cell size ........................................................................................................................................71
Table 10. Computation times for extra computation interval simulations for a 10x10 m mesh cell size ........................................................................................................................................72
Table 11. Summary of simulations to compare .................................................................................. 79
Table 12. Numerical analysis of the flooding extent results .................................................................................. 82
Table 13. Numerical analysis of the time arrival results .................................................................................. 95
List of figures

Figure 1. Computational mesh terminology (Source: HEC-RAS 2D Modelling User’s Manual) ................................................................. 13
Figure 2. Comparison of terrain resolution and computational cell size .................. 15
Figure 3. Closer look-up of a cell c5856 and its faces .................................... 16
Figure 4. Hydraulic property table: Cell 5856: Volume – Elevation relationship ...... 16
Figure 5. Hydraulic property table: Face 12793: Area - Elevation relationship ....... 17
Figure 6. Hydraulic property table: Face 12793: Profile .................................. 17
Figure 7. Hydraulic property table: Face 12793: Manning’s n - Elevation relationship 18
Figure 8. Hydraulic property table: Face 12793: Wetted Perimeter - Elevation relationship .................................................. 18
Figure 9. Wetting based on the terrain’s grid-cells ............................................ 20
Figure 10. Example of a canal thinner than computational cell size (Source: HEC-RAS 2D Modelling User’s Manual) ................................. 20
Figure 11. Leakage due to suboptimal cell alignment (Source: The Ras Solution) .... 21
Figure 12. Basin from the Ter (Source: PEF from the lower Ter basins) ................ 31
Figure 13. Satellite view of the study area ....................................................... 32
Figure 14. Selected sheets for the DEM of the study area .................................. 32
Figure 15. Hydrograph for a 500 year return period flood in the lower Ter river basins ............................................................................ 35
Figure 16. Land cover classification map of study area (MCSC-4) ......................... 37
Figure 17. Land cover classification map of study area (MCSC-3) ......................... 37
Figure 18. Attribute table for MCSC-3 ............................................................ 38
Figure 19. Study area divided in roughness intervals ....................................... 38
Figure 20. Visualization of the terrain model in RAS-Mapper ............................... 39
Figure 21. Computational mesh contour on satellite view ................................. 40
Figure 22. Generating points for 50x50 m cell sizes ....................................... 40
Figure 23. 2D computational mesh (left) and detail of cell faces aligning with breaklines (right) ................................................................. 41
Figure 24. Mesh generation problem: Two cell centres in one cell (Source: Combined 1D/2D Modelling with HEC-RAS) .......................... 41
Figure 25. Mesh generation problem: Cell with no cell centre (Source: Combined 1D/2D Modelling with HEC-RAS) ................................. 42
Figure 26. Mesh generation problem: Cell Face Crosses into Multiple cells (Source: Combined 1D/2D Modelling with HEC-RAS) ............... 42
Figure 27. Mesh generation problem: Cell has more than One Outer Boundary Face (Source: Combined 1D/2D Modelling with HEC-RAS) ................. 43
Figure 28. Boundary condition on the mesh. Downstream BC (left) and Upstream BC (right).  
Figure 29. Setting flow hydrograph and normal depth data for the boundary conditions.  
Figure 30. Recognizing Manning’s n values in HEC-RAS.  
Figure 31. Land cover layer in HEC-RAS for 2D Flow Areas.  
Figure 32. Unsteady Flow Analysis tab.  
Figure 33. Flood wave propagations for a computation interval of 30 min that show instabilities for all mesh cell size models.  
Figure 34. Errors in the computation messages window for a 10x10m mesh cell size and a 30 min computation interval (only shows part of the long list).  
Figure 35. Time series plot of a point close to the upstream boundary condition for different mesh cell sizes and a 30 min computation interval.  
Figure 36. Maximum water depth results for a 100x100 m mesh cell size and a 10 min computation interval.  
Figure 37. Maximum water depth results for a 100x100 m mesh cell size and a 5 min computation interval.  
Figure 38. Maximum water depth results for a 100x100 m mesh cell size and a 3 min computation interval.  
Figure 39. Maximum water depth results for a 100x100 m mesh cell size and a 1 min computation interval.  
Figure 40. Maximum water depth results for a 100x100 m mesh cell size and a 5 s computation interval.  
Figure 41. Maximum water depth results for a 100x100 m mesh cell size and a 1 s computation interval.  
Figure 42. Profile lines (in pink) drawn for plotting the simulations.  
Figure 43. Maximum water depth plot along the upstream profile for a 100x100 m mesh cell size and for computation intervals: 10 min, 5 min, 3 min, 1 min, 5 s and 1 s.  
Figure 44. Maximum water depth plot along the upstream profile line for a 100x100 m mesh cell size and for computation intervals: 5 min, 3 min, 1 min, 5 s and 1 s.  
Figure 45. Maximum water depth plot along the middle profile line for a 100x100 m mesh cell size and for computation intervals: 5 min, 3 min, 1 min, 5 s and 1 s.  
Figure 46. Maximum water depth plot along the downstream profile line for a 100x100 m mesh cell size and for computation intervals: 1 min, 5 s and 1 s.  
Figure 47. Maximum water depth plot along the downstream profile line for a 100x100 m mesh cell size and for computation intervals: 2 min, 1 min, 5 s and 1 s.  
Figure 48. Closer look-up of the four final simulations for a 100x100 m mesh cell size.
Figure 49. Maximum depth results for a mesh cell size of 100x100 m and a 1 min computation interval with 2D computational mesh with the terrain model as background. ................................. 61

Figure 50. Upper bound Courant number map for a 1 min computation interval on a 100x100 m mesh cell size ................................................................. 62

Figure 51. Maximum water depth results for a 50x50 m mesh cell size and a 10 min computation interval ............................................................................. 64

Figure 52. Maximum water depth results for a 50x50 m mesh cell size and a 5 min computation interval ............................................................................. 64

Figure 53. Maximum water depth plot along the downstream profile line for a 50x50 m mesh cell size and for computation intervals of: 12 s, 10 s, 5 s and 1 s ........................................... 65

Figure 54. Closer look-up of the four final simulations for a 50x50 m mesh cell size ... 66

Figure 55. Upper bound Courant number map for a 12 s computation interval on a 50x50 m mesh cell size ............................................................................. 66

Figure 56. Unstable wave propagation for a 25x25 m mesh cell size and computation interval of 3 min ............................................................................. 68

Figure 57. Maximum water depth plot along the downstream profile line for a 25x25 m mesh cell size and for computation intervals of: 10 min, 5 min, 3 min, 1 min, 30 s, 15 s, 10 s, 6 s, 5 s, 1 s ............................................................................. 68

Figure 58. Maximum water depth plot along the downstream profile line for a 25x25 m mesh cell size and for computation intervals of: 10 s, 6 s, 5 s and 1 s ......................... 69

Figure 59. Closer look-up of the four final simulations for a 25x25 m mesh cell size .... 69

Figure 60. Upper bound Courant number map for a 10 s computation interval on a 25x25 m mesh cell size ............................................................................. 70

Figure 61. Maximum water depth results for a 10x10 m mesh cell size and a 3 min computation interval ............................................................................. 71

Figure 62. Maximum water depth results for a 10x10 m mesh cell size and a 1 min computation interval ............................................................................. 71

Figure 63. Maximum water depth plot along the downstream profile line for a 10x10 m mesh cell size and for computation intervals of: 10 min, 5 min, 3 min, 1 min, 30 s, 10 s, 5 s and 3s ............................................................................. 72

Figure 64. Maximum water depth plot along the downstream profile line for a 10x10 m mesh cell size and for computation intervals of: 30 s, 10 s, 5 s and 3s ......................... 73

Figure 65. Closer look-up of the four final simulations for a 10x10 m mesh cell size .... 74

Figure 66. Upper bound Courant number map for a 5 s computation interval on a 10x10 m mesh cell size ............................................................................. 74

Figure 67. Maximum water depth map for a 100x100 mesh cell size and a 60 s computation interval model ............................................................................. 75

Figure 68. Maximum water depth map for a 50x50 mesh cell size and a 12 s computation interval model ............................................................................. 75
Figure 69. Maximum water depth map for a 25x25 mesh cell size and a 10 s computation interval model .......................................................... 76
Figure 70. Maximum water depth map for a 10x10 m mesh cell size and a 5 s computation interval model .......................................................... 76
Figure 71. Maximum flow velocity map for a 100x100 m mesh cell size and a 60 s computation interval model .......................................................... 76
Figure 72. Maximum flow velocity map for a 50x50 m mesh cell size and a 12 s computation interval model .......................................................... 77
Figure 73. Maximum flow velocity map for a 25x25 m mesh cell size and a 10 s computation interval model .......................................................... 77
Figure 74. Maximum flow velocity map for a 10x10 m mesh cell size and a 5 s computation interval model .......................................................... 77
Figure 75. Arrival time map for a 100x100 m mesh cell size and a 60 s computation interval model ............................................................. 78
Figure 76. Arrival time map for a 50x50 m mesh cell size and a 12 s computation interval model ............................................................. 78
Figure 77. Arrival time map for a 25x25 m mesh cell size and a 10 s computation interval model ............................................................. 78
Figure 78. Arrival time map for a 10x10 m mesh cell size and a 5 s computation interval model ............................................................. 78
Figure 79. Flooding extent difference between the 10x10 m and the 100x100 m models .................................................................................................................................. 80
Figure 80. Flooding extent difference between the 10x10 m and the 50x50 m models .................................................................................................................................. 81
Figure 81. Flooding extent difference between the 10x10 m and the 25x25 m models .................................................................................................................................. 81
Figure 82. Summary of flooding extent difference cell count for each model comparison .................................................................................................................................. 83
Figure 83. Sub-grid comparison for the upstream profile line .................................................................................................................................. 84
Figure 84. Sub-grid comparison for the middle profile line .................................................................................................................................. 84
Figure 85. Sub-grid comparison for the downstream profile line .................................................................................................................................. 85
Figure 86. Absolute difference in maximum water depth between the 10x10 and 100x100 m models .................................................................................................................................. 86
Figure 87. Absolute difference in maximum water depth between the 10x10 and 50x50 m models .................................................................................................................................. 87
Figure 88. Absolute difference in maximum water depth between the 10x10 and 25x25 m models .................................................................................................................................. 87
Figure 89. Classification statistics of the water depth difference between the 10x10 m and the 25x25 m, 50x50 m and 100x100 m models respectively (from left to right) .................................................................................................................................. 88
Figure 90. Upstream profile line (in pink) intersects the controversial flooded area (surrounded by a yellow rectangle) .................................................................................................................................. 89
Figure 91. Absolute difference in maximum flow velocity between the 10x10 and 100x100 m models ................................................................. 90
Figure 92. Absolute difference in maximum flow velocity between the 10x10 and 50x50 m models ................................................................. 90
Figure 93. Absolute difference in maximum flow velocity between the 10x10 and 25x25 m models ................................................................. 90
Figure 94. Flooding extent difference between the 10x10 m and the 100x100 m models ................................................................. 92
Figure 95. Flooding extent difference between the 10x10 m and the 50x50 m models ................................................................. 92
Figure 96. Flooding extent difference between the 10x10 m and the 25x25 m models ................................................................. 93
Figure 97. Cell count for time arrival difference categories for the 100x100 m mesh cell size model ................................................................. 94
Figure 98. Cell count for time arrival difference categories for the 50x50 m mesh cell size model ................................................................. 94
Figure 99. Cell count for time arrival difference categories for the 25x25 m mesh cell size model ................................................................. 94
1. Introduction

Ever since early times in history, the human civilization has gathered around waterbodies such as seas, lakes, rivers, streams and canals due to water being the single most vital element for the human species survival. Nowadays, floods, alongside earthquakes, represent the deadliest natural hazard in the world which results not only in significant life losses but also in huge economic repercussions in terms of damages to buildings, structures and business. Actually, coastal areas are expected to be further threatened by floods due to the alarming rate of increase of mean global sea water level. The level is currently forecasted to rise up to 0.82 m by the end of the twenty-first century and is the main sequel to climate change[1]. For those reasons, understanding the way water flow reacts in all kinds of environments has been of the utmost importance up to this day.

Flooding events are generally divided into three categories depending on their characteristics: fluvial floods, pluvial floods and coastal floods. Fluvial floods occur when water in rivers overtop their banks as a result of sustained or intense rainfall, thus flooding the nearby floodplains. They are governed by hydrological processes, namely precipitation, evaporation and infiltration, developing over large temporal and spatial scales. Contrarily, pluvial floods take place when excess water from rainfall cannot be absorbed due to both the intensity of the rain exceeding the infiltration capacity of the ground and the heavy downpour of rain saturating drainage systems. These occur on a noticeably smaller temporal and spatial scale than fluvial floods, making them less predictable. Lastly, coastal flooding is caused by rises in sea water level due to climate change, wind set up and extreme tidal conditions including high tides, storm surges and tsunamis.

The complex behaviour and detrimental consequences shown by the aforementioned natural hydraulic phenomena have led to considerable investments in the development of hydraulic models capable of predicting water flow in a large variety of situations. Moreover, the performance of hydraulic models has drastically improved over the last decades mainly because of the continuous progress made in establishing more powerful computers and more accurate remote sensing techniques which allow for higher quality input information such as rainfall data, topography and land cover classification. In spite of the efforts devoted to reduce the computation time of the hydraulic models, the latter still remains as one of the leading open research fields. In fact, one of the essential factors within hydraulic model optimization is dealing with elevated computation times since they tend to lead to increased monetary costs. This issue is primarily addressed by making trade-offs between model performance or detail of the results and model run time.
Water flow in rivers and channels has been traditionally modelled by means of one-dimensional (1D) models because flow is assumed to be perpendicular to the cross-sections of the rivers or channels to a large extent. For fluvial flooding this one-dimensional (1D) flow assumption does not hold any longer for most cases as water bursts over the river banks and spreads over the nearby floodplains in different directions. As a result, multitudes of two-dimensional (2D) models and quasi-two-dimensional (1D/2D) models have been released recently despite requiring more detailed river bathymetry to obtain satisfactory results than one-dimensional (1D) models, thus producing even higher run times. This fact aggravates the already existing modelling dilemma between lower computation times or more elaborated results. High resolution sub-grid models have been introduced very recently in hydraulic modelling software such as HEC-RAS 5.0 with hopes to reduce the high computation times from two-dimensional (2D) models while maintaining good model accuracy. In order to do that, the sub-grid model makes use of high resolution topographic data and larger computational mesh cell sizes.

Due to its novelty, the performance and implications of the two-dimensional (2D) sub-grid models are still being investigated. This study aims to evaluate the general capabilities of the two-dimensional (2D) sub-grid model in HEC-RAS 5.0 by considering a hypothetical fluvial flooding event caused by a 500 year return period flow hydrograph in the lower part of the Ter river catchment.

1.1 Objectives

Although the main ambition of this study concerns the effect of sub-grid models in two-dimensional flow models, other goals and pre-requisite objectives have been set with the purpose of splitting up the initial proposed task.

Specifically, the following objectives will be undertaken:

- Familiarizing with the new 2D features in HEC-RAS 5.0 and further exploration of its capabilities. Specifically, understanding the requirements for a 2D model set-up.
- Recognize the importance and implications of the sub-grid model in 2D flow modelling.
- Study of the numerical scheme followed by HEC-RAS 5.0 in 2D computations. Thorough analysis on how the sub-grid representation is implemented in such scheme.
- Choice of the study area and collection of the required input data, namely the terrain model and the flow hydrograph.
- Development and simulation of multiple models that allow future interpretation of the influence from the sub-grid representation.
• Selection of the variables to be compared by the different models.
• Analysis of the results obtained between the developed models. Assessment of the sub-grid model performance in terms of accuracy and computation time.
• Recommendations as to what models best predict the behaviour of the variables chosen to be examined.

1.2 Limitations
First and foremost, the simulations will be exclusively carried out by using the latest version of HEC-RAS. This study does not look to compare hydraulic modelling software between them, but rather focus on the robustness provided by HEC-RAS 5.0 as a whole and especially its newly featured 2D hydrodynamic module.

Finally, despite using real life data for the flow hydrographs and terrain model, the study strictly involves comparing different computational models between them. Contrasting the results obtained in the models with real life validation data is out of the scope of this project. It certainly would be enriching to compare the results obtained in HEC-RAS with a fluvial flood that took place back in time. In a way, the aim of this study revolves more about calibrating lower computation time consuming models that provide sufficiently accurate results in order to prevent future flooding disasters.
2. Hydraulic modelling

2.1 Types of models

The solutions of a problem related to hydraulics or a hydraulic engineering design can be approached in three different ways: theory and reasoning, experience (derived from similar structures) and by investigating the problem and testing the design on models. Solutions which stem from a purely theoretical approach and basic flow equations must be often disregarded as they appear to be accurate only in schematized and simplified situations. Although experience may be used as a way to learn from past events, it is certainly not an adequate approach due to the singularity of the boundary conditions and circumstances. Finally, models which incorporate properly derived theoretical analysis are usually the optimal solution, whereas exclusively experimental models are often carried out in vain due to a clear lack of theoretical guidelines[2].

Depending on the type of simulation, models can be classified into three different categories: direct simulation by use of hydraulic models, semi-direct simulation by making use of analogues or indirect simulation by applying mathematical, numerical and computational models. For simplicity, the main distinction to be made between the models is whether they follow a physical or a mathematical approach. On the one hand, direct and semi-direct simulation models are considered to be scale models, which are a product of physical simulation of a prototype. On the other hand, indirect simulation models follow a mathematical based approach.

The principle of the use of scale models consists of the possibility to reproduce the real problem (‘the prototype’) on a smaller scale in such a way that the phenomena in the scale model are similar in both model and prototype. The similitude between model and prototype must encompass all geometric, kinematic and dynamic similarities[3]. In order to carry out the model representation, a scale factor should be found for each parameter in such a way that it is the ratio between the prototype value and the model value of that parameter. These scale factors are calculated under the hypothesis that certain dimensionless parameter is constant for both the model and the prototype. The most common similitude criteria used in the field of hydraulics are the Froude, Reynolds, Weber, Mach and Euler similitude criterion which are based on the respective dimensionless parameters.

