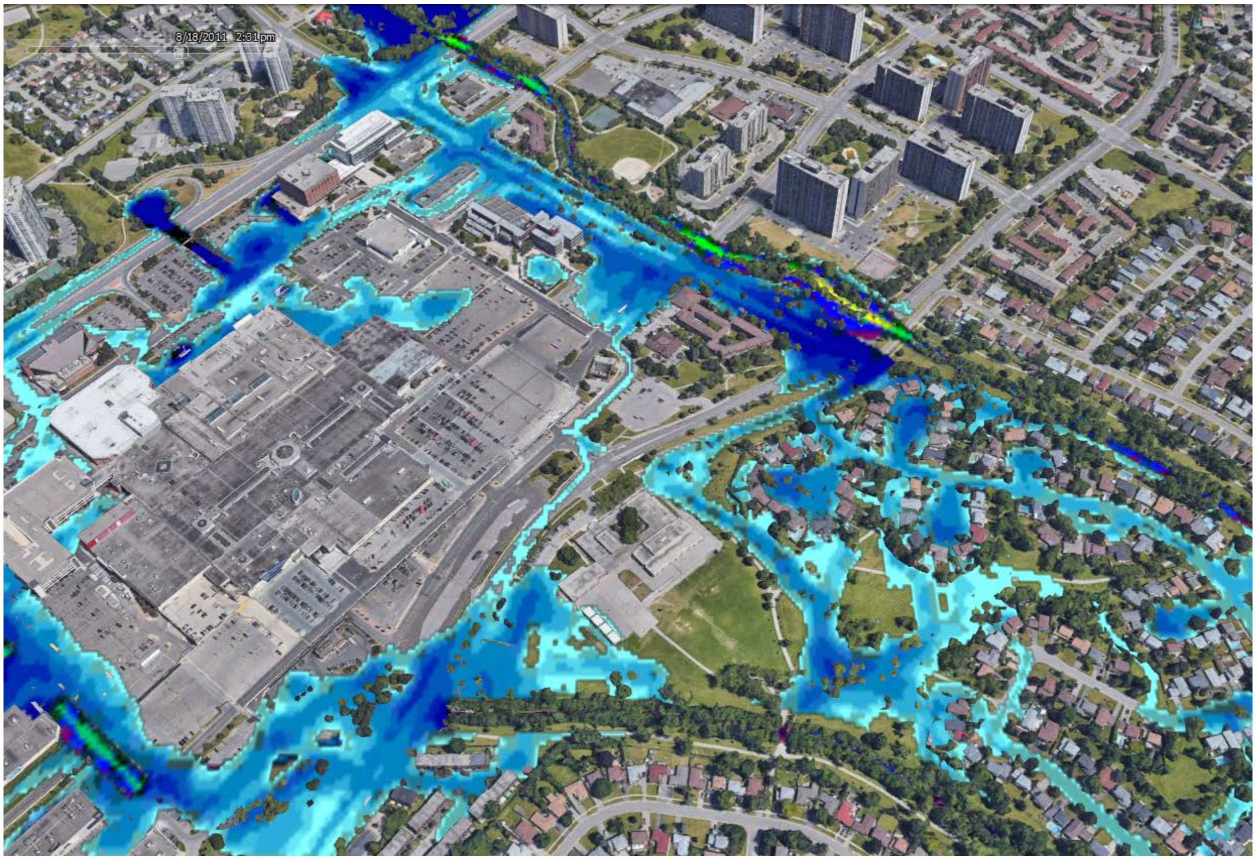
Technical Guideline:

MIKE FLOOD Modelling Approaches

for Floodplain Mapping



Prepared for:

Toronto and Region Conservation Authority
101 Exchange Avenue
Vaughan, ON L4K 5R6

Prepared by:

DHI Water and Environment Inc.
336 Eagle Street North – Unit 1A2
Cambridge, ON N3H 1N2

DHI Project #: 42801316

April 2018

336 Eagle Street North - Unit 1A2
Cambridge, ON N3H 1C2

Tel: (519) 650-4545
Fax: (215) 504-8498

e-mail: pad@dhigroup.com
Web: www.dhigroup.com

MIKE FLOOD Modelling Approaches for Floodplain Mapping

Client	Client's representative
Toronto and Region Conservation Authority	Mr. Nick Lorrain

Project	Project No
MIKE FLOOD Modelling Approaches for Floodplain Mapping	42801852

Author	Date
Patrick Delaney	April 27, 2018
	Approved by: Patrick Delaney

Rev	Description	By	Checked	Approved	Date
1	Final Report	PAD	PAD	PAD	04/27/18

Key words	Classification
	☐ Open ☐ Internal ☒ Proprietary

Distribution		No of copies
Client	Mr. Nick Lorrain	1 Copy (PDF)
DHI	Patrick Delaney	

Table of Contents

List of Figures

List of Tables

1 Introduction

Two-Dimensional (2D) flood modelling and mapping is becoming a more widely used approach for evaluating the impacts of overland flooding in urban environments, and for providing enhanced flood characterization for high risk flood prone areas. Applying the standard One-Dimensional (1D) modelling approach to delineate flood hazards within an urban environment is problematic due to the presence of engineered drainage systems and hydraulic controls such as buildings and structures which dictate the movement of water through an area. Further the TRCA has a number of high risk flood vulnerable areas which require accurate and reliable hydraulic modelling results, at a lot scale, to provide affected municipalities and land owners, appropriate guidance for emergency management, land use, and infrastructure planning.

In order to take advantage of the benefits of 2D flood modelling and the availability of enhanced data inputs like LiDAR, Toronto and Region Conservation Authority has selected DHI's MIKE FLOOD modelling platform as the preferred modelling tool for cases where a 2D modelling approach is required to accurately characterize flooding and evaluate the effectiveness of potential flood mitigation measures.