The mathematical models use a set of equations and expressions which try to reproduce the phenomenon that is being studied. The main asset of the mathematical model is the implementation of all physical processes which take place in the phenomenon. The disadvantage this model entails is that it is only representative of the simplest geometries. Within the mathematical models, distinction between deterministic and
stochastic models must be pointed out. In the deterministic models, all physical processes involved are expressed by means of deterministic functional relations which do not take into account the probability of occurrence of the phenomenon. Contrarily, stochastic models consider physical processes as random variables which involve the phenomenon in study[4].

Numerical models are an approximation of the above-mentioned mathematical models where flow parameters are described in a certain amount of discrete points. As with the mathematical models, the numerical models are not specific to a particular site so they do not take into account intrinsic characteristics of the surrounding geometry. Numerical models use initial boundary conditions and largely depend on the numerical method employed to solve the set of equations. The most renowned techniques for solving the equations are the finite differences method and the finite element method (FEM).

Lastly, computational models comprise what mathematical and numerical models offer by implementing a numerical model on a computer system while using specific data from a particular site. Computational models are currently the most used tool to model water flow in the hydraulics research field. This is primarily attributed to the fact that it is easier to change the geometry in computational models than in physical models. Moreover, computational models have turned out to be more cost-effective for the majority of the cases, although physical hydraulic modelling is still relevant since it is required in cases where unsteady vortex dynamics is a concern, such as in pump or turbine intakes[5].

2.2 Software for computational models
There is a large variety of software available for computational hydraulic modelling, and it is very dependent on the type of hydraulic analysis performed. It is safe to say that at least, there exists a program for each possible scenario related with water. Some of the most common situations in which hydraulic software is needed are dam failures, flooding, sewer and culvert calculations, water distribution systems and sediment transport analysis. This project will exclusively focus on water flow modelling and flooding in rivers.

The types of model differ in the way that the governing equations are defined, in the numerical solution of such equations and in the way that the geometric data is introduced. Therefore, flow modelling models are mainly distinguished by three aspects:

- Water flow dimension
- Equations for water flow and numerical method used to solve them
- Boundary conditions
There are up to four different ways to represent water flow in the case of flooding in a river. The water flow in such situations can be represented as a one-dimensional (1D) flow, a two-dimensional (2D) flow or a three-dimensional (3D) flow. Moreover, some of the existing software offer the possibility to combine a 1D flow model with a 2D flow model, commonly referred to in literature as quasi-2D models. The latter is mostly employed when the study area is focused on the floodplains of a river. It is possible to use such model in these cases because the water flow in a river can be assumed to be 1D for the most part, whereas the 2D flow is reserved for the study of the floodplain areas. The convenience of these 1D/2D models is the computation time saved by modelling the river as 1D instead of as 2D.

Furthermore, each program uses a different set of equations to represent water flow and numerical methods to solve them. However, water flow is mostly represented by the Saint-Venant equations, also known as the shallow water equations. With respect to the solving of the equations, there are more discrepancies between what method is more appropriate. The most common methods are: the finite differences method, the finite element method and the finite volume method.

Lastly, each program demands a certain type or minimum of boundary conditions to manage to perform the simulation. In general, the more options available to set as boundary conditions, the more complete the analysis will result. Boundary conditions may include but are not restricted to: flow hydrographs, stage hydrographs, rating curves, normal depth, specific water surface elevation and critical depth. In addition, some programs allow to establish other boundary conditions by adding hydraulic structures in the course of water flow.

In the following paragraphs a brief outline for commonly used programs in water flow modelling are presented:

- **HEC-RAS**: This is the program that is going to be used throughout the project. For detailed information check section 3.1 *Overview of features*.

- **MIKE-11**: It was developed by the software department of the Danish Hydraulic Institute (DHI) for 1D modelling in free-surface flow and unsteady flow conditions. It contains many modules which are capable of dealing with flooding, navigation, water quality, forecasting, sediment transport or a combination of these or other aspects of river modelling. The model solves the Saint-Venant equations by means of the finite differences method and an implicit scheme. The hydrodynamic module calculates water depths and velocities over time, under both subcritical and supercritical flow regimes and allows for quasi-2D calculations\[^{[6]}\].
- TUFLOW: It provides 1D and 2D solutions of the free-surface flow equations to simulate flood and tidal wave propagation. It is also one of the pioneers in 1D/2D linking for flood modelling. TUFLOW's 2D solution is based on the Stelling finite difference, alternating implicit (ADI) scheme that solves the full 2D free surface shallow water flow equations. Its applicability covers river flooding, urban flooding, pipe network modelling, storm tide and tsunami inundations and estuarine and coastal tidal hydraulics.[7]

- SOBEK: It has been developed jointly with Dutch public institutes and governmental organisation, research institutes, universities and private consultants all over the world. The modules represent phenomena and physical processes in an accurate way in 1D network systems and on 2D horizontal grids. The hydrodynamic 1D/2D simulation engine, which is able to work under a mixed flow regime, allows the combined simulation of pipe, river, channel and overland flow through an implicit coupling of 1D and 2D flow equations. It is a powerful modelling suite for flood forecasting, optimization of drainage systems, control of irrigation systems, sewer overflow design, river morphology, salt intrusion and surface water quality.[8]

2.3 Two-dimensional models
The development of river models has gone hand-in-hand with the accessibility to topographic data. The most remarkable trait of one-dimensional (1D) river models is the fact that they only require topographic data to be gathered at the cross sections because those are the only elements describing the topography of the river and floodplain. Simple geometry alongside limited access to topographic data and the adaptation of the shallow water equations (SWE) into a one-dimensional analysis turned 1D river models into the undisputed solution for modelling fluvial flooding events for a long time.

Although 1D models have proved to perform adequately when flow is restricted between channel banks, 2D modelling has shown to have the upper hand when assessing flows in topographically complex floodplains, where the flow is mostly attributed to be two-dimensional.[9]

As opposed to 1D models, 2D hydraulic models are constituted by a 2D computational mesh which is able to capture the underlying topography by connected cells. As a result, a topographic model capable of describing the entire area to be modelled in 2D is needed. Throughout recent years, this topographic data setback that used to hinder the development of 2D hydraulic models has been overcome on account of the development of Light Detection And Ranging (LIDAR) techniques for land surveying.
Consequently, the availability and progress made in recent years on terrain models has spurred the popularity of 2D hydraulic models.

A major drawback which is present in most 2D models is the large computation times depending on the size of the cells within the computational mesh. When the computations are performed on the same resolution as the terrain model acquired through LIDAR techniques, the models may go as far as leading to times which are considered exorbitant in terms of simulation time.

This issue has been addressed recently by Yu and Lane (2006)\cite{yu2006} and Casulli (2008)\cite{casulli2008} by means of developing models that can represent the topographic data on a sub-grid level, taking into account important features while maintaining the computational mesh cell size large, thus reducing computation times. The feedback from these studies was very positive, as they showcased better results than 1D/2D combined models and also permitted to analyse larger river reaches and floodplains.
3. HEC-RAS 5.0

3.1 Overview of features

HEC-RAS (Hydrological Engineering Center – River Analysis System) is a computer program developed by the U.S. Army Corps of Engineers’ Hydrologic Engineering Centre which forms part of the U.S Department of Defense. The purpose of its development is to facilitate the modelling of water flow hydraulics in natural rivers and other free-surface channels. Its popularity comes from the free distribution policy that surrounds it and its graphical user-friendly interface.

Up until recently, the river analysis components contained within HEC-RAS were\textsuperscript{[12]}:

- Steady Flow Water Surface Profiles
- One-Dimensional Unsteady Flow Simulation
- Sediment Transport
- Water Quality Analysis

Additionally, these four river analysis components can be further complemented by several hydraulic design features which can be invoked once the basic water surface profiles are generated.

Throughout the years, the capabilities of the program have been enhanced adequately through regular updates on the program versions. The version used in the entirety of this project is the latest version of the program, HEC-RAS 5.0. As such, this new version includes new features, amongst which the most outstanding one is the possibility of two-dimensional flow modelling in unsteady flow conditions, which is especially remarkable for flooding situations.

<table>
<thead>
<tr>
<th>HEC-RAS Version</th>
<th>1D Modelling</th>
<th>1D/2D Modelling</th>
<th>2D Modelling</th>
<th>Unsteady Flow</th>
<th>RAS Mapper</th>
<th>Sediment Transport</th>
<th>Water Quality Analysis</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>✓</td>
<td>×</td>
<td>×</td>
<td>×</td>
<td>×</td>
<td>×</td>
<td>✓</td>
</tr>
<tr>
<td>3</td>
<td>✓</td>
<td>×</td>
<td>×</td>
<td>✓</td>
<td>×</td>
<td>×</td>
<td>×</td>
</tr>
<tr>
<td>4</td>
<td>✓</td>
<td>✓</td>
<td>×</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>5</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
</tbody>
</table>

Table 1. Evolution of HEC-RAS features for different versions

3.2 Two-dimensional flow capabilities

The new HEC-RAS 5.0 version is most renowned for the possibility to model water flow in 2D. A series of advantages regarding this new feature is summarized below.
Effect of the sub-grid geometry on two-dimensional river flow models

Ricard González Blanch

Notice this information has been directly extracted from the HEC-RAS 5.0 2D Modelling User’s Manual[13].

- Can perform 1D, 2D and combined 1D and 2D modelling:
  HEC-RAS is able to perform flow simulations in 1D, 2D and combined 1D and 2D. The combined 1D/2D option within the same unsteady flow model allows the user to work on large river areas, modelling the main river channel as a 1D element and utilizing 2D modelling in areas which require more hydrodynamic detailing. The combined 1D/2D modelling is especially convenient when the area of study involves the surrounding floodplains of the river, and not so much the main course of the river. In this case, the 1D modelling of the river system allows for small computational costs in comparison with a full 2D model simulation.

- Saint-Venant or Diffusion Wave Equations in 2D:
  Users have the flexibility to select which equations should be solved for the 2D simulation. The program solves either the 2D Saint-Venant equations (with optional momentum additions for turbulence and Coriolis effects) or the 2D Diffusion Wave equations. Although the 2D Diffusion Wave equations allow the software to run faster and have greater stability properties, the 2D Saint-Venant equations are applicable to a wider range of problems. The 2D Diffusion Wave equations are set as default since they are able to accurately model any situation. However, users may experiment with one equation or another to analyse which one suits best for their modelling case.

- Implicit Finite Volume Solution Algorithm:
  The 2D unsteady flow equations solver uses an Implicit Finite Volume Algorithm. This implicit algorithm allows for larger computational time-steps than explicit methods, which translates into smaller computational costs. The Finite Volume Method offers improved stability and robustness over commonly used finite difference and finite element techniques in numerical models. The wetting and drying processes of 2D cells is extremely robust as 2D flow areas can start completely dry and handle a sudden rush of water into the area. Moreover, the algorithm is capable of working under different flow regimes, namely, subcritical, supercritical and mixed flow regimes.

- 1D and 2D Coupled Solution Algorithm:
  The 1D and 2D solution algorithms are coupled for each computational time-step, where it is possible to switch between 1D and 2D flow transfers within the same time-step. This coupled algorithm is the basis upon which the combined
1D/2D model is constructed. For instance, consider a river modelled in 1D that is connected by a lateral hydraulic structure (weir in this case) with the 2D area behind the levee. Flow over the levee is computed with the weir equation, using the headwater from the 1D river and a tailwater from the 2D flow area to which it is connected. In each time-step the weir equation uses the 1D and 2D results to compute the flow, allowing for accurate accounting of weir submergence at each time-step, as the 2D area fills up.

- **Unstructured or Structured Computational Meshes:**
  Even though HEC-RAS was originally designed to use unstructured computational meshes, it is capable of working with structured meshes too. In most cases, the computational mesh ends up being a mix of the two, known as hybrid mesh, containing both irregular cells, which refer to an unstructured mesh, and rectangular cells, which refer to a structured mesh. The software treats both meshes similarly, except it utilizes cells that are orthogonal to each other in order to simplify some of the required computations. Grid orthogonality is where the centres of two adjacent cells are perpendicular to the face between them\(^{[14]}\). In fact, the mesh is formed by computation cells which can be irregular or regular polygons of up to eight sides and it does not have to be formed by equal cells. Actually, the mesh is most usually computed as a mixture of different cell shapes and sizes.

  The outer boundary of the computational mesh is defined as an irregular polygon and the cells that form it can have very detailed multi-point lines that represent the outer face(s) of each cell. It is not necessary for the computational mesh to be orthogonal, but if the mesh happens to be orthogonal then the numerical discretization is simplified and more efficient.

- **Detailed Hydraulic Property Tables for 2D Computational Cells and Cell Faces:**
  HEC-RAS has a 2D flow area pre-processor that processes the cells and cell faces into detailed hydraulic property tables based on the underlying terrain used in the modelling process. In broad terms, the pre-process calculates a detailed relationship between Elevation-Volume for each cell, and for each cell face computes the following relationships: Elevation-Wetted Perimeter, Elevation-Area, Elevation-Roughness and more hydraulic parameters.

  These detailed hydraulic property tables created by HEC-RAS allow the users to create bigger computational cells while preserving the terrain details. This turns out to be extremely beneficial due to a decrease in the computation time.
As this is the main feature from the HEC-RAS 5.0 version to be studied in this project, a more in-depth analysis can be found in section 3.5 Sub-grid model.

- **Detailed Flood Mapping and Flood Animations:**
  Results of 2D simulations such as mapping of the inundated area, on top of animations of the flooding (dynamic mapping) can be viewed through a user-friendly tool within HEC-RAS, the RAS Mapper features. As in the case of the hydraulic property tables, the mapping of the 2D flow areas is based on the detailed underlying terrain. As a consequence, the wetted areas will have the same resolution as the one of the terrain provided, which will be different from the computational mesh cell size.

  The reason why this is possible is because of the way the computational algorithm operates: cells are computed as partially wet/dry since the computations take into account the resolution of the terrain by making use of the aforementioned pre-processed hydraulic table properties. Mapping of the results will reflect those details, rather than being limited to showing a computational cell as either all wet or all dry.

- **Multi-Processor Based Solution Algorithm (Parallel Computing):**
  The 2D flow area computational solution is programmed to take advantage of multiple processors on a computer (referred to as parallelization), allowing it to run much faster than on a single processor.

- **64-Bit and 32-Bit Computational Engines:**
  HEC-RAS now comes with both 64-bit and 32-bit computational engines. The software will use the 64-bit computational engines automatically if installed on a 64-bit operating system. The 64-bit computational engines run faster than the 32-bit and can handle much larger data sets.

### 3.3 Two-dimensional modelling limitations

In spite of the large variety of features and possibilities the new version brings with respect to 2D modelling, there are still a certain number of limitations which restrict HEC-RAS’ workability in 2D. The following items have been released in the HEC-RAS 5.0 2D Modelling User’s Manual[^13], and are currently under development in order to have them available for future versions.

- Offers limited flexibility for adding hydraulic structures inside a 2D flow area.
- Cannot currently perform sediment transport erosion/deposition in 2D flow areas.
- Cannot currently perform water quality modelling in 2D flow areas.
• Cannot connect pump stations to 2D flow area cells.
• Cannot use the HEC-RAS bridge modelling capabilities inside of a 2D flow area. Culverts, weirs, and breaching can be added by using the SA/2D Area Connection tool.

3.4 Computational mesh properties
To understand future references and explanations it is instrumental to define some of the mesh and cell properties since these terms are going to come up very often. Computational meshes are the elements that define the two-dimensional computation space, similar to how cross-sections construct the computation space for one-dimensional models.

The generation of the 2D computational mesh starts off by defining the 2D Flow Area that will cover the area of study and is where flow computations will take place. This stage of the mesh generation process is sometimes overlooked because it is simple but at the same time remains indispensable since it defines the boundary for which the computations will be carried out. Any terrain model outside of the boundaries of the computational mesh will not be taken into account for the simulation. The mesh boundary is a polygon with as many sides as needed that surrounds the previously defined 2D Flow Area. After the computational cells are formed, the mesh will look similar to the one of Figure 1.

Figure 1. Computational mesh terminology (Source: HEC-RAS 2D Modelling User’s Manual)
Cells can be defined by three elements\textsuperscript{[13]}:

- **Cell Centre**: The computational centre of the cell. This is where the water surface elevation is computed for the cell. The cell centre does not necessarily correspond to the exact cell centroid.
- **Cell Faces**: These are the cell boundary faces. Faces are generally straight lines, but they can also be multi-point lines, such as the outer boundary of the 2D flow area. As a reminder, the number of cell faces per cell is limited to eight.
- **Cell Face Points**: The cell Face Points (FP) are the ends of the cell faces. The Face Points (FP) numbers for the outer boundary of the 2D flow area are used to hook the 2D flow area to 1D elements and boundary conditions.

### 3.5 Sub-grid model

Most 2D flow modelling programs require the mesh cell sizes to be of the same size as the resolution of the terrain model used in order to obtain good results, leading to high computation times. Thus, the fundamental objective of the sub-grid model is to reduce these high computational costs, namely, decrease the run time of the simulations. That is accomplished by increasing the computational mesh cell size so that there are less cell sizes to compute. In most common 2D modelling software, where the sub-grid model is not present, by increasing the cell size it is guaranteed that the run times will be shorter; however this implies a reduction of the modelling accuracy since cells cover more terrain. Therefore, more terrain points will be neglected in the computation as all points in the cell are represented by the value of the cell centre. As a result, the user is often put in a situation where a decision between good model accuracy or low run times has to be made.

With the sub-grid model this does not occur to such extent because it allows mesh cells to have detailed bathymetry and properties based on the details of the underlying terrain model. This allows for larger cell sizes while preserving detailed information of the terrain, thus reproducing very similar results, in much lower computation times, to the ones obtained with small computational mesh cell sizes. The term “sub-grid” specifically makes reference to the fact that it uses the geometry of the underlying terrain. By implementing the sub-grid model, the cells and cell faces are neither restricted to having single elevation value nor being only described by a single point, as opposed to models without sub-grid representation (usually it is the centre for those models), thus preventing the cell from having a flat bottom. In other words, without the sub-grid representation the mesh cell sizes should be of the same size as the terrain model cells to obtain good results, while with the sub-grid model, mesh cell sizes can be bigger and still provide accurate results.
As introduced in the capabilities of the HEC-RAS 5.0, the program pre-processes hydraulic property tables for each cell and each one of its faces based on the geometry from the underlying terrain. This particular way of integrating the detailed geometry of the terrain in the modelling process is based on the theory developed by Casulli (2008)[11]. These tables contain a Volume-Elevation relationship for each cell and an Area-Elevation, Manning’s n-Elevation, Wetted perimeter–Elevation relationships and the Profile for each cell face. These pre-processed tables are only dependent on the properties of the terrain model, thus they can be computed prior to the start of the simulation.