MIKE FLOOD is a modelling tool that provides a number of different options for approaching a flood modelling study, including:

- 1D riverine modelling using MIKE HYDRO River
- 2D overland flow modelling using MIKE 21 HD Classic
- 2D overland flow modelling using MIKE 21 Flexible Mesh
- Coupled 1D-2D riverine and overland flow modelling using MIKE HYDRO River and MIKE 21 HD Classic
- Couple 1D-2D riverine and overland flow modelling using MIKE HYDRO River and MIKE 21 Flexible Mesh

Given the number of potential approaches to flood modelling study, the purpose of this document is to provide a guideline to assist in the selection of the most suitable approach to the flood modelling study as well as providing some guidelines on how to best utilized MIKE FLOOD to develop the models.

2 Hydraulic Modelling Approaches

The selection of the most appropriate modelling approach for a flood modelling and mapping project is dependent on many factors including the objectives of the study, the size of the study area, the expected flow conditions and extent of flooding, the number and types of hydraulic structures, and the skills and competencies of the professionals undertaking the modelling work. The purpose of this section is to facilitate a decision-making process that considers the different modelling approaches that are available and the advantages and disadvantages of each.

Although the decision-making process is often going to be iterative and will not always follow the same sequence of steps, the considerations described in the following sections are presented in the order in which they will typically be considered.

2.1 Dimensions (1D, 2D or Coupled 1D-2D)

A 2D flood modelling and mapping study is typically initiated after a 1D modelling study has been completed and has identified important short-comings in the ability of the model to accurately characterize flooding outside of the main river channel. However, for the purposes of this document the model selection will include considerations for 1D modelling because it provides an important frame of reference.

When a flood modelling and mapping project is initially conceived it is important to have a good understanding of the river system and where there is likely to be flooding such that an informed decision can be made regarding the suitability of a 1D, 2D or coupled 1D-2D flood modelling approach. A description of these options together with the advantages and disadvantages of each is provided in the following sections.

For the purposes of this document, the approaches are discussed in terms of the conceptual approach to the representation of flow and water levels in the river system and the floodplain rather than the equations associated with the 1D and 2D numerical solutions.

2.1.1 1D River Modelling

1D-steady-state river modelling has been the standard approach for flood modelling and mapping in Ontario since the 1980's. This type of model is generally easy to set up and quick to run so it tends to be a relatively inexpensive approach for flood modelling and mapping. In addition, since 2D modelling is a relatively new approach for modelling of river systems, many North American flood modelling and mapping standards and guidelines appear to have been developed with the base assumption that the modelling will be performed using a 1D-steady-state flow model (e.g. Technical Guide – River and Stream Systems: Flooding Hazard Limit, MNR 2002; Technical Manual Flood Hazard Area Control Act Rules, NJDEP 2018).

A 1D modelling approach makes the basic assumption that flow and water levels in the river channel can be assumed to be accurately represented as a 1D system where the direction of flow is only in the downstream direction and where the water level across the river channel is constant. In cases where the flow in the river is expected to overtop the banks of the main channel, the model cross-section width is usually expanded to include the entire river valley corridor / floodplain. Depending on the size and topographic complexity of the floodplain, the assumptions of 1D flow may still be valid, particularly for steady-state flow conditions. However, if the flow in the floodplain is not well confined within the valley walls (i.e. a typical spill condition) and the flow directions and extent of flooding throughout the floodplain are not easily defined, then a typical 1D modelling approach is no longer suitable.

The limitations of a 1D-steady-state river modelling approach are particularly notable in an urban setting where the engineered surface drainage takes flow in many different directions based on only subtle changes in topography (particularly when flooding is deeper than the curbs on the streets).

Based on the above noted advantages and disadvantages, a 1D river modelling approach is most suitable for projects:

- With well incised river channels with well constrained valley corridors
- Where flow patterns, velocities and depths in the floodplain can be generalized

2.1.2 2D River Modelling

2D river modelling refers to a 2-dimensional representation of the flow in the river and the floodplain. More specifically, the flow in the river channel and the floodplain is not constrained to a uniform, 1D flow direction across the entire width of the channel and floodplain but, rather, flow can occur in any horizontal direction according to the elevation and slope of the river bathymetry and land surface, the flow velocity and momentum, hydraulic structures, and model boundary conditions.