In order to further expand on this matter, let a model built from a detailed terrain model (10 m resolution) with a computational mesh cell size of 100 m resolution be considered. The difference between both resolutions can be appreciated in Figure 2, where computational cell c5856 out of a mesh containing 7437 cells has been used for this example. The big square formed by black lines is the computational cell size of 100 m resolution, while the smaller squares inside the cell are the 10 m grid-cells from the terrain model used. The smaller squares of 10 m side contain the geometric information with which the hydraulic property tables will be computed.

To understand how the pre-processing takes place with regard to cell faces, it is useful to think of cell faces as if they were cross-sections of a river. In that case, it is possible to define the wetted perimeter, area, profile and Manning’s roughness of cell faces in the same way as it would be done for a regular cross-section. In Figure 3, detailing of cell c5856 and its surrounding cell faces is provided, as well as the directions of the cell

Figure 2. Comparison of terrain resolution and computational cell size
faces and highlighted in pink, cell face f12793, for which the hydraulic property tables will be shown. Examples of the computed hydraulic property tables for cell c5856 and cell face f12793 are illustrated in Figure 4, Figure 5, Figure 6, Figure 7 and Figure 8.

Figure 3. Closer look-up of a cell c5856 and its faces

Figure 4. Hydraulic property table: Cell 5856: Volume – Elevation relationship
Effect of the sub-grid geometry on two-dimensional river flow models

Ricard González Blanch

Figure 5. Hydraulic property table: Face 12793: Area - Elevation relationship

Figure 6. Hydraulic property table: Face 12793: Profile
As mentioned before, by checking the profile of the cell face in Figure 6 one can see how the elevation of the terrain is represented across the 100 m profile, which is the length of the cell face. The profile correctly depicts the terrain as the elevation value
decreases with increasing profile value (in the direction of the arrow) just like the terrain model shows (red coloured grid-cells are higher elevation values and green ones are lower elevation values). Was it not for the implementation of the sub-grid model in HEC-RAS 5.0, the profile of the cell face would show a horizontal line which would mean a single elevation value for the entire length of the cell face and would result in a severe loss of accuracy in the modelling.

Although it is possible to use larger computational cell sizes in HEC-RAS 5.0 than in other programs without losing too much of the details of the terrain, there is still a trade-off between model accuracy and computation time. This is due to the fact that the hydraulic property tables are only developed for the cell faces and only one relationship computed for the features within the cell (Cell Volume vs Elevation). If cell sizes are set up to be really high, the faces would still be able to represent the terrain accurately, whereas all the points inside the cells would not be taken into account except for the cell’s elevation-volume relationship. This happens for any cell size bigger than the grid-cell size from the terrain model, but becomes more distinct for bigger cell sizes as more points are disregarded.

Furthermore, because of the nature of the sub-grid model algorithm, the modelling will only ever be as good as the detail provided by the terrain model. Very high resolution terrain models will yield extremely accurate hydraulic property tables, thus sharply refining water flow movement through the cells. Low resolution terrain models will not be as effective because the cell faces will have less data to form the hydraulic property tables because grid-cells size from the terrain model is bigger.

HEC-RAS 5.0 produces crisp modelling results fruit of the sub-grid model. Since the flooding algorithm is based on the grid-cells from the terrain model, a computational cell size can be partially wet or dry because it is in fact the cells from the terrain model that are either fully dry or wet. Wetting of the terrain grid-cells is possible because water moves between cells based on the details of the underlying terrain, as it is represented by the cell faces and the volume contained within that cell\textsuperscript{13}. In Figure 9, a flooding event is represented using the example seen before. For instance, the cell c5856 for which the hydraulic property tables were shown earlier can be seen as almost fully dry except for some grid-cells on the top-right corner of the computational cell. This method gives far better results than other 2D modelling software that would otherwise show computational cells as either fully dry or wet.
A relevant application of these traits is for example the modelling of the flow in a channel like the one in Figure 10. In this case, the canal cuts through the computational cells which are wider than the canal itself, but is still represented by the cell’s elevation-volume relationship and the hydraulic properties of the cell faces. This verifies the sub-grid model’s efficiency as the water can run through larger cells while being represented with its normal canal hydraulic properties.

Figure 9. Wetting based on the terrain's grid-cells

Figure 10. Example of a canal thinner than computational cell size (Source: HEC-RAS 2D Modelling User’s Manual)
A drawback of the sub-grid model is that depending on how the cell faces are oriented, these will not be able to include important parts of the terrain. For that reason, alignment of the cell faces with geographic obstacles or structures such as roads, high grounds, levees or walls is paramount to good water flow simulation. If the cell faces fail to be placed along these physical barriers, these flow-controlling features will not be perceived as such by HEC-RAS as their specific geometry will not be taken into account by the cell faces. Consequently, given these circumstances, their function of blocking or hindering water flow through them is not accounted for in the simulation. This 2D mesh issue is known as leakage and consists of water leaking through the high ground and available to move further down the topography even before the high ground is overtopped[15]. Figure 11 showcases an example of leakage: water has moved from bottom cells to top cells despite the high ground feature not being overtopped.

An easy way to overcome these difficulties is by either adapting the computational mesh manually to the terrain wherever necessary, or by placing 2D Area Breaklines along the crest of high grounds features in the topography that will completely or temporarily act as a barrier to water flow. The computational mesh will align the cell faces along the defined breaklines, thus making the cell faces capture the high ground topography and reducing the leaking effect[16]. This feature outstands for its simplicity and effectiveness. A direct application of this is on the levees at the bank sides of canals and rivers, where overtopping is the determining factor to simulate inundation of the floodplains in the vicinity.
3.6 Two-dimensional unsteady flow hydrodynamics
The flow in unsteady flow conditions is governed by the conservation of mass and momentum. These conservations can be described by a set of two-dimensional equations commonly known as the Shallow Water Equations (SWE) or Saint-Venant equations, which are the equations used in HEC-RAS. These are derived from the three-dimensional Navier-Stokes motion fluid equations by making a certain number of assumptions that are only applicable in case of open channel flow and flood modelling.

The following derivations have been extracted from the HEC-RAS Hydraulic Reference Manual[17].

3.6.1 Assumptions
The assumptions made to derive the two-dimensional Shallow Water Equations are the following:

- The fluid is incompressible.
- The pressure distribution is hydrostatic.
- The flow is two-dimensional since vertical variations in flow and velocity are neglected.
- The wave lengths are much larger than the water depth.
- Almost flat bottom and small bed channel slope.
- Bed friction can be calculated using Manning’s equation.
- The flow can be described as continuous functions of the velocity and water surface elevation.

3.6.2 Continuity equation
The conservation of mass for a control volume states that the net mass flux of into the control volume equals the change in storage inside the control volume. Thus, the unsteady differential form of the mass conservation (continuity) can be written as:

$$\frac{\partial H}{\partial t} + \frac{\partial (hu)}{\partial x} + \frac{\partial (hv)}{\partial y} + q = 0$$

(1)

Where $H$ is the water surface elevation, $h$ is the water depth, $u$ and $v$ are the averaged velocities in the x- and y-direction and $q$ is a source/sink flux term.

In vector form, the equation takes the following form:

$$\frac{\partial H}{\partial t} + \nabla \cdot hV + q = 0$$

(2)
Where $V = (u, v)$ is the velocity vector and $\nabla$ is the differential operator.

### 3.6.3 Momentum equation

The conservation of momentum for a control volume states that the net rate of momentum entering the volume (momentum flux) plus the sum of all external forces acting on the volume is equal to the rate of accumulation of momentum. Considering forces from gravity, eddy viscosity, bed friction and the Coriolis effect, the momentum balance equations are presented below.

**Momentum balance in the x-direction:**

$$
\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -g \frac{\partial H}{\partial x} + \nu_t \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right) - c_f u + f v
$$

(3)

**Momentum balance in the y-direction:**

$$
\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = -g \frac{\partial H}{\partial y} + \nu_t \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right) - c_f v + f u
$$

(4)

Where $H$ is the water surface elevation, $u$ and $v$ are the velocities in the $x$ and $y$ directions respectively, $g$ is the gravitational acceleration, $\nu_t$ is the horizontal eddy viscosity coefficient, $c_f$ is the bottom friction coefficient and $f$ is the Coriolis parameter. The left hand side of the equations contains the acceleration terms whereas the right hand side represents the internal or external forces acting on the fluid.

### 3.6.4 Diffusion-wave approximation

With a view to reduce computation time and numerical instability, the Shallow Water Equations are often simplified by neglecting some of the terms in the momentum equation provided the flow conditions for these simplifications are valid. This is the case when gravity and friction are considered to be the governing forces acting on the fluid. HEC-RAS offers the possibility of using either the full momentum equation or the diffusion-wave approximation.

In the diffusion-wave approximation of the shallow water momentum equation the acceleration, the eddy viscosity and the Coriolis terms are disregarded. This leaves the original momentum equations (3) and (4) in the following vector equation form:

$$
g \nabla H = -c_f V
$$

(5)
Where $H$ is the water surface elevation, $V$ is the velocity vector, $g$ is the gravitational acceleration, $c_f$ is the bottom friction coefficient and $\nabla$ is the differential operator.

The bottom friction coefficient can be expressed by means of the Manning’s formula:

$$c_f = \frac{n^2 g |V|}{R^{4/3}} \quad (6)$$

Where $n$ is the Manning’s friction coefficient and $R$ is the hydraulic radius.

Substituting equation (6) into the vector equation form (5) and rearranging terms gives:

$$V = \frac{-(R(H))^{2/3}}{n} \frac{\nabla H}{|H|^{1/2}} \quad (7)$$

Where $R(H)$ is the hydraulic radius expressed in function of the water surface elevation $H$.

By inserting equation (7) into the original continuity equation in vector form (2), the final expression for the Diffusion-Wave approximation of the Shallow Water Equations is obtained:

$$\frac{\partial H}{\partial t} - \nabla \cdot \beta \nabla H + q = 0 \quad (8)$$

Where:

$$\beta = \frac{(R(H))^{5/3}}{n|\nabla H|^{1/2}} \quad (9)$$

3.6.5 Sub-grid model numerical algorithm

After discussing the sub-grid representation capabilities and seeing the way it interacts within the model by creating pre-computed hydraulic property tables from the fine bathymetry, it is interesting to see how that information gathered is ultimately introduced to solve the governing two-dimensional flow equations.

Since the fine bathymetry from the high resolution sub-grid model is accounted for through the mass conservation equation, it can be used independently of the version of the momentum equation used.
By integrating equation (2) over a horizontal region with boundary normal vector \( n \) and using Gauss’ Divergence theorem, the integral form of the continuity equation is derived:

\[
\frac{\partial}{\partial t} \iiint_{\Omega} d\Omega + \iint_{S} V \cdot n \, dS + Q = 0
\]  

(10)

The volumetric region \( \Omega \) represents the finite volume cell occupied by the fluid while \( S \) defines the side boundaries of the cell. It is assumed that \( Q \) stands for any flow that crosses the bottom surface (infiltration) or the top water surface of \( \Omega \) (evaporation or rain).

The integrals of equation (10) are discretized by using the fine underlying topography from the sub-grid model. Firstly, the triple volume integral represents the volume \( \Omega \) of a horizontally bounded region. Assuming that the volume within the cell is a function of the water surface elevation \( H \), the first term can be rewritten as:

\[
\frac{\partial}{\partial t} \iiint_{\Omega} d\Omega = \frac{\Omega(H^{n+1}) - \Omega(H^n)}{\Delta t}
\]  

(11)

Where the superscripts are used to index consecutive time-steps and the difference between two time-steps is \( \Delta t \).

Similarly, if the cells are assumed to have a polygonal shape with no more than eight faces, the boundary double integral of equation (10) can be expressed as the sum over the vertical cell faces of the volumetric region:

\[
\iint_{S} V \cdot n \, dS = \sum_{k} V_k \cdot n_k A_k(H)
\]  

(12)

Where \( V_k \) and \( n_k \) are the average velocity and unit normal vector at face \( k \) and \( A_k(H) \) is the area of the cross-section corresponding to cell face \( k \) as a function of the water surface elevation.

Finally, equations (11) and (12) can be put back into equation (10) to come up with the sub-grid bathymetry mass conservation equation:

\[
\frac{\Omega(H^{n+1}) - \Omega(H^n)}{\Delta t} + \sum_{k} V_k \cdot n_k A_k(H) + Q = 0
\]  

(13)

It is important to notice that the obtained equation mainly needs two relationships from the sub-grid bathymetry (found in the pre-computed hydraulic tables): volume in the cell and face area as functions of the water surface elevation, \( \Omega(H) \) and \( A_k(H) \).
respectively. However, if this information is not available, a box-scheme may be employed to compute the required relationships in the following way:

\[ \Omega(H) = Ph \]  \hspace{1cm} (14)
\[ A_k(H) = l_k h \]  \hspace{1cm} (15)

Where \( P \) is the cell area, \( l_k \) is the length of the cell face and \( h \) is the water depth that can be expressed as \( h = H - z \), with \( z \) being the surface elevation.

For the diffusion-wave approximation the same methodology behind implementing the sub-grid model is followed. However, an extra step has to be done which consists of substituting equation (7), which was derived from neglecting all terms but the friction and gravity forces from the full momentum equation, into equation (13). By doing so, the final sub-grid bathymetry diffusion-wave equation results in:

\[ \frac{\Omega(H^{n+1}) - \Omega(H^n)}{\Delta t} + \sum_k \alpha\nabla(H)\cdot n + Q = 0 \]  \hspace{1cm} (16)

Where:

\[ \alpha = \alpha(H) = \frac{(R_k(H))^{2/3}A_k(H)}{n|\nabla H|^{1/2}} \]  \hspace{1cm} (17)

3.6.6 Boundary conditions

At any given time-step, boundary conditions must be given at all the edges of the mesh domain. HEC-RAS 5.0 offers three kinds. In the following equations, the index refers to the face on which the boundary condition applies.

- Water surface elevation: the value of the water surface elevation \( H = H_b \) is given at one of the boundary edges.
- Water surface gradient: the slope of the water surface \( S_b \) in the direction normal to the boundary is imposed. This is expressed as:

\[ \nabla H \cdot n = S_b \]  \hspace{1cm} (18)

- Flow: the flow \( Q_b \) that crossed the boundary is provided. In the continuity equation (9), this is implemented by direct substitution in the flow formula of the corresponding boundary faces. The surface double integral in equation (9) is constrained by:

\[ \iint_{b} V \cdot n dS = Q_b \]  \hspace{1cm} (19)
Where the integral is taken over the boundary surface on which the boundary condition applies. Applying the sub-grid bathymetry information, the boundary constraint equation (19) is expressed as:

\[ V_b \cdot n_b A_b(H) = Q_b \] (20)

3.7 Numerical methods
Due to the fact that the Shallow Water Equations and Diffusion-Wave approximation equations are represented by a set of differential equations, they cannot be integrated analytically. For that reason, HEC-RAS resorts to using numerical methods for solving them. As seen by equations (13) and (16), the original Shallow Water Equations and Diffusion-Wave approximation respectively, are discretised in space and time. The water surface elevation and the flow velocity must be computed for each given time-step throughout the entire domain since they vary both in space and time.

The HEC-RAS two-dimensional engine uses a hybrid discretization scheme that combines the finite differences method with a finite volume method. The hybrid scheme is especially useful as it takes advantage of orthogonality in grids. The numerical solution is solved implicitly, meaning that the program iterates the solution for the next time-step instead of calculating it directly from the previous time-step, which corresponds to the explicit approach.

3.8 Computational mesh cell size and computation interval
Successful 2D modelling in HEC-RAS primarily depends on two factors: the level of terrain model representation and the simulation stability. While both are influenced by the cell size of the computational mesh, the latter is also explicitly controlled by the computation interval chosen.

The cell size to be used in the computational mesh should comply with two requirements. Firstly, as discussed in the sub-grid geometry chapter, the cells should adapt to the terrain model as precisely as possible. Secondly, the cell must have a size that allows it to interpret the water surface slope and its changes adequately. Due to the fact that the water surface elevation of a cell is only computed in its centre, the averaged water slope between two cells will decrease the larger the cells are. For that reason, rapidly varying sections require smaller cell sizes in order to have enough computation points to describe the changes in water surface and to compute the energy losses in those areas. For that purpose, cells can be manually adjusted to make them smaller in critical sections where higher precision is required.

Although low computation intervals and small cell sizes should in theory provide better results, it is not always the case due to instability issues and likely much higher
computation times. Two criteria should be contemplated when setting up these two parameters. The first criterion revolves around the computation time of the simulation, which is always preferred to be as slight as possible, and the second one deals with the stability of the simulation by means of the CFL (Courant-Friedrichs-Lewy) condition.

Two important characteristic numbers for stability consideration of transport problems are the diffusion number $D$ and the Courant number $C$, which express the ratios of the time-step size to the diffusive and convective transports, respectively$^{[18]}$. In HEC-RAS, only the implications of the Courant number are studied.

The dimensionless Courant number is defined as:

$$C = \frac{V \Delta T}{\Delta X}$$  \hspace{1cm} (21)

Where:

- $C$ = Courant number [-]
- $V$ = Total velocity, which is composed of the average water flow velocity plus the celerity [m/s]
- $\Delta T$ = Computation time-step [s]
- $\Delta X$ = Average cell size [m]

The physical meaning of the Courant number is the distance travelled by the fastest wave in one time-step expressed in terms of the mesh spacing$^{[19]}$.

The 2D HEC-RAS 5.0 Modelling User’s Manual gives some recommendations as to what Courant numbers are advised for stable and accurate simulations for two different scenarios depending on what equations are employed to solve the 2D flow of the fluid:

- **2D Saint-Venant Equations (Full Momentum):**
  
  $C \leq 1.0$ with a maximum of $C = 3$.

- **2D Diffusion-Wave Equations:**
  
  $C \leq 2.0$ with a maximum of $C = 5$.

The Manual also states some exceptions with regard to the Courant number restrictions when using the 2D Diffusion Wave equations. Specifically, in case of very rapidly rising hydrographs and routing rapidly changing hydrographs over a completely dry channel, a time-step that would produce a Courant number of 1.0 or less is suggested.
Given these constraints, the user should be able to find both suitable computational time-steps and cell sizes for the event being modelled in terms of stability. On top of that, the combination should be such that computation times are not excessively high, although that is completely left for the user to decide. As long as the stability is guaranteed, low computation time-steps and small cell sizes will produce more accurate results.
4. Pre-processing required information for simulation of the lower Ter

Before the simulation can be performed, there are a certain number of prerequisites that the model needs, which cannot be obtained from HEC-RAS, in order to function properly. Before going into the specifics, an overview of the study area is introduced to acknowledge the areas prone to flooding and the risks associated with them, as well as have a chance to get to know some characteristics of the river and its surroundings.

The data that has been processed outside of the HEC-RAS environment and is essential for the simulation are:

- Terrain model
- Boundary conditions
- Hydrograph
- Land cover classification and roughness map

The way these are obtained is further explained in the following sections.

4.1 Study area

The Ter is the longest and one of the largest rivers from the Catalonian basins. It rises in Ulldeter, at the foot of a glacial cirque, in the region of Ripollès, just a bit above the municipality of Setcases, at an approximate elevation of 2500 m. The Ter goes through many Catalonian regions such as Ripollès, Osona, Selva, Gironès and Baix Empordà until finally discharging into the Mediterranean Sea at Gola del Ter (river mouth), just downstream of the municipality of Torroella de Montgrí. The Ter follows a course of 199.7 km, drains an area of approximately 3275.6 km$^2$ and its mean discharge at the river mouth is 27 m$^3$/s.
Moreover, in spite of the fact that its headwaters are located in the Pyrenees, the Ter collects a noteworthy part of its discharge due to compelling inflow from tributaries in the middle and lower plains. Thus it is prone to flooding during the spring and autumn seasons. Some of the tributaries within the lower Ter basin that contribute to the discharge of the Ter are: el Brugent, la Riera d’Osor, el Llèmena, l’Onyar, el Terri, la Riera Gotarra and el Daró. The latter is important for our study because of the constructed canal that connects the Daró with the Ter, which will explicitly be taken into account in our geometry data as it will be shown later.

The area of study in this project focuses on the zones affected by the lower reach of the Ter close to the river mouth, which include the lower Ter basin and the Daró basin. In particular, the area of study covers a floodplain area of approximately 6657 ha which comprises vast agricultural areas and the municipality of Torroella de Montgrí and l’Estartit, hence the importance of the flooding simulation.
4.2 Terrain model

In order to carry out a 2D simulation, HEC-RAS requires a terrain model of the study area so that the program is able to identify different elevations of the surface analyzed. The terrain model was acquired through the ICGC (Institut Cartogràfic i Geològic de Catalunya) by downloading the digital elevation map (DEM) of 2x2 m resolution in text format (.txt) through their website. The corresponding sheets which represent the study area are the ones shown in Figure 14.

HEC-RAS supports a large variety of DEM formats, but unfortunately the text format is not one of them. For that reason, ArcGis 10.0 was required to convert the twelve different digital elevation maps from text to a raster format. For that purpose, the conversion tool ASCII to Raster from ArcToolBox was employed. Afterwards, the data
management tool *Mosaic to New Raster* from ArcToolBox was used to unite the twelve raster maps into a single one.

Since the aim of this project is to assess the role and implications of the sub-grid model by comparing computational cells of the same size as the ones from the terrain model with larger computational cells, an extremely high detailed model is not required. On top of that, a terrain model with less resolution will speed up the computation process for the different simulations. Based on this information, it has been opted to change the sizes from cells within the raster map containing the terrain model to 10x10 m from the original 2x2 DEM. This is done in ArcGis 10.0 by means of the data management tool *Resample* for the raster format found in ArcToolBox. The final DEM in raster format of 10x10 m resolution and containing the 12 downloaded sheets is saved as an image with .tif extension.

It is worth mentioning that at all times, the terrain model has been geo-referenced by a Spatial Reference System (SRS). The SRS elected is the ETRS89-UTM-Zone 31-N.

### 4.3 Boundary conditions

The model requires boundary conditions upstream and downstream of the area that is studied. These are essential for the model as poor boundary conditions may result in instabilities or straight false results. These boundary conditions have to be applied far away from the area of interest in order to minimize the interference with the study area. Although the project is not particularly aimed at studying a certain location, the most susceptible parts to flooding are the floodplains on the sides of the Ter in the reach between Torroella de Montgrí and the river mouth. It is also fundamental to check flooding in both municipalities Torroella de Montgrí and l’Estartit due to the potential of life losses.

The upstream boundary condition will be based on discharge hydrograph which will be introduced in the following section 4.4 *Hydrograph*. The downstream boundary condition will be set at the coast line in connection with the sea and will consist of the requirement of normal depth along all the length of the coast line of the study area. This boundary condition needs the slope of the river and can be calculated by averaging the bed slope over the whole stream length.

### 4.4 Hydrograph

A hydrograph is a graph showing stage, discharge, velocity, or other properties of water flow with respect to time. When the discharge is shown against time, the graph is called a discharge hydrograph. Since it is the most commonly used hydrograph, it is often referred to as just hydrograph. In this case, the graph shows the rate of flow (discharge) in units of m³/s versus time past a specific point in a river, canal or
conduit\textsuperscript{[23]}. Specifically, a flood hydrograph represents the discharge associated with a return period versus time. The term return period makes reference to the probability of occurrence of an event. For example, a flood discharge related to a return period of 50 years is that discharge such that, statistically speaking, it is expected to happen once in every 50 years. The higher the return period of a flood is, the higher the expected discharge becomes and therefore it results in an increase in potential harmful consequences.

A realistic hydrograph must be adopted to accurately assess the impact flood waves could potentially have on the study area. A return period of 500 years with its corresponding hydrograph has been considered. The discharge hydrograph will be one of the boundary conditions used for the flow simulation. In the best of circumstances, the discharge hydrograph that would be most suitable for the simulation is a discharge hydrograph obtained from the specific point in the river where the boundary condition is set. In order to do that, a detailed study of that cross-section would need to be carried out, however this is outside the scope of this project.

General hydrographs from the lower Ter basin are available by means of the PEF (Planificació de l’Espai Fuvial) of the lower Ter basins\textsuperscript{[21]} carried out by the ACA (Agència Catalana de l’Aigua). Consequently, it has been assumed that the alleged hydrograph corresponding to the point where the upstream boundary condition is set follow the hydrographs made by the ACA. The table below shows the hydrograph values for the flood return period considered.

\begin{table}
\centering
\begin{tabular}{|c|c|}
\hline
Time (h) & T500 Q (m$^3$/s) \\
\hline
0 & 0 \\
1 & 3.11 \\
2 & 11.8 \\
3 & 110.54 \\
4 & 577.74 \\
5 & 1732.68 \\
6 & 1993.61 \\
7 & 2289.96 \\
8 & 2878.1 \\
9 & 3059 \\
10 & 3033.28 \\
11 & 2741.68 \\
12 & 2344.7 \\
13 & 1934.44 \\
14 & 1613.78 \\
15 & 1406.19 \\
16 & 1236.01 \\
17 & 1076.25 \\
18 & 903.73 \\
\hline
\end{tabular}
\end{table}
Effect of the sub-grid geometry on two-dimensional river flow models

Ricard González Blanch

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>19</td>
<td>759.13</td>
</tr>
<tr>
<td>20</td>
<td>605.5</td>
</tr>
<tr>
<td>21</td>
<td>494.94</td>
</tr>
<tr>
<td>22</td>
<td>411.89</td>
</tr>
<tr>
<td>23</td>
<td>342.43</td>
</tr>
<tr>
<td>24</td>
<td>289.09</td>
</tr>
<tr>
<td>25</td>
<td>247.7</td>
</tr>
<tr>
<td>26</td>
<td>214.21</td>
</tr>
<tr>
<td>27</td>
<td>189.38</td>
</tr>
<tr>
<td>28</td>
<td>171.81</td>
</tr>
<tr>
<td>29</td>
<td>164.47</td>
</tr>
<tr>
<td>30</td>
<td>165.3</td>
</tr>
</tbody>
</table>

Table 2. Hydrograph values for a 500 years return period flood in the lower Ter basins

Figure 15. Hydrograph for a 500 year return period flood in the lower Ter river basins

4.5 Land cover classification and roughness map

The characterization of the roughness coefficient in hydraulic problems is very significant as it represents the resistance to flow and, as of today, quantifying said resistance still remains an arduous task which involves a lot of uncertainties. This is primarily attributed to the large amount of elements which alter its value. For instance, the roughness coefficient is a function of the material forming part of the conveyance bed, how well the latter is constructed and for how long these materials have been in use. Furthermore, the roughness coefficient is determined by means of
measurements in natural streams and laboratory tests, and cannot be derived mathematically.

One of the most renowned and widely used equations for steady flow in open channel flow is the Manning formula\(^{[24]}\):

\[ n = \frac{R^{2/3} \cdot S^{1/2}}{V} \]  

(22)

Where:

- \( n \) = Manning’s coefficient [-]
- \( R \) = Hydraulic radius [m]
- \( S \) = Slope [-]
- \( V \) = Average mean velocity [m/s]

HEC-RAS makes use of the Manning’s coefficient to quantify the resistance to flow, and has the option to either apply a constant Manning’s coefficient for the entire 2D Flow Area or divide the area in different zones, each with its own Manning’s \( n \) value. For the up-coming simulations, it has been opted to divide the 2D Flow Area in multiple zones because the study area is sufficiently large that there is a broad variety in land cover classification, resulting in a lot of different Manning’s \( n \) values. This approach brings a more realistic simulation since the distribution of land usage is recognized.

The land cover classification map of the study area is obtained from the CREAM (Centre d’ Investigació Ecològica i Aplicacions Forestals)\(^{[25]}\). Specifically, the map used is the MCSC (Mapa de Cobertes i Sòls de Catalunya) in the vector format (polygons). Initially, the latest edition, MCSC-4 (2009) was intended to be used, but as it can be appreciated in Figure 16, there was some inconsistency regarding one of the polygons. The big green triangular-shaped polygon allegedly represented an area of roads, but this is obviously false as the area is too big. This imperfection is probably due to a wrongly defined polyline.
Instead, the same map but from the third edition, MCSC-3 (2005-2007), was used. Figure 17 demonstrates how the previous complication is fixed.

The attribute table for the MCSC-3 in raster format is shown in Figure 18. Each field value (polygon) has a layer related to it describing what kind of land cover represents it. Amongst these, the most common are farms, crops, forests, different types of vegetation, kinds of buildings, etc. Another field representing the Manning roughness
coefficient is created for each layer, introducing the roughness coefficient value manually for each layer. The relationship between the type of land cover and the corresponding Manning’s n value is set according to the recommendations of the PEF from the lower Ter basins study\textsuperscript{[21]} carried out by the ACA. The table of equivalences between the land cover classification and Manning’s n coefficients is attached in the Appendix.

![Table of equivalences](image)

**Figure 18. Attribute table for MCSC-3**

After assigning the corresponding Manning’s n values to each layer, the area of study divided in roughness intervals can be appreciated in Figure 19.

![Study area divided in roughness intervals](image)

**Figure 19. Study area divided in roughness intervals**
5. Two-dimensional model development

5.1 Importing terrain model and computational mesh generation

Once the terrain model is in raster format and has the resolution desired to work with, it is necessary to import from ArcGis 10.0 into HEC-RAS, in this case the imported raster had a .tif extension. When importing the terrain model, HEC-RAS requests a spatial reference system so that it is able to show the terrain accordingly. In this case, the same SRS used for the pre-processing in ArcGis 10.0 will be selected, the ETRS89-UTM-Zone 31-N. Once the terrain map is successfully created in HEC-RAS, it is possible to visualize it in the RAS-Mapper. Figure 20 shows how RAS Mapper represents the terrain model used in this project.

![Figure 20. Visualization of the terrain model in RAS-Mapper](image)

The computational mesh can now be created in the geometric data tab from the HEC-RAS interface. An advantage HEC-RAS provides in this regard is the possibility to create the contour of the 2D computational mesh while having in the background an image. This is helpful since it becomes easy to mark out the study area by defining the limits due to having the terrain model in the background as a guide.

The computational mesh created should include all the topographic features inside the study area. Nevertheless, one should seek to adapt it smartly to the shape of the terrain to reduce its magnitude so that future computation times are diminished. For instance, the very high areas in the north and northeast part of the terrain model (they appear in white, grey and black colour in Figure 20) are likely not to be flooded, and therefore should not be included in the 2D computational mesh to make computation times less of a hassle. Later on, when discussing the results, it will be shown how the floods do not reach these neglected areas. The contour of the computational mesh can be visualized in Figure 21.
Once the contour of the mesh is created, it is necessary to input the value of the cell size so that the mesh can generate the computational cells. At any point, the cell spacing can be modified given the defined 2D Flow Area contour. This is advantageous as it allows working with multiple geometric data with the same 2D Flow Area but with different cell spacing. For the example shown in Figure 22, a spacing of 50 m has been chosen.

As it was previously mentioned, breaklines help the computational cell faces align with the terrain features. In our model, several breaklines have been created for that purpose. The breaklines have been drawn for the levees of the main course of the Ter, the Ter-Daró canal and some irrigation channels located in the area. Figure 23 shows the computational mesh of cell size 50x50 m created with the breaklines (in pink) and detail of the cell faces aligning with the breaklines.
Figure 23. 2D computational mesh (left) and detail of cell faces aligning with breaklines (right)

Upon creating the computational 2D mesh some problems may arise due to the fact that the automated mesh generation tool in HEC-RAS may occasionally create a bad cell because of the combination of the 2D Flow Area boundary and the mesh cell size defined by the user. A list with the most common mesh generation errors is found below\textsuperscript{26}. The tools found in the Geometric editor under the “Edit” menu will be of use to get around these cell creation issues. The available tools are: Add Points, Remove Points and Move Points/Objects.

- More than one cell centre in a single cell: Sometimes, on the outer boundary cells, the automatic mesh generator will create a cell with more than 1 cell centre point inside the cell (see Figure 24 below), which computationally is not allowed. What must be done is simply remove the points which represent the other cell centres in the cell that has more than one cell centre.

Figure 24. Mesh generation problem: Two cell centres in one cell (Source: Combined 1D/2D Modelling with HEC-RAS)

- Cell has no cell centre: Every computational cell must have one and one only cell centre. It may occur that the automatic mesh generator creates a cell with no cell centre (see Figure 25 below). This is easily overcome by moving the 2D area boundary points or cell centres.
• Cell Face Crosses over into Multiple Cells: Very rarely the automated mesh generator may create a cell face that extends way past that single cell (see Figure 26 below). This only happens for boundary cells, and usually where the boundary has a very sharp corner. There are various approaches to solve this problem. One of them is to add points to the boundary polygon and smooth out the boundary. The other solutions include adding more cell centres, deleting cells, or moving some cell centres.

• Cell has more than one outer boundary face: In case of an area that the 2D Polygon necks down, the automatic mesh generator may create a cell that has two outer boundary faces (see Figure 27 below). Currently this is not working computationally speaking and is expected to be fixed in future HEC-RAS patch
releases. In order to fix it, one must break up the cell into multiple cells such that each cell has only one outer boundary face. A boundary face can be made up of many points/sides, but it must be continuous.

Figure 27. Mesh generation problem: Cell has more than One Outer Boundary Face (Source: Combined 1D/2D Modelling with HEC-RAS)

- Too Many Faces on a Cell: Provided a cell has more than 8 sides, the cell and/or cells that bound it must be edited since each cell is limited to having 8 faces only. This can be done by making use of the aforementioned editing tools from the Geometric editor.

5.2 Boundary conditions

The upstream boundary condition will consist of the discharge hydrograph whereas the downstream boundary will be based on normal depth. In HEC-RAS 2D, it must be specified where the boundary conditions are applied by means of drawing BC (Boundary condition) lines on the 2D Flow Area. The upstream boundary condition is applied along the width of the most upper cross-section of the Ter, while the downstream boundary condition is applied along the coast line.

Figure 28. Boundary condition on the mesh. Downstream BC (left) and Upstream BC (right)

Finally, the flow hydrograph and normal depth data are introduced in the unsteady flow data tab as shown in Figure 29.
5.3 Land cover layer

The land cover classification map with associated Manning’s n values must be imported into HEC-RAS so that the model can utilize the roughness coefficients for each area instead of having to assume the same one for the entirety of the 2D Flow Area. Afterwards, the imported land cover layer must be associated with the geometric data of the model.)
generated for 2D Flow Areas as it can be seen in Figure 31. Afterwards, the only thing left to do is associate the land cover map with the geometric data.

![Figure 31. Land cover layer in HEC-RAS for 2D Flow Areas](image)

5.4 Two-dimensional unsteady flow simulation parameters
After defining the geometry of the simulation, including the terrain model, the land cover layer and the computational mesh, and the unsteady flow file containing the boundary conditions, only the computational settings are left to start simulating. After clicking on the “Run unsteady flow analysis” tab, a window similar to Figure 32 will pop up.

![Figure 32. Unsteady Flow Analysis tab](image)
The remaining parameters left to be introduced are:\textsuperscript{[27]},

- Computation Interval: This is the known computational time-step that is crucial to the model’s overall workability. It should satisfy two primary conditions. Firstly, it must be able to depict the flow hydrograph perfectly, and secondly, it must verify the Courant condition depending on what equations are solved. A general rule of thumb is to use a computation interval that is equal to or less than the time of rise of the hydrograph divided by 20. This parameter is often the root of many instability issues.