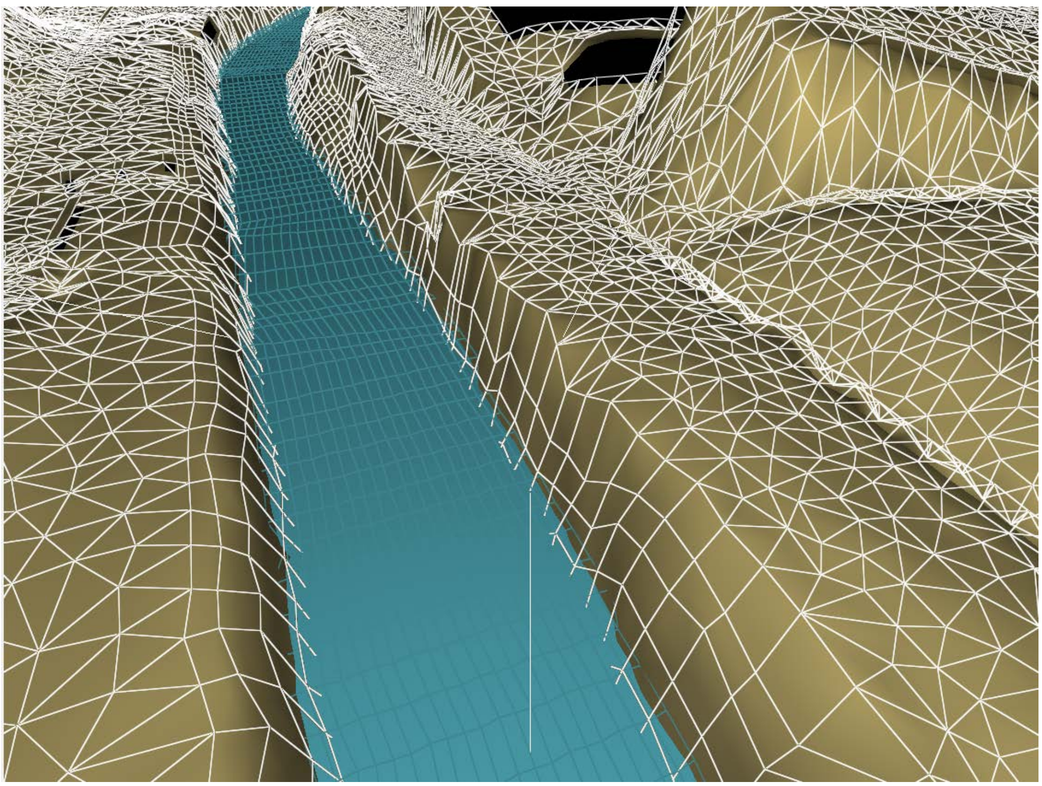


Figure 1: 2D River Modelling

Although the concept of 2D river modelling may seem to indicate a more complex and time-consuming process for model build, the data requirements are nearly identical to those used to develop a 1D model. Both 1D and 2D models require river channel bathymetry and roughness, valley corridor and adjacent land surface topography and roughness, hydraulic structure descriptions, representation of buildings and structures, and boundary conditions. Given the availability of LiDAR and GIS data as well as the associated data processing tools, the preparation of data is not significantly different between the 1D and 2D modelling approach. In fact, the advantage of using a 2D modelling approach is that it eliminates the need for selection and management of river cross-section locations and alignments. In addition, the flow for the entire study area is solved using only one model and it produces a single 2D modelling result files representing detailed flow, velocities, water levels and depth in the river channel and floodplain across the entire study area.

However, at present, there are also some limitations associated with 2D river modelling:

- There is no classic 'steady-state' solution so, in order to achieve a similar steady flow condition. the simulation needs to be run for a long period of time with constant boundary conditions until a near steady flow conditions develops throughout the study area.
- It is a relatively new approach so the flood modelling and floodplain mapping guidelines for Ontario (Technical Guide – River and Stream Systems: Flooding Hazard Limit, MNR 2002) lack the flexibility to accommodate modern 2D flood modelling approaches.
- The representation of hydraulic structures is different than the hydraulic structure representations used in 1D river modelling.
- The model requires a very fine resolution of the grid/mesh in the river channel in order to accurately represent the channel geometry – in particular for deep channels with steep sloping banks.
- The 2D models take longer run times than a typical 1D river modelling approach.
- The results files are much larger and are more time-consuming to interpret and analyze than with a typical 1D river modelling approach.

Based on the above noted advantages and disadvantages, a 2D river modelling approach is most suitable for flood modelling and mapping projects where:

- There is a need for a detailed representation of overbank/floodplain flows.
- The model will need to be used for other purposes requiring a 2D representation of flow in the river channel (e.g. sediment transport, bank erosion, ecological considerations).

2.1.3 Coupled 1D-2D River Modelling

Coupled 1D-2D river modelling involved a dynamic coupling between a 1D model of the main river channel and 2D model of the floodplain. The main coupling between the 1D model and the 2D model occurs along the banks of the river such that when the water level in the river channel rises above the banks it will effectively 'spill' into the 2D model domain. Once the flooding enters the 2D model domain it can flow in any horizontal direction according to the gradient of the water surface, the slope, elevation and roughness of the land surface, and any influencing structures and boundary conditions. The mechanisms of the coupling between the 1D and 2D models are handled in a number of different ways depending on the modelling software and the settings used in the model but, in general, the direction of the exchange is driven by the water level differences between 1D and 2D model (i.e. if the calculated water level in the 1D river model is higher than the calculated water level in the 2D model at that location then water will spill from the 1D model into the 2D model). An example of 1D-2D model coupling is illustrated in Figure 2.

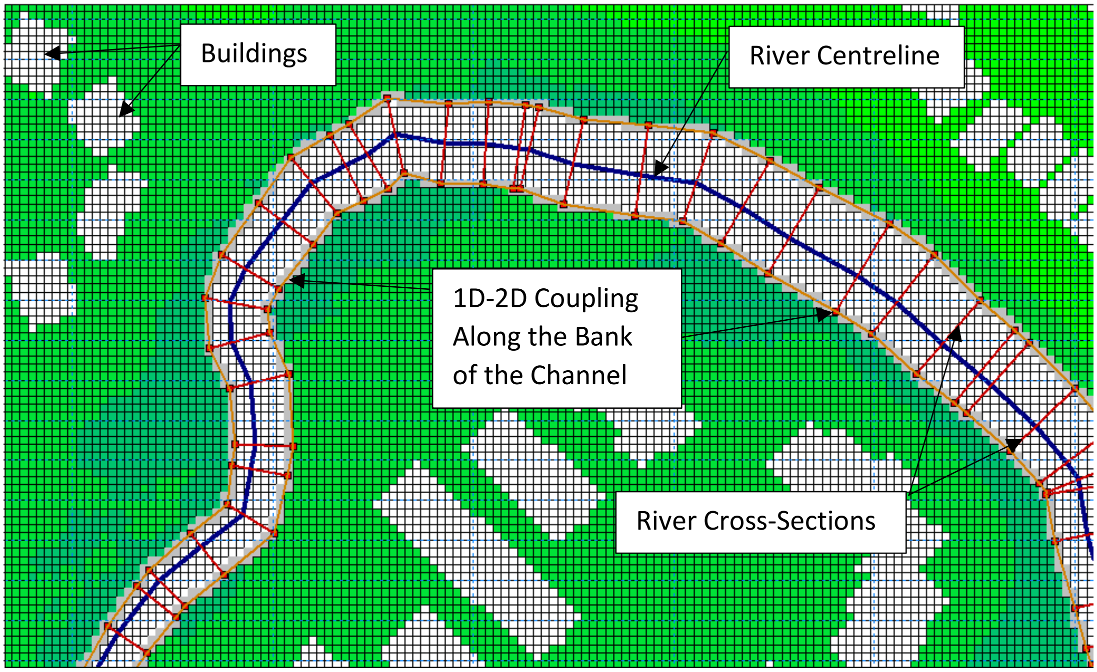


Figure 2: Example of 1D-2D River Model Coupling

Depending on the software selected for the modelling, some additional 1D-2D model coupling options may be also available including:

- Coupling between the 2D model and the upstream or downstream ends of the 1D river model
- Coupling between a 1D urban collection system model, a 1D river model and a 2D overland flow model

The advantages of a 1D-2D coupled river modelling approach is that it combines the classic approach and formulations typically associated with modelling a 1D river channel with the ability to more accurately and realistically represent flooding outside of the main channel. It also allows for a better representation of in-line structures, specifically long, buried structures such as culverts and diversion pipes, as well as real-time operational structures.

The disadvantages of a 1D-2D coupled river modelling approach is that it requires the development and maintenance of a 1D river model, a 2D overland flow model, and the coupling mechanisms connecting the 1D and 2D models. It also generates two different result file formats (i.e. a 1D river channel result file and a 2D overland flow result file). In addition, depending on the model setup and flow conditions, the run times for a 1D-2D coupled river model are very similar to a 2D river model.

2.2 Solution Methods

Keeping in mind the purpose of this section of the document is to aid in the selection of an appropriate modelling approach for addressing the needs of a flood modelling study, the term 'Solution Methods' refers to the mathematical equations that are being solved as well as the available computational processing options associated with the available modelling approaches. While the equations used to solve for flow and water levels in river systems are obviously the key component in the model, this document does not provide a detailed discussion of the equations other than what is necessary to differentiate between the different approaches and identify their advantages and disadvantages.

2.2.1 1D Unsteady Flow using MIKE 1D

MIKE 1D is the name of the numerical engine that is used in the MIKE HYDRO River software product. MIKE Hydro River is the successor to the MIKE 11 river modelling software product.

MIKE 1D solves for unsteady flow in a river system based on the solution of the 1D, fully dynamic St. Venant equation for conservation of mass and momentum using an implicit, finite difference numerical scheme (DHI, 2017). It offers 64-bit, multi-core processing capabilities using an OpenMP parallelization scheme that can improve solution speeds vs. a single core processing option.

Although the unsteady flow solution is considered to be more representative of flow conditions than the steady-state solution, it is not typically used in flood modelling and mapping unless it is dynamically coupled with a 2D model. In the case where a steady flow solution is needed in order to satisfy the requirements of flood modeling and mapping guidelines and standards, the unsteady flow model can be run with constant inflows for as long as required to achieve a steady flow condition throughout the model domain.

2.2.2 2D Finite Difference using MIKE 21 HD Classic

The MIKE 21 HD Classic model requires the spatial domain of the model to be discretized using a uniform grid cell size throughout the entire model domain. It solves for water levels and flow in each grid cell using an implicit, finite difference solution of the 2D shallow water flow equations (DHI, 2017). It offers a 64 bit solution with support for multi-core processing using an OpenMP parallelization scheme but the gains in solution speed are usually limited to a factor of less than 2 regardless of how many CPU cores are available.

The advantages of the MIKE 21 HD Classic model are:

- It is a very stable, efficient and reliable solution.
- It is relatively easy to set up the model grid.
- The gridded format is typically familiar and easy to work with for data preparation and editing. Comparing results from different scenarios is straight-forwards as long as the model domain and cell size don't change between scenarios.

The disadvantages of the MIKE 21 HD Classic model are:

- The same grid size must be used throughout the entire model domain.
- The shape of the model domain must be rectangular.
- The solution times cannot be significantly reduced with additional computational resources.

The necessity of using a uniform grid size throughout the entire model domain is often a limiting factor in determining the suitability for using it in 2D river flood modelling projects. For 2D river flood modelling in urban areas a small grid size (e.g. 2x2 m) must be used in order to achieve an accurate representation of the topography, buildings, and river channel geometry/conveyance. The small grid size must be applied everywhere throughout the model domain. If the study area is relatively large (e.g. greater than 5 km^2) then the model will have more than one million grid cells and it will require long times to run each simulation. Parallel processing may help to speed up the simulation but, typically, with an implicit, finite difference solution the improvement is less than a factor of 2, regardless of how many CPU cores are available.

In addition, although MIKE 21 HD Classic can be used to simulate flow in a river channel and floodplain, it is not typically applied for these types of applications because the flexible mesh version of MIKE 21 is better suited for 2D modelling of flow in a river channel. However, the simplicity of the model setup and the stability of the MIKE 21 HD Classic models makes it a very practical choice for a coupled 1D-2D river modelling project.

2.2.3 2D Finite Volume using MIKE 21 Flexible Mesh

The MIKE 21 Flexible Mesh model provides the ultimate flexibility for discretizing the model domain with a fine resolution mesh in the most sensitive areas, larger mesh elements in less sensitive areas, mixed triangular and quadrangular mesh element shapes, any shape of model domain, and any number of separate model domains. It solves the 2D shallow water flow equation using a finite volume numerical scheme solving for water levels at each node and flow across each element face (DHI, 2017). The MIKE 21 Flexible Mesh model offers a 64-bit solution with support for multi-core processing using an MPI scheme capable of near linear increases in speed with each additional processing core. It can also be solved using a high-performance GPU using CUDA Core technology.