- Hydrograph Output Interval: The Hydrograph Output Interval is used to define at what interval the computed stage and flow hydrographs will be written to HEC-DSS. This interval should be selected to give an adequate number of points to define the shape of the computed hydrographs without losing information about the peak or volume of the hydrographs. This interval must be equal to or larger than the selected computation interval.

- Mapping Output Interval: This field is used to enter the interval at which the user will be able to visualize mapping output within HEC-RAS Mapper.

- Detailed Output Interval: This field allows the user to write out profiles of water surface elevation and flow at a user specified interval during the simulation. Profiles are not written for every computational time-step because it would require too much space to store all of the information for most jobs. The selected interval must be equal to or greater than the computation interval. However, it is suggested that this interval is made fairly large in order to reduce the amount of post-processing and storage required for a detailed hydraulic output.

For our simulations, all the output intervals are set to 1 hour because the hydrograph values are given for each hour. With regard to the computation interval, initially it will be set at 30 minutes, and depending on how the stability of the model evolves for the mesh spacing, it may be modified. The choice of the 30 minutes as the initial computation time-step is due to the aforementioned rule of thumb (hydrograph takes 9 hours to reach its peak, 1/20 of 9 hours is 27 minutes which is rounded to 30 minutes). The appropriate time-step will be analysed in section 6.1 \textit{Computation interval calibration}. Moreover, the flow regime is considered to be mixed because as the flow discharge will vary, fluctuations between subcritical and supercritical flow regime may occur.
6. Results

Multiple simulations have been run with varying computation intervals and mesh cell sizes. Ultimately, the verification of the sub-grid model role is achieved by comparing between different mesh cell sizes, so it has been opted to start off with a computation interval (30 minutes) and check whether the stability is assured for the different mesh cell sizes. In case they are not, other computation intervals will be contemplated for each mesh cell size.

Due to the fact that HEC-RAS does not have an option to deactivate the sub-grid model feature, a solution must be figured out in order to have the possibility to compare between simulations that incorporate the sub-grid model and those who do not. The sub-grid model can be neglected in a way if the cell size of the computational mesh is the same as the resolution of the underlying terrain. The reason for this is that if the cells have the same size as the resolution of the terrain, then the pre-calculated hydraulic tables will be of no use since the cells alone are already capable of capturing the maximum detail provided by the terrain model.

The comparison will then revolve around how much accuracy is lost using higher mesh cell sizes compared to a cell size of 10x10 m (mesh cell size that would neglect the sub-grid model since it is the resolution of the terrain model used) which is set as the reference model with no sub-grid representation for future comparisons, and at the same time how much is gained with respect to computation time. That is why, for each simulation, the computation time will be noted down as it is a determining factor to take into account. It is worth mentioning the computation time will not include the time HEC-RAS takes to develop the hydraulic property tables as these will be computed beforehand.

The comparison between different cell sizes can only take place if the simulations for each mesh cell size are stable. Additionally, since computation times play an important role in the comparison, an effort should be made to find the lowest computation time, so the lowest computation interval, that is able to provide accurate results and a stable simulation. Therefore, the first task is to find the most appropriate computation intervals for the following mesh cell sizes: 10x10, 25x25, 50x50 and 100x100 m. These mesh cell sizes will tell to what extent the sub-grid model offers good results for larger sizes than the terrain model resolution. Firstly, the rule of thumb assumption for the computation interval suggested by the HEC-RAS manual will be tested for all the mesh cell sizes and depending on the results further detailing will ensue.

Once the computation interval is set for each mesh cell size model, the gathering of map results of flooding extent, maximum water depth, maximum flow velocity and
time arrival for each mesh cell size model will ensue. Lastly, comparisons between the 100x100, 50x50 and 25x25 m mesh cell sizes models and the 10x10 m mesh cell size model will be performed for each variable analysed.

6.1 Computation interval calibration

6.1.1 Initial computation interval assumption for all mesh cell sizes

<table>
<thead>
<tr>
<th>Mesh cell size (m)</th>
<th>Computation interval</th>
<th>Computation time [hh:mm:ss]</th>
<th>Stability</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>30 min</td>
<td>00:00:41</td>
<td>Unstable</td>
</tr>
<tr>
<td>25</td>
<td>30 min</td>
<td>00:00:06</td>
<td>Unstable</td>
</tr>
<tr>
<td>50</td>
<td>30 min</td>
<td>00:00:02</td>
<td>Unstable</td>
</tr>
<tr>
<td>100</td>
<td>30 min</td>
<td>00:00:02</td>
<td>Unstable</td>
</tr>
</tbody>
</table>

Table 3. Computation times for different mesh cell sizes with a 30 minute computation interval

Only by checking the maximum depth results shown above, it is clear that neither one of the simulations was stable. The results do not make sense because they show very large water depths and only the upstream reach of the river and floodplains are flooded. This is a clear example of how the computation interval was too high as the flood wave could not even develop along the entirety of the river.

Moreover, in the case of mesh cell size of 10x10 m, the compute messages after the simulation revealed instabilities as some errors revealed. These errors can be seen in Figure 34.
Figure 34. Errors in the computation messages window for a 10x10m mesh cell size and a 30 min computation interval (only shows part of the long list)

The number 2 under the convergence column means that for that specific cell, the maximum number of iterations was reached without converging. In other words, the water surface elevation could not be calculated appropriately for that cell due to the large computation interval.
A time series plot for a point in one of the floodplains close to the upstream boundary condition is shown in Figure 35. This plot demonstrates the variety of behaviours depending on the size of the mesh cells. Furthermore, the sudden rise of water depth in all cases is another way to see the instability of the simulations. A stable simulation would show a progressive increase and decrease of the water depth with respect to time.

Assuming the same computation interval has served to realise the distinct behaviour between the different mesh cell sizes. In addition, it has been demonstrated that the rule of thumb consisting of setting the computation interval equal to 1/20 of the time it takes to reach the peak of the flood is nowhere suitable for the simulations within this project.

Because of the diverse behaviour between mesh cell sizes, different computation intervals will now be tested for each mesh cell size studied.

The full process on how to derive the optimal computation interval for a given cell size is explained thoroughly for the case of a mesh cell size of 100x100 m. For the other cases, since the process does not vary, only final figures showing the results will be provided.
6.1.2 Mesh cell size of 100x100 m

6.1.2.1 Appropriate computation interval for a 100x100 m mesh cell size

The methodology followed to determine what is the maximum computation interval with trustworthy results is to find the computation interval threshold from which the results converge. Both by plotting time series of water depth evolution at a point and by viewing the overall flood spot distribution it will be possible to determine whether a simulation has converged or not.

<table>
<thead>
<tr>
<th>Computation interval</th>
<th>Computation time [hh:mm:ss]</th>
</tr>
</thead>
<tbody>
<tr>
<td>30 min</td>
<td>00:00:02</td>
</tr>
<tr>
<td>10 min</td>
<td>00:00:07</td>
</tr>
<tr>
<td>5 min</td>
<td>00:00:14</td>
</tr>
<tr>
<td>3 min</td>
<td>00:00:24</td>
</tr>
<tr>
<td>1 min</td>
<td>00:00:49</td>
</tr>
<tr>
<td>5 s</td>
<td>00:05:58</td>
</tr>
<tr>
<td>1 s</td>
<td>00:19:46</td>
</tr>
</tbody>
</table>

Table 4. Computation times for initial computation interval simulation for a 100x100 m mesh cell size

Table 4 shows the computation times for a set of simulations. The computation intervals of the simulations were chosen arbitrarily although an attempt was made to select a wide range of possible values. For starters, each mesh cell size will include computation interval simulations of 30 minutes, 10 minutes, 5 minutes, 3 minutes, 1 minute, 5 seconds and 1 second. As it will be seen, it is possible more simulations are needed to narrow down the possible optimal computation interval values.

The maximum depths for the flood propagation for each computation interval are represented in the following figures:
A 5 minute computation interval seems to reduce the maximum water depths from the previous simulation considerably. Some unflooded areas appear (red rectangles in Figure 37) with respect to the previous simulation.
Figure 38 shows the maximum water depth map for a computation interval of 3 minutes. The areas in the red rectangles show new unflooded areas with respect to previous computation interval (5 minutes, Figure 37). Also the areas which were already unflooded in Figure 37 seem to have increased in area.

Figure 39 shows how only two new unflooded areas (red rectangles again) have appeared with respect to a computation interval of 3 minutes. Past unflooded areas may have slightly increased. The difference between a computation interval of 3 minutes and 1 minute is very slim, so convergence must lie within this or the following computation intervals chosen.
The difference between a computation interval of 1 minute, 5 seconds and 1 second (Figure 39, Figure 40 and Figure 41 respectively) cannot be perceived visually. This is either due to the fact that convergence has been reached, or that the difference is so small that it can’t be noticed.

One way to visualize the differences amongst the previous simulations is to draw profile lines across the study area and plot the maximum water depth along those profile lines for each simulation at a certain time. Since the water flow behaviour may change depending on the profile studied, it has been decided to draw three different profile lines: one near the upstream boundary condition, one in the middle reach of the river
and lastly, one next to the downstream boundary condition. The profile lines drawn can be visualized in Figure 42. This methodology will be useful to acquire a more thorough examination of the differences in maximum water depth between the simulations.

![Profile lines (in pink) drawn for plotting the simulations](image)

Figure 42. Profile lines (in pink) drawn for plotting the simulations

![Maximum water depth plot along the upstream profile](image)

Figure 43. Maximum water depth plot along the upstream profile for a 100x100 m mesh cell size and for computation intervals: 10 min, 5 min, 3 min, 1 min, 5 s and 1 s
Figure 44. Maximum water depth plot along the upstream profile line for a 100x100 m mesh cell size and for computation intervals: 5 min, 3 min, 1 min, 5 s and 1 s

Figure 43 presents how the simulation with a computation interval of 10 minutes overestimates the maximum water depths, therefore the 10 minutes computation interval is already ruled out. In the plot, only two simulations are visible (10 minute one and 1 second one) because the rest is hidden under the simulation of 1 second (brown in the graph). Proof of that is that if the 10 minute simulation is removed, a plot with again only the 1 second simulation visible appears. This can be seen in Figure 44, and it is obvious because the graph line is thicker. The fact that these simulations appear to be so similar in the upstream profile line might lead to believe that the preferred computation interval for the case of a mesh cell size of 100x100 m is 5 minutes. However, the other profile lines must be checked as water flow behaviour may be unalike.
Effect of the sub-grid geometry on two-dimensional river flow models

The middle profile lane plot (Figure 45) tells a different story. In this case, the 5 minute computation interval simulation differs quite a lot from the rest, displaying higher water depths than the other simulations. In addition, from station 500 meters to 1000 meters, it can be seen how the 3 minute computation interval simulation water depths marginally surpass the simulations of computation intervals of 1 minute, 5 seconds and 1 second. Thus, the 5 and 3 minute computation interval are also discarded as they can’t be assumed to have reached convergence with respect to the other simulations.

Finally, the last downstream profile line is tested for the simulations with computation intervals that have so far given the same results (so all of them excluding the 10 minutes, 5 minutes and 3 minutes simulations). The corresponding plot for the downstream profile line is shown in Figure 46.
Figure 46 reveals that indeed the simulations with computation intervals of 1 minute, 5 seconds and 1 second give very similar results, as from the plot only one of the simulations can be seen. This means that for a computation interval of 1 minute, the results have already converged because even if the computation interval is further lowered, the results stay the same. The difference in water depths between the 1 minute and 1 second simulations is of the order of 1 mm (extracted from the tables) which of course is negligible. In addition, 1 mm errors will be the approximate standard set error for the rest of the mesh cell sizes to decide whether two simulations have converged.

This is in line with what was analysed from the visual inspections of the flood inundation results, where it was observed that flood inundation results were extremely similar between the simulations of computation intervals of 1 minute, 5 seconds and 1 second while the rest of simulations did show discrepancies between them in terms of unflooded and flooded areas.

From these simulations we have gathered enough data to determine that for a 1 minute simulation the results have converged whereas for a 3 minute simulation it has not. Since the target computation interval is the highest computation interval that still gives as accurate results as the simulation with lowest computation interval, it is
necessary to check whether a 2 minute simulation (only possible simulation left between the 1 minute and 3 minute computation interval) is also capable of reproducing such behaviour.

Figure 47. Maximum water depth plot along the downstream profile line for a 100x100 m mesh cell size and for computation intervals: 2 min, 1 min, 5 s and 1 s
Figure 47 points out how for the downstream profile line at around station 2600 m, the 2 minute simulation is not consistent with the simulations with lower computation interval. Therefore, the 2 minute simulation is completely neglected and it has now been verified that indeed the 1 minute simulation is the highest computation interval simulation for which results are trustworthy. Moreover, Figure 48 demonstrates how a 2 minute simulation implies an error of around 4-5 mm whereas the 1 minute simulation sticks to the 1 mm error range set beforehand.

As a result, for a mesh cell size of 100x100 m, a computation interval of 1 minute gives the same accuracy in the results as a 5 seconds and 1 second simulation. The advantage in using the 1 minute computation interval is the reduction in computation time. The computation times for each simulation can be checked in Table 3. A computation interval of 1 minute yields practically the same results while doing it 5 minutes faster than a computation interval of 5 seconds and 19 minutes faster than a computation interval of 1 second. In conclusion, a 1 minute computation interval is the most suitable for a mesh cell size of 100x100.
In section 5.1 Importing terrain model and computational mesh generation it was argued how the computational 2D mesh should not include the high elevation areas of the map in order to save computation time. Figure 49 corroborates this theory as water flow does not reach the higher parts of the study area (red and grey pixels in the image). Once it has proven that water is likely not to flood certain parts, additional adjustments to the 2D mesh contour may be done so that the mesh contains fewer cells and therefore run times decrease. Nevertheless, it is highly recommended to verify flooding events are not neglected when modifying the 2D computational mesh for other mesh cell sizes and computation intervals as results could differ.

6.1.2.2 Stability of the 100x100 m mesh cell size model

The last step involves checking whether the selected simulation is stable. In order to proceed with this, it is necessary to go back to section 3.8 Computational mesh cell size and computation and make use of the Courant definition, as well as verify the recommendations for these values so as to assure the model stability.

\[
C = \frac{V \Delta T}{\Delta X} = \frac{(v + c) \Delta T}{\Delta X} = \frac{(v + \sqrt{gh}) \Delta T}{\Delta X}
\]  

(23)

The first term of the velocity makes reference to the average flow velocity whereas the second term accounts for the celerity of the flood wave.

In ideal conditions, checking the stability of the model would imply verifying, for all cells and for each time-step, the Courant condition. This is not possible because it would be extremely time-consuming and because HEC-RAS does not output sufficient
data to accomplish this. One way to get around the problem would be by calculating the Courant number for each cell in extreme situations of maximum velocity or maximum water depth, as they would result in the highest Courant number across all time-steps performed.

HEC-RAS provides through RAS Mapper the results of the maximum water depths and maximum velocities for each cell achieved during the simulation. The only issue is that maximum water depths and maximum velocities are not bound to take place at the same time, so it is very likely that those two occur at different time-steps during the simulation. However, by taking the maximum water depth and the maximum velocity of a cell during the simulation, it is feasible to compute an upper bound for the Courant number of that cell. This will ensure that the Courant number for that specific cell will not surpass the computed upper bound, as maximum values of water depth and velocity have been taken for the calculation.

The maximum water depth and velocity maps of the 1 minute simulation for a cell size of 100x100 m were exported into ArcGis 10.0 to go on with the calculations of the Courant number. A new raster was created from those two by means of introducing the equation to calculate the Courant number through the Raster Calculator from the Spatial Analyst Tools from ArcToolBox. The time-step was set equal to $\Delta T = 60$ s, $\Delta X = 100$ m and $g = 9.81$ m/s$^2$. For future simulation stability analysis, the time-step and the size of the mesh cell will have to be modified according to the parameters of the simulation being treated. The water depth and velocity values were extracted from the corresponding maximum water depth and maximum velocity raster maps.

Figure 50 displays the Courant number map calculation for a 1 minute computation interval on the 100x100 m mesh cell size. According to the HEC-RAS 2D Modelling
User’s Manual, when using the 2D Diffusion-Wave equations the Courant number should be limited to a maximum of 5. By taking a look at the map, it is possible to spot only a few small areas in the course of the river where the upper bound Courant number is greater than 5. If it is considered that an upper bound was computed and that the actual high Courant numbers depicted on the map are rarely reached because maximum velocities and water depths are mutually exclusive, then it is safe to say that the model stability is assured. Moreover, areas with higher upper bound Courant number than 5 are only found in the river course so the floodplain results would not be affected as much.

6.1.3 Mesh cell size of 50x50 m

6.1.3.1 Appropriate computation interval for a 50x50 m mesh cell size

As mentioned previously, the process followed to obtain the computation intervals is the same as in the case of a cell size of 100x100 m. Only relevant plots and figures showing the final results will be given.

<table>
<thead>
<tr>
<th>Computation interval</th>
<th>Computation time [hh:mm:ss]</th>
</tr>
</thead>
<tbody>
<tr>
<td>30 min</td>
<td>00:00:02</td>
</tr>
<tr>
<td>10 min</td>
<td>00:00:16</td>
</tr>
<tr>
<td>5 min</td>
<td>00:00:50</td>
</tr>
<tr>
<td>3 min</td>
<td>00:01:37</td>
</tr>
<tr>
<td>1 min</td>
<td>00:04:39</td>
</tr>
<tr>
<td>5 s</td>
<td>00:28:41</td>
</tr>
<tr>
<td>1 s</td>
<td>01:11:02</td>
</tr>
</tbody>
</table>

Table 5. Computation times for initial computation interval simulations for a 50x50 m mesh cell size
Figure 51. Maximum water depth results for a 50x50 m mesh cell size and a 10 min computation interval

Figure 52. Maximum water depth results for a 50x50 m mesh cell size and a 5 min computation interval

Figure 51 and Figure 52 indicate how for computation intervals of 10 minutes and 5 minutes the flood wave has not stabilized in the case of cell sizes of 50x50 m whereas for a 100x100 m mesh cell size it has. This is already an indicator of how smaller mesh cell sizes require lower computation intervals due to the Courant condition regarding model stability.
Additional computation intervals were required to find out the largest computation interval that had converging results with the lower computation intervals (around 1 mm of error was again the accepted value).