The main advantages of using the MIKE 21 Flexible Mesh model are:

- It allows for mixed mesh sizes throughout the model domain and a model domain shape that fits to the area of interest – thus eliminating unnecessary computation points and improving the efficiency of the solution.
- It is capable of utilizing any number of computational cores or a high-performance GPU to significantly reduce the time required to run the simulation.

The main disadvantages of using the MIKE 21 Flexible Mesh model are:

- The design and configuration of the mesh takes longer than setting up a MIKE 21 HD Classic model grid.
- The solution takes longer to solve than a similar MIKE 21 HD Classic model if you don't have access to a high powered, multi-core CPU or a high-performance GPU
- The numerical solution, while quite stable, isn't as robust as the MIKE 21 HD Classic model

- Comparing results from different scenarios is difficult if the mesh design has been changed between scenarios, this is a particular issue when using the model to evaluate design concepts or the performance of a design.
- Merging 1D channel and 2D overland flow results is challenging due to the different result data storage formats (e.g. Raster format vs. TIN format).

In spite of the disadvantages noted above, the ability to tailor the model mesh in areas of interest and the ability to leverage computational power to reduce the simulation time have shown to be the more dominant factors leading to a general trend toward more flexible mesh modelling throughout the flood modelling, river modelling and coastal modelling industries.

2.2.4 1D-2D Model Coupling Methods

When selecting the modelling approach that is best suited for the study area it is important to not only understand the advantages and disadvantages of the different model solutions, but to also understand and appreciate the advantages and disadvantages of the methods used to couple the 1D and 2D models. The 1D-2D model coupling describes the location where exchanges between the two models are considered, the method used to determine whether or not flow is exchanged between the two models, and the formula used to calculate the exchange of flow between the 1D and 2D models. The methods available for coupling a 1D river model to a 2D overland flow model in MIKE FLOOD include:

- Lateral Links
- Standard Links
- Implicit Structure Links

The following is a brief overview of each coupling method.

<u>Lateral Links</u>

Lateral links are used to couple the 1D channel model with the 2D overland flow model by establishing a connection line along the left and/or right bank of the channel - effectively defining the location across which the two models can exchange flow (see example in Figure 2). Each lateral link line has a chainage corresponding to the chainage of the branch to which it is connected. This chainage is then used to determine which 2D model grid cells/mesh elements are compared to which water level nodes in the 1D model in order to determine the direction in which the flow is being exchanged (i.e. is water from the 1D channel flooding the 2D model domain, or is water from the 2D model flow into the 1D channel). Figure 2 shows an example of the 1D-2D lateral link connectivity between the 1D and 2D models.

The rate of exchange of flow is calculated by determining the difference in water level between the two models at each Lateral Link location at each time step, and then using either a weir equation or an exponential function to calculate the exchange of flow at each time step. Figure 3 shows a schematic of how water levels in the linked grid cells of a MIKE 21 HD Classic model (M21) are compared to the water levels in the linked grid points of a MIKE 11 model (M11).

It is important to note that momentum is not preserved during the exchange of flow, either from the 1D model to the 2D model or vice versa. However, it is important to note there is an adjustment of the momentum in the 1D model when flow leaves the channel and when flow is added to the 1D channel. This could have significant consequences if a normally meandering channel is overtopped during a flood event such that flow from the 2D model is running perpendicular to the channel during peak flow periods, or if the depth of water in floodplain is deeper than the bankfull depth of the channel itself. The significance of the momentum adjustment depends to a large extend on how much deeper the 2D flooding is than the 1D channel. If the flooding event of concern creates a condition where flow in the main channel is completely submerged by a larger floodplain channel, then some consideration must be taken as to whether the momentum losses across the channel will have a significant influence on the results. Unfortunately, there are no benchmarking studies to provide suitable references or guidelines for this.

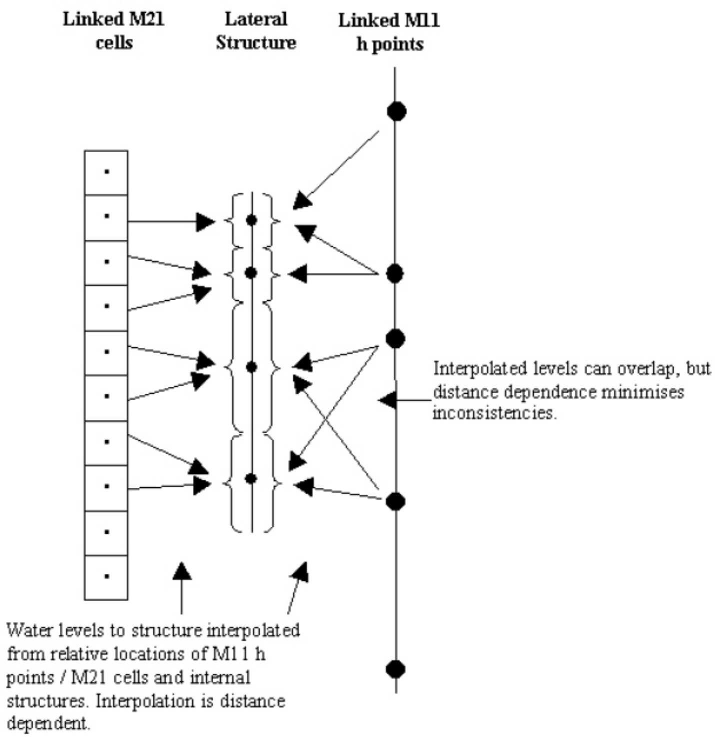


Figure 3: Schematic of Lateral Link water level comparison method

It is also important to note that when lateral links are used to link a 1D MIKE HYDRO River model to a 2D MIKE 21 HD Classic model where the 1D channel is wider than 2 grid cells, the grid cells overlapping the channel should be deactivated in the 2D model in order to avoid double counting the flow in both the 1D and 2D models. While the lateral links act as a transfer mechanism to exchange water from the 1D and 2D models, they do not act as a flow barrier. As such, if the 2D grid cells overlapping the 1D channel area are not deactivated then they would eventually fill with water and act as artificial storage in the 2D model. Figure 2 provides a good illustration of the situation where the 1D channel is much wider than the grid cells. If the riverbed bathymetry is represented in the 2D model topography, then the 2D model could potentially store a significant amount of overland flow. Although this amount of storage is typically minor compared to the overall volume of water considered in a major flood event, it could be significant when evaluating smaller flooding events. This issue is not a concern for the Flexible Mesh model because the 1D channel area can be omitted from the mesh.