<table>
<thead>
<tr>
<th>Computation interval</th>
<th>Computation time [hh:mm:ss]</th>
</tr>
</thead>
<tbody>
<tr>
<td>30 s</td>
<td>00:07:38</td>
</tr>
<tr>
<td>15 s</td>
<td>00:12:49</td>
</tr>
<tr>
<td>12 s</td>
<td>00:13:58</td>
</tr>
<tr>
<td>10 s</td>
<td>00:16:30</td>
</tr>
</tbody>
</table>

Table 6. Computation times for extra computation interval simulations for a 50x50 m mesh cell size

The comparison of water depths from each simulation is carried out for the downstream profile line as it is the most restraining out of the three profile lines, meaning more difference in water depth results between different computation interval simulations. This is due to this profile line being the furthest away from where the hydrograph data is introduced. This translates into water having to flow and develop across more cells, thus dragging the inaccuracies from past computation cells in each time-step.

Figure 53. Maximum water depth plot along the downstream profile line for a 50x50 m mesh cell size and for computation intervals of: 12 s, 10 s, 5 s and 1s
Finally it was found out a computation interval of 15 seconds produced errors of around 5 mm so the next computation interval was tested (12 seconds). As shown in Figure 54, the 12 seconds simulation resulted in less noticeable errors (around 2 mm) and therefore was chosen as the definite computation interval.

6.1.3.2 Stability of the 50x50 m mesh cell size model

Figure 54. Closer look-up of the four final simulations for a 50x50 m mesh cell size

Figure 55. Upper bound Courant number map for a 12 s computation interval on a 50x50 m mesh cell size
Figure 55 shows that the stability for the 12 second time-step chosen on a 50x50 m mesh cell size is assured as all cells comply with the upper Courant number condition that states that for the 2D Diffusion-Wave equations such number should be limited to 5. In fact, the majority of the cells upper bound for the Courant number lie within the recommended interval of $C \leq 2.0$.

6.1.4 Mesh cell size of 25x25 m

6.1.4.1 Appropriate computation interval for a 25x25 m mesh cell size

<table>
<thead>
<tr>
<th>Computation interval</th>
<th>Computation time [hh:mm:ss]</th>
</tr>
</thead>
<tbody>
<tr>
<td>30 min</td>
<td>00:00:06</td>
</tr>
<tr>
<td>10 min</td>
<td>00:00:30</td>
</tr>
<tr>
<td>5 min</td>
<td>00:02:20</td>
</tr>
<tr>
<td>3 min</td>
<td>00:06:21</td>
</tr>
<tr>
<td>1 min</td>
<td>00:23:25</td>
</tr>
<tr>
<td>5 s</td>
<td>02:41:00</td>
</tr>
<tr>
<td>1 s</td>
<td>04:55:22</td>
</tr>
</tbody>
</table>

Table 7. Computation times for initial computation interval simulations for a 25x25 m mesh cell size

For a mesh cell size of 25x25 m, the models seem to be stable from the visual analysis of the flood wave propagation for computation intervals lower than 3 minutes. A computation interval of 3 minutes is the lowest computation interval that produces unstable results, as shown in Figure 56.

<table>
<thead>
<tr>
<th>Computation interval</th>
<th>Computation time [hh:mm:ss]</th>
</tr>
</thead>
<tbody>
<tr>
<td>30 s</td>
<td>00:44:27</td>
</tr>
<tr>
<td>15 s</td>
<td>01:15:14</td>
</tr>
<tr>
<td>10 s</td>
<td>01:52:00</td>
</tr>
<tr>
<td>6 s</td>
<td>02:24:45</td>
</tr>
<tr>
<td>1 s</td>
<td>04:55:22</td>
</tr>
</tbody>
</table>

Table 8. Computation times for extra computation interval simulations for a 25x25 m mesh cell size
Effect of the sub-grid geometry on two-dimensional river flow models

Ricard González Blanch

Figure 56. Unstable wave propagation for a 25x25 m mesh cell size and computation interval of 3 min

Figure 57. Maximum water depth plot along the downstream profile line for a 25x25 m mesh cell size and for computation intervals of: 10 min, 5 min, 3 min, 1 min, 30 s, 15 s, 10 s, 6 s, 5 s, 1 s
Effect of the sub-grid geometry on two-dimensional river flow models

Ricard González Blanch

Figure 58. Maximum water depth plot along the downstream profile line for a 25x25 m mesh cell size and for computation intervals of: 10 s, 6 s, 5 s and 1 s

Figure 59. Closer look-up of the four final simulations for a 25x25 m mesh cell size
After reducing the simulations to four computation intervals (Figure 58), a closer look-up (Figure 59) was required in order to discern the differences between them with more precision.

After taking a closer look at the simulations, it is concluded that a 10 second computation interval is enough for results to converge within the 1 mm range and therefore is elected as the appropriate time-step for this mesh cell size. Notice how the simulations of 6 seconds and 5 seconds are exceptionally good approximations for the 1 second model as the error is actually in the magnitude of a hundredth of a millimetre. Although these simulations are very accurate it is not in the interest of the project to consider them as they require higher computation times. The 10 second computation interval suffices because a 1 mm error in water depth prediction is acceptable.

6.1.4.2 Stability of the 25x25 m mesh cell size model

![Figure 60](image.png)

The stability is assured because only very isolated cells in the river have an upper bound Courant number higher than 5 as is presented in Figure 60.

6.1.5 Mesh cell size of 10x10 m

6.1.5.1 Appropriate computation interval for a 10x10 m mesh cell size

The results of the first round of simulations are shown in the table below.

<table>
<thead>
<tr>
<th>Computation interval</th>
<th>Computation time [hh:mm:ss]</th>
</tr>
</thead>
<tbody>
<tr>
<td>30 min</td>
<td>00:00:02</td>
</tr>
<tr>
<td>10 min</td>
<td>00:01:04</td>
</tr>
<tr>
<td>5 min</td>
<td>00:03:16</td>
</tr>
<tr>
<td>3 min</td>
<td>00:17:25</td>
</tr>
</tbody>
</table>
Effect of the sub-grid geometry on two-dimensional river flow models

Ricard González Blanch

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1 min</td>
<td>02:49:05</td>
</tr>
<tr>
<td>5 s</td>
<td>30:50:41</td>
</tr>
<tr>
<td>1 s</td>
<td>54:48:32</td>
</tr>
</tbody>
</table>

Table 9. Computation times for initial computation interval simulation for a 10x10 m mesh cell size

For instance, computation intervals of 3 minutes and 1 minute and greater values did not even produce stable results (see Figure 61 and Figure 62 below).

Figure 61. Maximum water depth results for a 10x10 m mesh cell size and a 3 min computation interval

Figure 62. Maximum water depth results for a 10x10 m mesh cell size and a 1 min computation interval
This again proves the predicted trend of smaller mesh cell sizes requiring lower computation intervals. This effect is very noticeable as a computation interval of 1 minute resulted to be the optimal computation interval for a mesh cell size of 100x100 m whereas for a mesh cell size of 10x10 m it does not even provide sufficient stability to even consider the results.

<table>
<thead>
<tr>
<th>Computation interval</th>
<th>Computation time [hh:mm:ss]</th>
</tr>
</thead>
<tbody>
<tr>
<td>30 s</td>
<td>07:04:02</td>
</tr>
<tr>
<td>10 s</td>
<td>17:39:12</td>
</tr>
<tr>
<td>3 s</td>
<td>39:48:37</td>
</tr>
</tbody>
</table>

Table 10. Computation times for extra computation interval simulations for a 10x10 m mesh cell size

The conclusions of which computation interval best suits a mesh cell size of 10x10 m are drawn out from the downstream profile line analysis for the reasoning detailed in the previous mesh cell size case.

Figure 63. Maximum water depth plot along the downstream profile line for a 10x10 m mesh cell size and for computation intervals of: 10 min 5 min, 3 min, 1 min, 30 s, 10 s, 5 s and 3s

Figure 63 clarifies what was foreseen in Figure 61 and Figure 62, where the computation intervals of the corresponding simulations and higher values were not appropriate for stability.
Finally, the selection of the convenient computation interval is reduced to four simulations as shown in Figure 64. A closer look-up of these simulations (see Figure 65 below) clears the uncertainties regarding which simulation is best cut out for this mesh cell size. Particularly, the 30 second and 10 second simulations yield water depth outcomes which surpass the 1 mm standard set for convergence. For that reason, a 5 second computation interval is opted for despite resulting in an error between the 5 second and 3 second simulations much smaller than 1 mm.

![Figure 64. Maximum water depth plot along the downstream profile line for a 10x10 m mesh cell size and for computation intervals of: 30 s, 10 s, 5 s and 3s](image-url)
6.1.5.2 Stability of the 10x10 m mesh cell size model

The upper bound Courant number map for a 10x10 m mesh cell size with the right time-step (Figure 66) is very similar to the one of the 100x100 m mesh cell size. As a result, similar conclusions apply in this case. There are only a few areas with higher upper bound Courant numbers than the highest recommended, and those are located along the Ter so the floodplains are not affected much by this fact. It is worth
reminding this map represents an upper bound, therefore the real Courant numbers for each cell in each time-step ought to be lower.

6.2 Maximum water depth, maximum flow velocity and arrival time map results
A total of four variables were tested for each one of the simulations. The selection of these variables was done according to the importance they could have in a real life fluvial flooding application. Under that condition, and checking that the results could be computed from the RAS Mapper, the following were chosen for analysis: flooding extent, maximum water depth, maximum flow velocity and arrival time.

Figure 67. Maximum water depth map for a 100x100 mesh cell size and a 60 s computation interval model

Figure 68. Maximum water depth map for a 50x50 mesh cell size and a 12 s computation interval model
Effect of the sub-grid geometry on two-dimensional river flow models

Figure 69. Maximum water depth map for a 25x25 mesh cell size and a 10 s computation interval model

Figure 70. Maximum water depth map for a 10x10 m mesh cell size and a 5 s computation interval model

Figure 71. Maximum flow velocity map for a 100x100 m mesh cell size and a 60 s computation interval model
Figure 72. Maximum flow velocity map for a 50x50 m mesh cell size and a 12 s computation interval model

Figure 73. Maximum flow velocity map for a 25x25 m mesh cell size and a 10 s computation interval model

Figure 74. Maximum flow velocity map for a 10x10 m mesh cell size and a 5 s computation interval model
Effect of the sub-grid geometry on two-dimensional river flow models

Ricard González Blanch

Figure 75. Arrival time map for a 100x100 m mesh cell size and a 60 s computation interval model

Figure 76. Arrival time map for a 50x50 m mesh cell size and a 12 s computation interval model

Figure 77. Arrival time map for a 25x25 m mesh cell size and a 10 s computation interval model
6.3 Sub-grid model comparison analysis

Once each mesh cell size model is adjusted to a fitting computation interval, it is feasible to perform an analysis between the different geometries to find out how the sub-grid geometry models fare with respect to the 2D simulations modelled. In particular, the 100, 50 and 25 m mesh cell sizes will be contrasted with the 10 m simulation which represents the special case where the sub-grid geometry is theoretically neglected. Comparisons will be made taking into consideration the 10x10 m mesh cell size model provides more accurate results due to a lower mesh cell size and computation interval. The corresponding computational intervals for the simulations are 60 seconds, 12 seconds, 10 seconds and 5 seconds respectively, as derived earlier in section 6.1 Computation interval calibration. The following discussions will be based on the post-processing of the mapping results shown in section 6.2 Maximum water depth, maximum flow velocity and arrival time map results.

<table>
<thead>
<tr>
<th>Mesh cell size [m]</th>
<th>Computation interval [s]</th>
<th>Computation time [hh:mm:ss]</th>
</tr>
</thead>
<tbody>
<tr>
<td>100x100</td>
<td>60</td>
<td>00:00:49</td>
</tr>
<tr>
<td>50x50</td>
<td>12</td>
<td>00:13:58</td>
</tr>
<tr>
<td>25x25</td>
<td>10</td>
<td>01:52:00</td>
</tr>
<tr>
<td>10x10</td>
<td>5</td>
<td>30:50:41</td>
</tr>
</tbody>
</table>

Table 11. Summary of simulations to compare

6.3.1 Flooding extent

The extent of a flood is the primary aspect concerning the study of floods as it determines which areas are affected by the flood and which are safe. For now, the analysis of the magnitude of water depths is disregarded because the distinction will be made between cells that are wet at some point in the 30 hour simulation and those that remain dry for the entirety of the simulation. With that in mind, the maximum water
depth maps exported from HEC-RAS are going to be employed as they will discretise cells whose maximum water depths are above 0 into cells that have been wet at some point and those that have a maximum water depth equal to 0 into dry cells. Comparing the flooding extent in this way ensures a safe assessment with respect to the flooding capabilities as it includes cells that get wet in the process but end up drying, meaning that at some point whatever lies within those cells is exposed to related flooding hazards and therefore should be taken into account.

The post-processing of the maximum water depth maps has been carried out in ArcGis 10.0 by means of the Reclassify tool which allowed grouping cells into either wet or dry categories. Finally, the flooding extent maps of the 100x100, 50x50 and 25x25 m mesh cell sizes models were overlapped with the 10 m model to find out what cells are considered to be wet in the sub-grid models and dry in the 10x10 m model and vice-versa.

Figure 79. Flooding extent difference between the 10x10 m and the 100x100 m models
Figure 79, Figure 80 and Figure 81 showcase the flooding extent differences between the 3 mesh cell sizes models and the 10x10 m model. In all of these figures, only the cells coloured in yellow or blue present differences in the wetting process of the cells. Therefore, if a cell is not painted in any of these colours it means that the cell was dry or wet in both of the models being compared which suggests the sub-grid model does performs well when predicting the flooding extent.

From a visual examination it becomes evident that when reducing the sub-grid model cell size the differences are prone to decreasing as the overall yellow and blue areas diminish. However, the differences between the yellow and blue areas have a nuance attached to them that is worth mentioning.

The blue areas represent those areas that appear to be flooded in the sub-grid model (100x100, 50x50 or 25x25 m cell size in each case) but not in the 10 m mesh cell size
model. In spite of the fact that these areas display wrong flooding extents, they are not hazardous because if the sub-grid models were considered to model the flooding extent, the blue areas would represent an over-estimation of said flooding extent, thus staying on the safe side since the model predicts areas to be flooded when in reality they should not. Unfortunately, the yellow areas are a consequence of the opposite phenomena, meaning that the sub-grid models are not able to capture these areas as wet when they end up flooded in the 10x10 m model. Thus, in these specific areas, the sub-grid models are not representative of a safe approach.

<table>
<thead>
<tr>
<th>Flooded extent comparison with the 10x10 model</th>
<th>100x100</th>
<th>50x50</th>
<th>25x25</th>
<th>10x10</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yellow area cell count</td>
<td>21378</td>
<td>12017</td>
<td>5801</td>
<td>0</td>
</tr>
<tr>
<td>Blue area cell count</td>
<td>14677</td>
<td>5939</td>
<td>5900</td>
<td>0</td>
</tr>
<tr>
<td>Terrain model cell size [m]</td>
<td>10</td>
<td>10</td>
<td>10</td>
<td>10</td>
</tr>
<tr>
<td>Total yellow area [m²]</td>
<td>2137800</td>
<td>1201700</td>
<td>580100</td>
<td>0</td>
</tr>
<tr>
<td>Total blue area [m²]</td>
<td>1467700</td>
<td>593900</td>
<td>590000</td>
<td>0</td>
</tr>
<tr>
<td>Total misrepresented flooding extent area [m²]</td>
<td>3605500</td>
<td>1795600</td>
<td>1170100</td>
<td>0</td>
</tr>
<tr>
<td>Total study area [m²]</td>
<td>66570000</td>
<td>66570000</td>
<td>66570000</td>
<td>66570000</td>
</tr>
<tr>
<td>Percentage of misrepresented flooding extent area with respect to the total study area [%]</td>
<td>5.4</td>
<td>2.7</td>
<td>1.8</td>
<td>0</td>
</tr>
<tr>
<td>Percentage of misrepresented potentially hazardous flooding extent area (yellow area) with respect to the total study area [%]</td>
<td>3.2</td>
<td>1.8</td>
<td>0.9</td>
<td>0</td>
</tr>
<tr>
<td>Computation time of the model [hh:mm:ss]</td>
<td>00:00:49</td>
<td>00:13:58</td>
<td>01:52:00</td>
<td>30:50:41</td>
</tr>
</tbody>
</table>

Table 12. Numerical analysis of the flooding extent results

Table 12 provides an overview of the models’ performances by means of a numerical approach on the flooding extent differences between the models. The purpose of adding the 10x10 m model on the table is simply for the sheer comparison of the computation time (obviously it does not involve any misrepresented flooding extent area because the 10x10 m model is set as a reference for a model which can’t utilize the sub-grid geometry).
All in all, the sub-grid models manage to approximate the flooding extent with success in much lesser computation times. Assuming only the potential endangered flooding extent areas are considered for the reasons exposed earlier, the cell size models of 100x100, 50x50 and 25x25 m are capable of attaining results with a 3.2%, 1.8% and 0.9% error in 2266, 133 and 17 faster computation times than the 10 m model respectively. The sub-grid models may estimate with an acceptable precision the extent of the floods in small computation times. This can result in an essential advantage provided the sub-grid models are thought of as an early warning flood prevention system that may predict the flooding extent given forecasted flow discharges of the study area. They could be instrumental to the safe evacuation of populated areas due to the lower computation times.

6.3.2 Maximum water depth

6.3.2.1 Maximum water depth along profile lines

Figure 83, Figure 84 and Figure 85 (see below) highlight how the selected mesh cell sizes represent the maximum water depth for all three profile lines. For the upstream profile line there are large discrepancies between the models for the first 3000 meters of the profile which go as far as showing 3 meter differences in water depth between the 100x100 and 10x10 m models. For the rest of the profile the variability in water depth is severely reduced.
Effect of the sub-grid geometry on two-dimensional river flow models

Ricard González Blanch

Figure 83. Sub-grid comparison for the upstream profile line

Figure 84. Sub-grid comparison for the middle profile line
Contrary to the upstream profile line, the middle profile line depicts far more accurate results regarding maximum water depth. The difference in values between the 100x100 and the 10x10 m simulations can go up to the 20 cm mark in some sections but mostly remain near the 10 cm value. The 50x50 m mesh cell size produces somewhat closer values to the 10x10 m model than the 100x100 m mesh cell size, resulting in a maximum difference of 10 cm and an average of around 5 cm.