Standard Links

Standard Links are used to couple the ends of the 1D channel model to the 2D overland flow model by establishing a line along the furthest upstream and/or downstream cross-section location across which flow can be exchanged between the two models. Figure 4, shows an example of a Standard Link coupling the end of a river branch with a 2D surface water body.

The 2D model grid cells / mesh elements intersecting the Standard Link line will be able to exchange flow with the 1D model according to the discharge in the 1D model and the average water level of the coupled grid cells/mesh elements in the 2D model. Unlike the Lateral Links, momentum transfer between the 1D and 2D models can be preserved using a Standard Link.

The Standard Link is often used to couple a 1D river channel to 2D open water body, or to facilitate representation of a long, buried culvert in a 2D model, or drainage of a stormwater management pond into a creek.

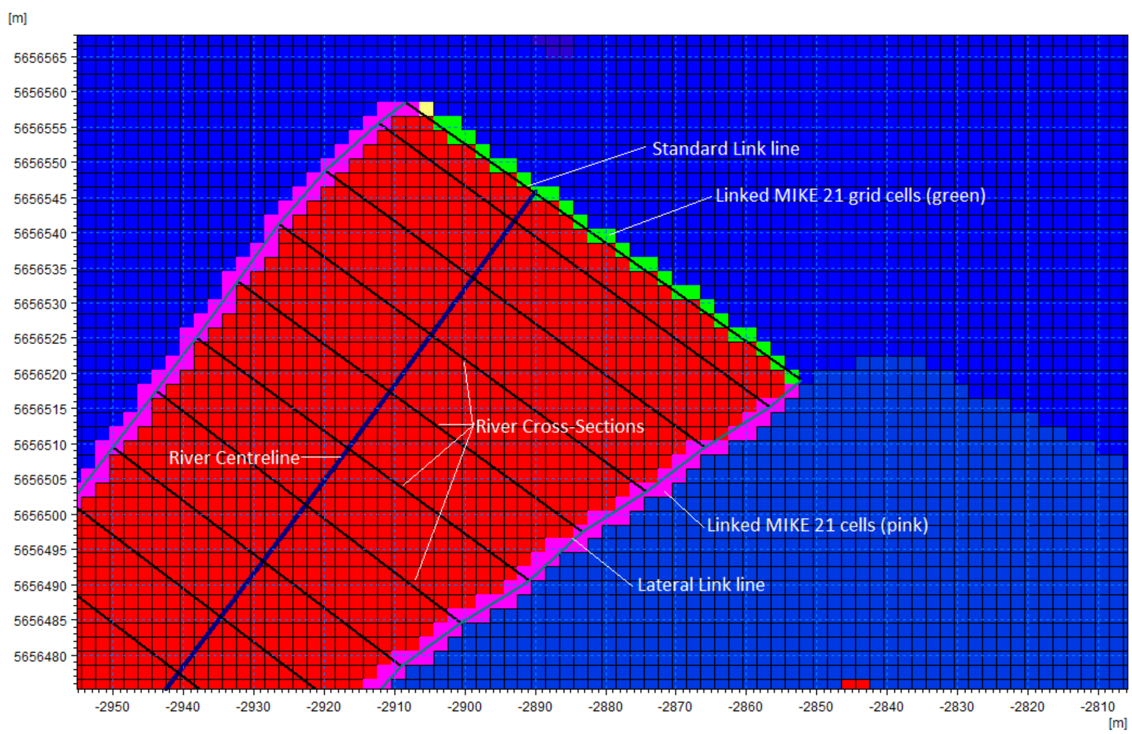


Figure 4: Example of a Standard Link coupling the end of a river branch to a 2D water body

If a Standard Link is used in a rain-on-grid or rain-on-mesh model (i.e. where rainfall is applied across the entire model domain) care should be taken to carefully examine the results at the Standard Link location to ensure it makes sense. There are potential problems with calculating the exchange of flow between the models because the Standard Link uses the average water level in the linked MIKE 21 grid cells/mesh element to determine the water level in the 2D model. If there is rain applied on all grid cells then every linked 2D grid cell/mesh element will have a water level (even though it is very shallow) and this may over-predict the average water level in the 2D model at that location.

Structure Links

Structure Links are only available with MIKE 21 HD Classic. The Structure Link takes the implicit terms describing momentum through a 3 point (H-q-H) MIKE 1D branch and uses them to replace or modify the implicit terms describing momentum across the face of a MIKE 21 cell. In this way, the flow properties from one MIKE 21 cell to another are modified to represent a structure. The Structure Link is primarily there for legacy reasons as it was developed prior to structures being directly supported in MIKE 21. However, modern versions of MIKE 21 support the description of structures directly in the MIKE 21 grid or mesh.

2.3 Grid/Mesh Resolution

What is the appropriate computational spacing (or resolution) for the study? This could be one of most important questions to answer when working with any fully hydrodynamic 1D/2D models. If a coupled 1D and 2D modeling approach is selected for the study, then the discretization of both the 1D model and the 2D model needs to be determined.

Computational spacing for 1D model refers to the spacing between cross-sections. When constructing a 1D-2D coupled model the spacing between cross-sections of the 1D model should typically be approximately 5-20x the length of the sides of the 2D model grid cells or mesh elements. The reasons for the tight spacing are; (a) to ensure the water level calculation points of the 1D model are sufficiently close to the corresponding calculation points of the 2D model to provide an accurate calculation of the exchange of water between the two models; and (b) to ensure the bank elevation of the 1D model accurately reflects the uneven terrain along the channel in order to capture the appropriate spill points. Figure 2 shows a good example of a coupled 1D-2D flood model where the 1D model cross-sections are 5 m to 15 m apart and the 2D model grid cells are 2x2 m.

Computational grid spacing for a 2D model refers to the cell size or mesh size depend which 2D engine is chosen. When selecting computational spacing for a 2D model the following factors should be considered:

1) **Resolution of the available topography data:** Typically, the size of the model grid cells or mesh elements should not be smaller than the resolution of the available topographic data. If the available topography data is only available in with 10 m spacing between data points, it doesn't usually make practical sense to construct a model using 5 m grid cells or mesh elements because the model will not be any more accurate and it will take much longer to run the simulation. The only potential reason to construct the model with smaller grid cells or mesh elements is for the purpose of better representing buildings or known topographic features or structures that will influence the overland flow.
2) **Level of detail needed to meet the objectives:** Although the topography has the most significant influence on the overland flow direction and depth, the level of detail of the

model should not necessarily be dictated by the resolution of the available topographic data. As LiDAR and other forms of high resolution topographic data become more accessible and available, care must be taken to balance the desire for accuracy in the results against the excessive computational burden of running models with very small grid cells or mesh elements. While this particular issue is more applicable for the MIKE 21 HD Classic model (since all grid cells in the entire model domain are the same size), it also applies to the MIKE 21 Flexible Mesh model. The size of the grid cells or mesh elements only need to be as small as necessary to represent the important topographic features that will influence the direction and depth of 2D overland flow. In a natural, undeveloped setting a grid cell size of 5 x 5 m, supplemented by 1D models for channelized flow, may be sufficient to capture the important overland flow routes. While in highly developed, urban areas the engineered drainage features, including roadways, drainage ditches, and berms, may require a grid cell size of 2 x 2 m in order to accurately represent overland flow and flooding. Figure 5 shows and example of variable mesh sizing to represent different landforms and land uses.

3) **Budget and schedule of the project:** The size of the grid cells or mesh elements impacts the budget and schedule of a project because models with smaller grid cells or mesh elements will have many more computational points and will take much longer to run the simulations. In a typical flood modeling project, it is usually required to run the model with a range of different design storms, and sometimes for both steady and unsteady conditions. In addition, it is sometime also required to run a number of sensitivity runs, scenarios or alternatives to evaluate different potential flood mitigation schemes, and development concepts. Between model sensitivity analyses, flood mitigation and development impact analyses, and production runs for mapping, the total number of model runs can easily exceed 75. If each model takes 24-72 hours to run through to completion it’s not difficult to see how this can impact the deadlines and budget for a project. Care should be taken when developing the schedule for a project to ensure there is sufficient time and computational resources available to complete the model runs.

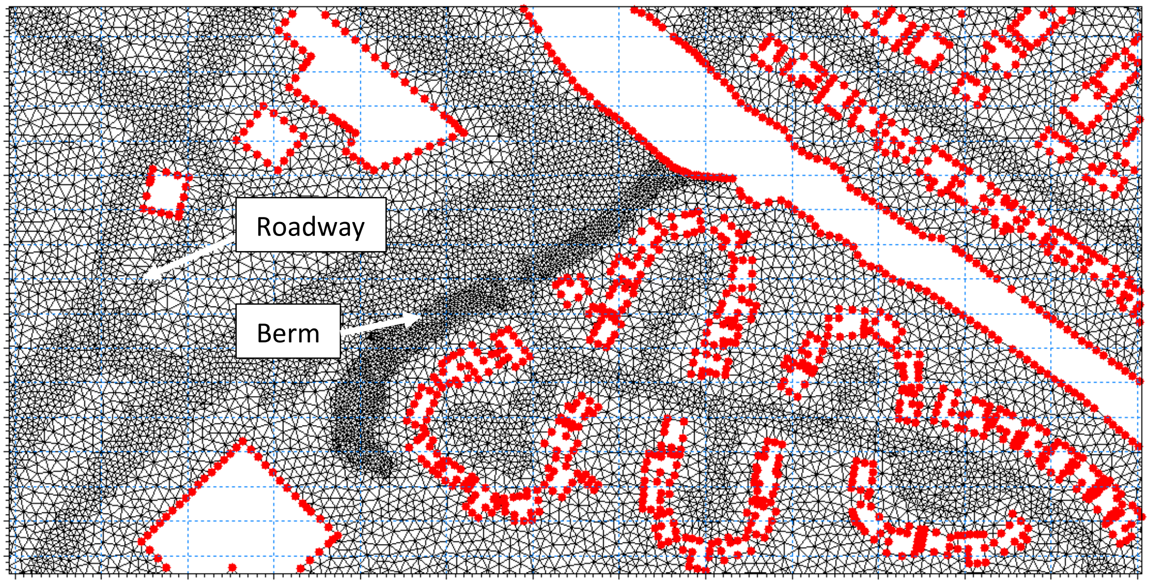


Figure 5: Example of variable mesh resolutions for representing different landforms

2.4 Steady or Unsteady Flow

The most commonly accepted and recommended approach to flood modelling and mapping has been to apply peak flows in a 1D hydraulic model using a steady-state solution. This approach is both numerically efficient as well as being conservative in terms of modelling the probable worst-case conditions. Due to the additional complexities of 2D overland flow modelling it is not possible to solve a 2D overland flow model using a steady-state solution. However, a 2D model can represent a steady flow condition using inflow boundary conditions with a constant flow rate (see left graph in Figure 6) for as long as necessary to achieve a near steady state hydraulic condition throughout the study area.