Finally, the downstream profile line provides maximum water depths which are too distinct between the models. In this case, the difference is large enough not to consider the 100x100 and 50x50 m models as successful, as opposed to the middle profile line plot.

The observed pattern for these simulations is that the sub-grid geometry appears to have more weight for the middle profile line as the 100x100 and 50x50 m models are able to provide similar results to the ones from the 10x10 m model. This could be attributed to what was discussed in section 4.3 Boundary conditions regarding the behaviour of hydraulic models in areas close to where boundary conditions are applied. Flow is forced to comply with the set boundary conditions causing flow not to have enough time to fully develop in areas which are still affected by the boundary conditions. Due to the fact that in the model both the upstream and downstream profile lines are extremely close to the position of the boundary conditions, it is entirely...
conceivable that the nature of the detected inconsistencies is in fact the adaptation flow areas surrounding the boundary conditions.

6.3.2.2 Maximum water depth difference map

In order to identify the regions of the study area where the 100x100, 50x50 and 25x25 m models succeed in approximating the water depth with the 10x10 m model as a reference, three maps with the absolute difference in maximum water depth between the models with sub-grid geometry and the 10x10 m have been devised.

These computed maps were computed through the Raster Calculator tool in the ArcGis 10.0 environment after having exported from HEC-RAS the maximum depth maps for each simulation. They grant the chance to assess which areas of the study zone may be susceptible to being modelled by larger mesh sizes. Only the absolute difference between models is studied as it is an indisputable illustration of how close the simulations predict maximum water depths.

![Image](image.png)

Figure 86. Absolute difference in maximum water depth between the 10x10 and 100x100 m models
The results obtained are in consonance with what is expected when working with larger and smaller meshes. By looking at Figure 86, Figure 87 and Figure 88, one can see the progression of the difference in water depth for subsequently smaller mesh cell sizes and realize the accuracy of the models increase as the mesh cell sizes decrease because water depth differences become slimmer. This is derived from the fact that the maps with lower mesh cell size as the comparison with the 10 m model appear to have fewer red and yellow areas, which represent high differences, and more blue areas which are synonym of small differences.
Effect of the sub-grid geometry on two-dimensional river flow models

Ricard González Blanch

Effect of the sub-grid geometry on two-dimensional river flow models

Ricard González Blanch

Figure 89. Classification statistics of the water depth difference between the 10x10 m and the 25x25 m, 50x50 m and 100x100 m models respectively (from left to right)

More explicitly, the 25x25 m mesh cell size model is able to achieve very satisfactory results for the floodplains whose areas lie mostly within the 5 centimeter error with some exceptions such as the part closer to the downstream boundary and the obvious red outlier area which will be addressed shortly. The river tends to present higher water depth differences than the floodplains, especially in the reach closer to the river mouth. The 100x100 and 50x50 m mesh cell sizes seem to have a worse performance as the differences with the 10 m model become more apparent, especially for the downstream part of the study area where differences higher than 0.3 m are more frequent. Statistics of the water depth difference for each case are detailed in Figure 89, and they seem to be in line with what was gathered from the visual inspection of the computed maps. The mean difference in water depth between the three different cell sizes and the 10x10 m model increases for larger mesh cell sizes, and so do the maximum difference and standard deviation. Although the mean supposes to be a good indicator, it is not representative of the behaviour of the models in the entire study area. If the analysis of the maps were solely based on the mean values which take values of 6 cm, 12 cm and 20 cm for the 25x25, 50x50 and 100x100 m mesh cell sizes respectively, then, areas with smaller and larger differences than those means would be discriminated. For that reason, in particular, it is still important to spot the areas where differences are high because there the model does not accomplish to approximate the water depth adequately.

Furthermore, the above mentioned water depth figures back up the assumption constructed in the previous section regarding the downstream boundary that stipulated that the existing large water depth variabilities in the models for the downstream profile line were due to the existence of a zone of influence nearby the downstream boundary that hampered the results because of the need to adjust the water depths to the imposed normal depth condition at the coastline.

The mapping of the water depth differences also served to refute the aforementioned boundary condition assumption for the upstream reach. Similarly to the case of the downstream boundary, in the previous section it was speculated that the noticeable inconsistency of water depths along the upstream profile line between the models was due to the profile line being too close to the upstream boundary and being distorted by
the flow hydrograph condition. However, the water depth difference maps reveal that the observed inconsistency was caused by the isolated flooded area that appears in red in all of the maps because the upstream profile line goes across that poorly represented flooded area (see Figure 90) where water depth differences between models are substantial. As a result, the first stations of the upstream profile line plot (Figure 83) showed large water depths variability.

The area represented in Figure 90 clearly stands out from the rest because the differences are much higher. This can also be seen for the rest of variables studied. Further study of this specific area is required to determine the odd results obtained, although it is suspected to be caused by a structure or particular topography that sub-grid model is not able to capture while the 10x10 m mesh cell size model is due to finer cells. It would be advisable to manually edit the computational mesh of the 100x100, 50x50 and 25x25 m models in that area to have fine cells of 10x10m able to capture the topography and yield proper results.

![Figure 90. Upstream profile line (in pink) intersects the controversial flooded area (surrounded by a yellow rectangle)](image)

Anyhow, the level of accuracy in the water depth analysis is subject to what the aim of the study requires and to the time available for running the simulation. In case the interest of the study is to perform a study of the damages in the study area caused by a 500 year return period flood, then it would be advisable to use the 25x25 m mesh cell size model as the difference in maximum water depth are negligible in comparison to the computation time saved. This would apply to the floodplains barring unusual behavioural flooded areas and to a lesser degree those close to the downstream boundary.
6.3.3 Maximum flow velocity

Figure 91. Absolute difference in maximum flow velocity between the 10x10 and 100x100 m models

Figure 92. Absolute difference in maximum flow velocity between the 10x10 and 50x50 m models

Figure 93. Absolute difference in maximum flow velocity between the 10x10 and 25x25 m models
In the same way it was carried out for the water depth differences, the maps shown in Figure 91, Figure 92 and Figure 93 were processed by overlapping the sub-grid models and the 10x10 m mesh cell size model maximum flow velocity maps obtained in 6.2 Maximum water depth, maximum flow velocity and arrival time map results by means of the Raster Calculator tool in ArcGis 10.0.

Similarly to the water depth analysis and as proof of what is expected from different mesh cell sizes, the smaller the mesh cell size, the less differences with the 10x10 m model. The 25x25 m mesh cell size model, the most accurate of all three, goes as far as obtaining a mean difference with the 10x10 m mesh cell size model of only 0.07 m/s. As it was found for the water depth case, the models reveal higher discrepancies near the lower boundary which become more obvious for the 100x100 m and 50x50 m mesh cell size models.

Furthermore, all of the sub-grid models concur that the highest maximum flow velocity differences are found within the river course (blue areas). The rest of the blue profiles and other high difference areas found in the maps actually correspond in its vast majority to both highways and roads separating irrigation parcels. Also, the original Ter-Daró canal and other irrigation channels that were manually added in section 5.1 Importing terrain model and computational mesh generation by ‘breaklines’ present high flow velocity differences too.

On the one hand, this phenomenon suggests that the sub-grid models are not able to capture the flow velocity in parts where the water flow is canalised by levees of the river and canals or by shoulders or limiting obstructions barriers located in the margins of highways and roads. On the other hand, the models are capable of predicting the floodplain flow velocity development successfully where differences between 0 and 0.05 m/s prevail for the 25x25 m model. Altogether, the 100x100 m model produces results with too large of a difference even for the floodplains, whereas the 50x50 and 25x25 m mesh cell size models depict the floodplains flow velocity quite precisely. Notwithstanding the efficiency of the 50x50 m mesh cell size model, it is recommended to employ the 25x25 m mesh cell size model for the maximum flow velocity analysis at it produces more robust results while still reducing the 10x10 m mesh cell size model computation time by a lot (29 hour computation time difference).

6.3.4 Arrival time

This section deals with the prediction of the time it takes since the start of the simulation for the flood wave to reach locations within the study area. This parameter
is a deciding factor when weighing a flood since it can be used to prevent future damages.

The time differences, expressed in hours, have been divided into categories in order to ease the interpretation of the processed maps by means of the Reclassify tool from ArcGis 10.0, similarly to what was done for the flooding extent maps.

Figure 94. Flooding extent difference between the 10x10 m and the 100x100 m models

Figure 95. Flooding extent difference between the 10x10 m and the 50x50 m models
At first glance the 25x25 m mesh cell size model in particular seems to predict the arrival time of the flood wave quite flawlessly as yellow and orange areas, representing a 0 hour and -1 hour difference, predominate in the maps. As it has been the tendency for past variable analysis, smaller mesh cell sizes offer better results for the time arrival as well, though it is expected. The highest time differences are found in locations that are furthest away from the river course and other high differences may be spotted on the floodplains in the downstream reach of the river.

The idea behind safely and unsafely misrepresented areas debated for the flooding extent analysis may be applied for the time arrival scenario. The computed difference maps shown in Figure 94, Figure 95 and Figure 96 were obtained by subtracting the 10x10 m mesh cell size model arrival time map from the sub-grid models arrival time maps. Therefore, if a cell takes a negative value in the computed maps presented above, it actually means the sub-grid model predicts the flood to arrive at that cell earlier than the 10 m mesh cell size model. As a result, it is reasonable to consider these misrepresentations of the arrival time as safe because in a real life application prevention measures should already be put up for earlier times, thus not taking by surprise the residents from the study area. On the contrary, if a cell has a positive value, then that misrepresentation could be assumed to be hazardous as the sub-grid predicts the flood wave to arrive later than what the 10x10 m mesh cell size model says.
Figure 97. Cell count for time arrival difference categories for the 100x100 m mesh cell size model

Figure 98. Cell count for time arrival difference categories for the 50x50 m mesh cell size model

Figure 99. Cell count for time arrival difference categories for the 25x25 m mesh cell size model
Effect of the sub-grid geometry on two-dimensional river flow models

Ricard González Blanch

Table 13. Numerical analysis of the time arrival results

<table>
<thead>
<tr>
<th>Arrival time comparison with the 10x10 model</th>
<th>100x100</th>
<th>50x50</th>
<th>25x25</th>
<th>10x10</th>
</tr>
</thead>
<tbody>
<tr>
<td>Positive value cell count (hazardous)</td>
<td>49413</td>
<td>29454</td>
<td>20509</td>
<td>0</td>
</tr>
<tr>
<td>Negative value cell count</td>
<td>219460</td>
<td>176023</td>
<td>133500</td>
<td>0</td>
</tr>
<tr>
<td>Terrain model cell size [m]</td>
<td>10</td>
<td>10</td>
<td>10</td>
<td>10</td>
</tr>
<tr>
<td>Total positive arrival time area [m^2]</td>
<td>4941300</td>
<td>2945400</td>
<td>2050900</td>
<td>0</td>
</tr>
<tr>
<td>Total negative arrival time area [m^2]</td>
<td>21946000</td>
<td>17602300</td>
<td>13350000</td>
<td>0</td>
</tr>
<tr>
<td>Total misrepresented arrival time area [m^2]</td>
<td>26887300</td>
<td>20547700</td>
<td>15400900</td>
<td>0</td>
</tr>
<tr>
<td>Total study area [m^2]</td>
<td>66570000</td>
<td>66570000</td>
<td>66570000</td>
<td>66570000</td>
</tr>
<tr>
<td>Percentage of misrepresented arrival time area with respect to the total study area [%]</td>
<td>40.4</td>
<td>30.9</td>
<td>23.1</td>
<td>0</td>
</tr>
<tr>
<td>Percentage of hazardous arrival time area with respect to the total study area [%]</td>
<td>7.4</td>
<td>4.4</td>
<td>3.1</td>
<td>0</td>
</tr>
<tr>
<td>Computation time of the model [hh:mm:ss]</td>
<td>00:00:49</td>
<td>00:13:58</td>
<td>01:52:00</td>
<td>30:50:41</td>
</tr>
</tbody>
</table>

The percentages of misrepresented arrival time area are not satisfactory at all since they reveal that 40%, 31% and 23% of the cells for the 100x100, 50x50 and 25x25 m mesh cell size model respectively appear to show some sort of error in predicting the arrival time, although not all of them of the same severity. Thus, the sub-grid models seem not to be effective for the modelling of the arrival time, contrary to what they showed for the flooding extent.

The only redeeming feature attainable from Table 13 is the fact that the percentages of hazardous arrival time area lie within respectable boundaries even though they are not nearly as promising as the ones from the flooding extent analysis. In order to contextualize the results above, a simple contrast between the arrival time and the flooding extent analysis is pointed out: the 25x25 m mesh cell size model is required to obtain a 3% of hazardous arrival time area whereas for flooding extent the 100 m mesh cell size model achieves the same percentage of hazardous flooding extent area in approximately 120 faster times.
7. Conclusions

7.1 HEC-RAS 5.0 2D module
After an extensive period of working with the HEC-RAS 5.0 2D module, it can be established that the software appears to be a very robust tool for 2D modelling that is definitely on par with the most advanced 2D modelling programs. It allows performing large scale computations while keeping a simple way to set-up the corresponding model. Its uniqueness comes from a user-friendly interface coupled with the availability to check the mapping results of the simulations dynamically through RAS Mapper. However, most of the post-processing was carried out in ArcGis 10.0 as HEC-RAS 5.0 lacks the tools to do so.

Furthermore, the option to perform a quasi-2D simulation through the 1D/2D feature was considered. The idea behind it was to model the river as a 1D element and connect it to the surrounding 2D floodplains in order to reduce computation costs. It was finally disregarded as the lateral hydraulic structure that was used to connect both elements did not function properly because some floodplains did not show flooding despite the water depth in the corresponding reach of the river clearly overflowing the main channel.

7.2 Development of the models
Different models are necessary to assess the impact of the sub-grid representation. For that reason, four geometries with computational mesh of cell sizes 100x100, 50x50, 25x25 and 10x10 m are developed, with the 10x10 m model being considered as a model without sub-grid geometry as it has the same mesh cell size as the resolution of the terrain model used. For the design of the computational mesh it is found out that high elevation terrain can be left out of the mesh boundaries since water depth is not prone to reaching those parts. As such, the area covered by the computational mesh is reduced, therefore lower computation times are possible. Another important step towards designing the computational meshes is the use of the ‘breakline’ tool to model levees and limits of rivers and irrigation channels, making cell faces align along the crest of high ground features in the topography that completely or temporarily act as a barrier to water flow with it. This is carried out so that cell faces are able to capture the details of these barriers and have them taken into account by the sub-grid representation, which result essential to prevent the so-called leakage effect in 2D flow modelling.

7.3 Simulation of the models
With respect to the simulation of the models, the single most determining factor is the computation interval. A lot of time and resources are spent setting up an appropriate
computation interval for each model mesh cell size. On the one hand, the computation interval for each model has to be the highest possible to reduce the computation time while still providing converging results with lower computation time-steps. The calibration of the computation interval is based on a downstream profile close to where the downstream boundary condition is applied as results proved to differ more between computation intervals there than in more upstream reaches. The corresponding computation intervals found for the models were: 1 minute, 12 seconds, 10 seconds and 5 seconds for the 100x100, 50x50, 25x25 and 10x10 m models respectively.

On the other hand, the stability of each simulation for each model with its corresponding appropriate computation interval mentioned above must be assured. Although the program does not give any errors in the computation messages window and the wave propagation through time visualized in RAS Mapper does not seem to produce instabilities for any of the simulations with the appropriate computation interval, the verification of the Courant condition still remains to be studied. The condition must be verified for all cells at any given time-step which is not possible to do as the HEC-RAS 5.0 2D module does not currently have an option to export the water depth and flow velocity maps at all time-steps. In view of this, an upper bound of the Courant number is computed with the highest flow velocity and highest water depth for each cell throughout the 30h simulation, which provides the highest possible Courant number for a cell. According to the Courant number recommendations for the Diffusion-Wave approximation, only the 50x50 m model presents all cells with an upper bound Courant number smaller than 5 (maximum allowed recommended number), thus the stability is more than assured. The 100x100 and 10x10 m models show very small cells in the river with greater upper bound Courant numbers than 5. Additionally, the 25x25 m model also has some cells in the river with greater upper bound Courant number than 5, although to a lesser extent than the 100x100 and 10x10 m models. Considering that the computations are carried out for an upper bound of the Courant number, where it is assumed maximum water depths and flow velocities happen at the same time step (which is not necessarily true), and that it happens for cells within the river where velocities tend to be higher, it can be concluded that the 100x100, 25x25 and 10x10 m simulations with the corresponding computation intervals are also stable.

The computation time of each model with the aforementioned computation intervals are approximately: 1 minute, 14 minutes, 2 hours and 31 hours for the 100x100, 50x50, 25x25 and 10x10 m models respectively. This is already an indicator of how the model without sub-grid representation (10x10 m) takes longer to perform the simulation. However, just from considering the computation times it is not possible to evaluate the sub-grid model performance as accuracy of the results are yet to be weighed in.
7.4 Sub-grid model results

The methodology followed to assess the accuracy of the results obtained by using the sub-grid model is based on overlapping the 100x100, 50x50 and 25x25 m models, which fully use the sub-grid representation, with the 10x10 m model, which neglects the sub-grid representation, for each variable to find the differences between them in the entire domain of the study area. Comparisons are taking into consideration the 10x10 m model provides more accurate results due to a lower mesh cell size and computation interval.

Some general conclusions may be drawn out that apply to all four variables studied. First of all, the larger the mesh models, the worse performance in terms of differences with the 10x10 m mesh cell size model. This is expected since they capture more terrain for each computational cell. Secondly, in all of the model cases the most noticeable differences are found near the downstream boundary condition. This is due to the fact that boundary conditions tend to alter the behaviour of flow in nearby zones as the flow is being obliged to adjust to the condition. Lastly, there appears to be an area that clearly stands out from the rest, in which the differences are much higher. Further study of this specific area is required to determine the odd results obtained, although it is suspected to be caused by a structure or particular topography that the sub-grid model is not able to capture while the 10x10 m mesh cell size model is due to finer cells. It would be advisable to manually edit the computational mesh of the 100x100, 50x50 and 25x25 m models in that area to have fine cells of 10x10 m able to capture the topography and yield proper results.

The percentages of total misrepresented flooding extent area with respect to the total study area are: 5.4%, 2.7% and 1.8% for the 100x100, 50x50 and 25x25 m models respectively. Moreover, the misrepresented areas can be divided in two types: areas that appeared as flooded in the sub-grid model simulations and not in the 10x10 m model, and areas that do not get flooded in the sub-grid model simulations but do in the 10x10 m mesh cell size model. If the simulation were to be applied in a real life scenario, the first type of misrepresented area corresponds to a safe approach since the flooding extent is overestimated while the second one is potentially hazardous as the sub-grid models predict areas to be safe from flooding when in reality they are not. The percentages of potentially hazardous misrepresented flooding extent area with respect to the total study area are: 3.2%, 1.8% and 0.9% for the 100x100, 50x50 and 25x25 m models respectively. The 25x25 m model with sub-grid representation is highly recommended as it can predict the flooding extent with only a 0.9% error in just 2 hours which could be a pretty strong asset in a real case flooding.

The differences in maximum water depth turned out to be pretty small. The mean value of the differences was 6 cm, 12 cm and 20 cm for the 25x25, 50x50 and 100x100
m respectively. In addition, a vast majority of the floodplains lie within the 0 and 5 mm for the 25x25 m model, which is exceptional. The maximum water depth difference in the floodplains is found to be between 10 and 20 cm. All in all, the differences of the 25x25 and 50x50 models with sub-grid representation are acceptable enough in case a flooding damage analysis by water depth categories were to be performed since the differences are not high enough to cause a major alteration in the category distribution. Besides, the computation times for the 25x25 and 50x50 m models are 29 hours and 31 hours faster respectively than that of the 10x10 model.

The maximum flow velocity differences are slightly larger in magnitude than for water depth. The mean difference for the 25x25 m model is 0.07 m/s and even lower values than that for the floodplains. The largest differences, higher than 0.3 m/s, are located in the Ter, the Ter-Daró channel and other irrigation channels. This is probably due to fact that in spite of the use of ‘breaklines’, the sub-grid representation still does not manage to fully capture the topography since the mesh cell sizes are too big. Similarly to the case of the particular area explained before, in this case it is recommended to edit these water channels manually and put finer cells. This fact also opens up the possibility of exploring the 1D/2D capabilities for the flow velocity analysis, with the river being modelled as a 1D element and the floodplains as 2D elements. This could solve the found discrepancies in the maximum flow velocity for the river and channels while still modelling the floodplains in 2D which appears to work quite well.

Finally, for the arrival time analysis the same approach used for the flooding extent is considered here. The potentially hazardous areas in this case are those that the models with sub-grid representation predict water to arrive at a later time than the 10x10 m model. The percentages of potentially hazardous areas with respect to the total study area are: 7.4%, 4.4% and 3.1% for the 100x100, 50x50 and 25x25 m models respectively which are about two or three times higher than the ones for the flooding extent. The percentages of total misrepresented areas are: 40.4%, 30.9% and 23.1% for the 100x100, 50x50 and 25x25 models respectively, which are totally not considerable errors. Thus, it can be concluded that the sub-grid model is not appropriate for the arrival time modelling with the sizes of the meshes used.

In conclusion, the sub-grid model yields effective flooding extent, maximum water depth and flow velocity results for a mesh cell size of 25x25 m, whereas the 50x50 m model is only recommended for the maximum water depth analysis. These results are obtained in much lower computation times than the 10x10 m model. Specifically, the 25x25 and 50x50 m models do it 29 hours and 31 hours faster respectively. The arrival time modelling does not seem to be well predicted for any of the mesh sizes studied.
References


Appendix

Table of equivalences between the land cover classification (MCSC v.5) and Manning’s n coefficients. Source: PEF from the lower Ter basins, ACA.

<table>
<thead>
<tr>
<th>ID</th>
<th>Land cover classification (MCSC)</th>
<th>Manning's n</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Pineda de pi pinyer (&gt;= 20%cc)</td>
<td>0.12</td>
</tr>
<tr>
<td>2</td>
<td>Pineda de pinastre (&gt;= 20%cc)</td>
<td>0.12</td>
</tr>
<tr>
<td>3</td>
<td>Pineda de pi blanc (&gt;= 20%cc)</td>
<td>0.12</td>
</tr>
<tr>
<td>9</td>
<td>Alzinar (&gt;= 20%cc)</td>
<td>0.12</td>
</tr>
<tr>
<td>10</td>
<td>Sureda (&gt;= 20%cc)</td>
<td>0.12</td>
</tr>
<tr>
<td>19</td>
<td>Roureda de roure martinec (&gt;= 20%cc)</td>
<td>0.12</td>
</tr>
<tr>
<td>24</td>
<td>Altres caducifolis (&gt;= 20%cc)</td>
<td>0.12</td>
</tr>
<tr>
<td>25</td>
<td>Boscos caducifolis de ribera (&gt;= 20%cc)</td>
<td>0.12</td>
</tr>
<tr>
<td>26</td>
<td>Pineda de pi pinyer (5-20%cc)</td>
<td>0.12</td>
</tr>
<tr>
<td>27</td>
<td>Pineda de pinastre (5-20%cc)</td>
<td>0.12</td>
</tr>
<tr>
<td>28</td>
<td>Pineda de pi blanc (5-20%cc)</td>
<td>0.12</td>
</tr>
<tr>
<td>34</td>
<td>Alzinar (5-20%cc)</td>
<td>0.12</td>
</tr>
<tr>
<td>35</td>
<td>Sureda (5-20%cc)</td>
<td>0.12</td>
</tr>
<tr>
<td>44</td>
<td>Roureda de roure martinec (5-20%cc)</td>
<td>0.12</td>
</tr>
<tr>
<td>49</td>
<td>Altres caducifolis (5-20%cc)</td>
<td>0.12</td>
</tr>
<tr>
<td>50</td>
<td>Boscos caducifolis de ribera (5-20%cc)</td>
<td>0.12</td>
</tr>
<tr>
<td>51</td>
<td>Franja de protecció de pi pinyer</td>
<td>0.08</td>
</tr>
<tr>
<td>52</td>
<td>Franja de protecció de pinastre</td>
<td>0.08</td>
</tr>
<tr>
<td>53</td>
<td>Franja de protecció de pi blanc</td>
<td>0.08</td>
</tr>
<tr>
<td>56</td>
<td>Franja de protecció d'alzina</td>
<td>0.08</td>
</tr>
<tr>
<td>57</td>
<td>Franja de protecció d'alzina surera</td>
<td>0.08</td>
</tr>
<tr>
<td>66</td>
<td>Plantacions de pi pinyer</td>
<td>0.08</td>
</tr>
<tr>
<td>67</td>
<td>Plantacions de pinastre</td>
<td>0.075</td>
</tr>
<tr>
<td>68</td>
<td>Plantacions de pi blanc</td>
<td>0.075</td>
</tr>
<tr>
<td>73</td>
<td>Vivers forestals</td>
<td>0.08</td>
</tr>
<tr>
<td>74</td>
<td>Plantacions de coníferes no autòctones</td>
<td>0.08</td>
</tr>
<tr>
<td>78</td>
<td>Plantacions d'eucaliptus</td>
<td>0.075</td>
</tr>
<tr>
<td>79</td>
<td>Plantacions de pollancrecs</td>
<td>0.08</td>
</tr>
<tr>
<td>80</td>
<td>Plantacions de plàtans</td>
<td>0.08</td>
</tr>
<tr>
<td>81</td>
<td>Regeneració de pi blanc</td>
<td>0.09</td>
</tr>
<tr>
<td>92</td>
<td>Matollars</td>
<td>0.06</td>
</tr>
<tr>
<td>93</td>
<td>Matollars en línies elèctriques</td>
<td>0.05</td>
</tr>
<tr>
<td>94</td>
<td>Franja de protecció de matollars</td>
<td>0.065</td>
</tr>
<tr>
<td>96</td>
<td>Matollars procedents de tallades arreu</td>
<td>0.06</td>
</tr>
<tr>
<td>97</td>
<td>Matollars de formacions de ribera</td>
<td>0.07</td>
</tr>
<tr>
<td>98</td>
<td>Canyars</td>
<td>0.08</td>
</tr>
<tr>
<td>99</td>
<td>Vegetació d'aiguamolls litorals</td>
<td>0.075</td>
</tr>
<tr>
<td>100</td>
<td>Vegetació d'aiguamolls continental</td>
<td>0.075</td>
</tr>
<tr>
<td>103</td>
<td>Prats i herbassars</td>
<td>0.055</td>
</tr>
<tr>
<td>Número</td>
<td>Descripció</td>
<td>Percentatge</td>
</tr>
<tr>
<td>---------</td>
<td>----------------------------------------------------------------------------</td>
<td>-------------</td>
</tr>
<tr>
<td>104</td>
<td>Prats i herbassars en lílines elèctriques</td>
<td>0.05</td>
</tr>
<tr>
<td>105</td>
<td>Franja de protecció de prats i herbassars</td>
<td>0.055</td>
</tr>
<tr>
<td>109</td>
<td>Prats i herbassars procedents de tallades arreu</td>
<td>0.05</td>
</tr>
<tr>
<td>111</td>
<td>Vegetació arbustiva de dunes i sorrals</td>
<td>0.05</td>
</tr>
<tr>
<td>112</td>
<td>Vegetació herbàcia de dunes i sorrals</td>
<td>0.04</td>
</tr>
<tr>
<td>113</td>
<td>Zones cremades</td>
<td>0.045</td>
</tr>
<tr>
<td>114</td>
<td>Penya-segats marins</td>
<td>0.05</td>
</tr>
<tr>
<td>115</td>
<td>Roquissars</td>
<td>0.05</td>
</tr>
<tr>
<td>116</td>
<td>Tarteres</td>
<td>0.05</td>
</tr>
<tr>
<td>117</td>
<td>Lleres naturals</td>
<td>0.035</td>
</tr>
<tr>
<td>119</td>
<td>Sòl erosionat per agent natural</td>
<td>0.035</td>
</tr>
<tr>
<td>120</td>
<td>Sòl nu per acció antròpica</td>
<td>0.03</td>
</tr>
<tr>
<td>121</td>
<td>Sòl nu en lílines elèctriques</td>
<td>0.03</td>
</tr>
<tr>
<td>122</td>
<td>Platges</td>
<td>0.03</td>
</tr>
<tr>
<td>124</td>
<td>Llacunes litorals</td>
<td>0.025</td>
</tr>
<tr>
<td>125</td>
<td>Llacunes litorals</td>
<td>0.025</td>
</tr>
<tr>
<td>126</td>
<td>Rius</td>
<td>0.035</td>
</tr>
<tr>
<td>130</td>
<td>fruiters no cítrics</td>
<td>0.055</td>
</tr>
<tr>
<td>131</td>
<td>fruiters no cítrics en regadiu</td>
<td>0.055</td>
</tr>
<tr>
<td>132</td>
<td>Vivers agrícoles</td>
<td>0.07</td>
</tr>
<tr>
<td>133</td>
<td>Vinyes</td>
<td>0.055</td>
</tr>
<tr>
<td>134</td>
<td>Oliverars</td>
<td>0.065</td>
</tr>
<tr>
<td>135</td>
<td>Oliverars en regadiu</td>
<td>0.065</td>
</tr>
<tr>
<td>138</td>
<td>Arrossars</td>
<td>0.03</td>
</tr>
<tr>
<td>139</td>
<td>Altres conreus herbacís</td>
<td>0.04</td>
</tr>
<tr>
<td>140</td>
<td>Altres conreus herbacís en regadiu</td>
<td>0.045</td>
</tr>
<tr>
<td>142</td>
<td>Conreus en transformació</td>
<td>0.05</td>
</tr>
<tr>
<td>143</td>
<td>Rompudes agrícoles</td>
<td>0.05</td>
</tr>
<tr>
<td>144</td>
<td>Hivernacles</td>
<td>0.08</td>
</tr>
<tr>
<td>153</td>
<td>Altres conreus herbacís en bancals</td>
<td>0.05</td>
</tr>
<tr>
<td>155</td>
<td>conreus abandonats - boscos</td>
<td>0.075</td>
</tr>
<tr>
<td>156</td>
<td>conreus abandonats - matollars</td>
<td>0.065</td>
</tr>
<tr>
<td>157</td>
<td>conreus abandonats - prats en zones forestals</td>
<td>0.065</td>
</tr>
<tr>
<td>161</td>
<td>fruiters no cítrics abandonats - prats en zones agrícoles</td>
<td>0.055</td>
</tr>
<tr>
<td>162</td>
<td>fruiters no cítrics abandonats regadiu no regat</td>
<td>0.065</td>
</tr>
<tr>
<td>165</td>
<td>vineyes abandonades - prats en zones agrícoles</td>
<td>0.065</td>
</tr>
<tr>
<td>166</td>
<td>Oliverars abandonats - prats en zones agrícoles</td>
<td>0.07</td>
</tr>
<tr>
<td>170</td>
<td>Altres conreus herbacís abandonats</td>
<td>0.065</td>
</tr>
<tr>
<td>171</td>
<td>Altres conreus herbacís abandonats regadiu no regat</td>
<td>0.065</td>
</tr>
<tr>
<td>181</td>
<td>horta familiar</td>
<td>0.065</td>
</tr>
<tr>
<td>185</td>
<td>canals artificials</td>
<td>0.02</td>
</tr>
<tr>
<td>186</td>
<td>basses agrícoles</td>
<td>0.04</td>
</tr>
<tr>
<td>187</td>
<td>urbanitzacions</td>
<td>0.1</td>
</tr>
<tr>
<td></td>
<td>Category</td>
<td>Probability</td>
</tr>
<tr>
<td>---</td>
<td>---------------------------------------------</td>
<td>-------------</td>
</tr>
<tr>
<td>188</td>
<td>Centre urbà</td>
<td>0.15</td>
</tr>
<tr>
<td>189</td>
<td>Eixample</td>
<td>0.15</td>
</tr>
<tr>
<td>190</td>
<td>Habitatges unifamiliars</td>
<td>0.1</td>
</tr>
<tr>
<td>191</td>
<td>Colònies i nucli aïllats</td>
<td>0.1</td>
</tr>
<tr>
<td>192</td>
<td>Cases aïllades</td>
<td>0.1</td>
</tr>
<tr>
<td>193</td>
<td>Polígon industrial ordenat</td>
<td>0.1</td>
</tr>
<tr>
<td>194</td>
<td>Polígon industrial sense ordenar</td>
<td>0.1</td>
</tr>
<tr>
<td>195</td>
<td>Indústries aïllades</td>
<td>0.1</td>
</tr>
<tr>
<td>197</td>
<td>Complexos comercials i d'oficines</td>
<td>0.1</td>
</tr>
<tr>
<td>198</td>
<td>Vivers</td>
<td>0.065</td>
</tr>
<tr>
<td>199</td>
<td>Complexos hotelers</td>
<td>0.1</td>
</tr>
<tr>
<td>200</td>
<td>Altres construccions</td>
<td>0.1</td>
</tr>
<tr>
<td>201</td>
<td>Naus d'ús agrícola</td>
<td>0.1</td>
</tr>
<tr>
<td>202</td>
<td>Granges</td>
<td>0.1</td>
</tr>
<tr>
<td>204</td>
<td>Cementiris</td>
<td>0.15</td>
</tr>
<tr>
<td>207</td>
<td>Parcs urbans</td>
<td>0.08</td>
</tr>
<tr>
<td>208</td>
<td>Basses urbanes</td>
<td>0.04</td>
</tr>
<tr>
<td>209</td>
<td>Autopistes i autovies</td>
<td>0.025</td>
</tr>
<tr>
<td>210</td>
<td>Carreteres</td>
<td>0.025</td>
</tr>
<tr>
<td>211</td>
<td>Aeroports</td>
<td>0.01</td>
</tr>
<tr>
<td>213</td>
<td>Zones portuàries</td>
<td>0.025</td>
</tr>
<tr>
<td>214</td>
<td>Zones verdes viàries</td>
<td>0.03</td>
</tr>
<tr>
<td>216</td>
<td>Àrees de servei en xarxa viària</td>
<td>0.05</td>
</tr>
<tr>
<td>218</td>
<td>Zones d'esport</td>
<td>0.05</td>
</tr>
<tr>
<td>219</td>
<td>Parcs recreatius</td>
<td>0.08</td>
</tr>
<tr>
<td>220</td>
<td>Càmpings</td>
<td>0.08</td>
</tr>
<tr>
<td>221</td>
<td>Camps de golf</td>
<td>0.04</td>
</tr>
<tr>
<td>222</td>
<td>Complexos administratius</td>
<td>0.1</td>
</tr>
<tr>
<td>224</td>
<td>Equipaments educatius</td>
<td>0.1</td>
</tr>
<tr>
<td>226</td>
<td>Centres religiosos</td>
<td>0.1</td>
</tr>
<tr>
<td>227</td>
<td>Centres culturals</td>
<td>0.1</td>
</tr>
<tr>
<td>232</td>
<td>Infraestructures elèctriques</td>
<td>0.1</td>
</tr>
<tr>
<td>233</td>
<td>Depuradores i potabilitzadores</td>
<td>0.1</td>
</tr>
<tr>
<td>236</td>
<td>Zones d'extracció minera</td>
<td>0.06</td>
</tr>
<tr>
<td>237</td>
<td>Abocadors</td>
<td>0.08</td>
</tr>
<tr>
<td>238</td>
<td>Plantes de tractament</td>
<td>0.1</td>
</tr>
<tr>
<td>239</td>
<td>Sòls nus urbanos no edificats</td>
<td>0.07</td>
</tr>
<tr>
<td>240</td>
<td>Zones urbanes en construcció</td>
<td>0.12</td>
</tr>
<tr>
<td>241</td>
<td>Moviments de terres</td>
<td>0.05</td>
</tr>
</tbody>
</table>