TRCA currently uses a steady flow approach for floodplain mapping purposes in order to maintain consistency with the current set of provincial flood mapping guidelines (MNR, 2002), and because it is not possible to preserve the storage components located within the floodplain. However, there are limitation to this approach when screening property development applications because any change in the floodplain will naturally produce some off-site impacts (i.e. if you remove some storage from the site then that water has to be allocated elsewhere since it's a steady-state model). A property development application assessment using an unsteady flow provides a more practical reflection of the potential impacts.

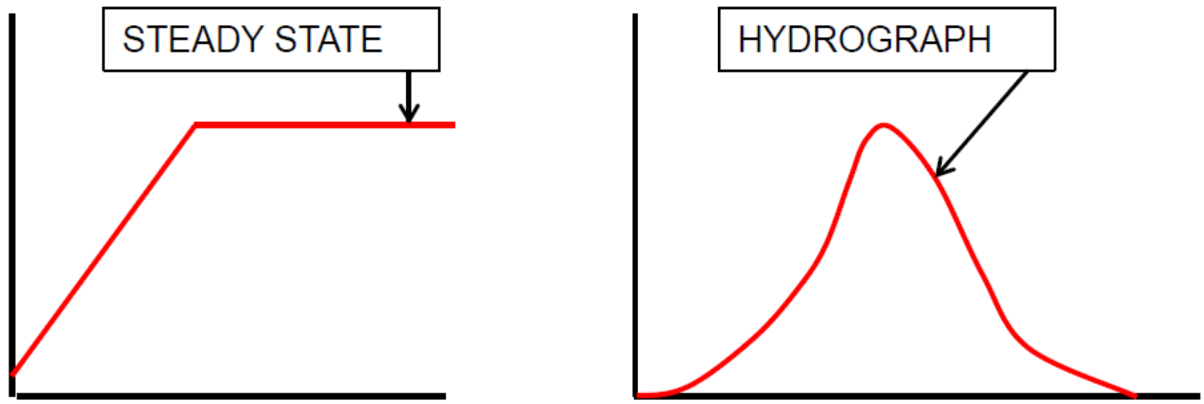


Figure 6 Comparison of steady flow rate and unsteady hydrograph

3 Data Requirements and Preparation

As mentioned previously, whether the modelling approach is a 1D, 2D or coupled 1D-2D, the data requirements for developing a hydraulic flood model are very similar:

- Previous models and reports
- Hydrology model
- Surface topography
- Channel bathymetry and characterization
- Hydraulic structure locations and geometry
- Channel inflows and extractions
- Land use and vegetation
- Building footprints
- Significant surface drainage control features
- Aerial imagery
- Flow and water level monitoring data

Although the data requirements are similar, the way the data is used in a 2D model is much different than how it is used in a 1D model. The following sections will discuss each of these data requirements and provide some recommendations for preparing the data to be used in a 2D flood model.

3.1 Previous Models and Reports

Although it isn't necessary to have a previous model and/or study of the area, many 2D modelling studies are initiated because it was determined the existing 1D river hydraulic model was not able to sufficiently represent the expected flooding. While the existing 1D model may not provide a suitably accurate representation of the flooding conditions, the data and results can provide some valuable information about the location(s) where flooding is likely to occur and the potential extents of flooding. This information is useful in determining the required spatial extents of the 2D model domain.

If a coupled 1D-2D flood modelling approach is being used then a significant portion of the existing 1D model data can be re-used including the topology, the centerline, cross-sections and roughness of the channels, and the hydraulic structure locations and geometry.

In addition, if the existing model extends upstream and/or downstream of the area of interest and it has been calibrated and/or accepted for use in floodplain mapping, it can also be used to define the upstream and/or downstream boundary conditions for a more localized 2D flood model.

Reports from previous modelling studies will also be useful for reviewing assumptions made during the development of the previous model, finding potential data gaps for the 2D modelling study, evaluating the quality of the calibration of the model, and determining where the site characteristics may have changed since the previous model was prepared.

3.2 Hydrology Model

The hydrology model provides the inflows to the river hydraulic model from the contributing catchments within the study area and, potentially, from the segment of the river basin upstream of the study area. The inflows from the hydrology model are a critical component for calibrating the river hydraulic model to observed flows and water levels, and it also provides the inflows used for evaluating design storm events and/or future climate change.

If the hydraulic model is being calibrated it is important to remember that any uncertainties and errors in the hydrology model will be carried over into the hydraulic model. As such, any expectations for the quality of calibration of the river hydraulic model need to be balanced against the uncertainty associated with the contributing inflows from the hydrology model.

If the existing hydrology model for the 2D flood modelling study area is too coarse and does not sufficiently resolve inflows from catchments within the study area then the hydrology model may need to be refined prior to proceeding with the 2D flood model.

3.3 Surface Topography

A high-resolution and accurate surface topography data set is generally considered to be a prerequisite for a 2D flood modelling study, particularly when the study area is located in an urban setting with engineered surface drainage features. At the time of writing this document, most 2D flood modelling studies rely on LiDAR data with at least a 1m horizontal resolution and 10-15 cm vertical accuracy. However, in some cases, the LiDAR data may not reflect recent changes in topography due to known human or natural processes. In these cases the LiDAR data needs to be combined with other data sources including design drawing, as built drawings and/or land surveys.

If the topography data is provided in small tiles, these tiles need to be combined to form a single topography data set in order to be used by the MIKE tools (e.g. MIKE HYDRO River or MIKE 21).

If topography data will be used to define the channel geometry then the original (i.e. finest) resolution should be preserved for generating 1D model cross-sections or for defining 2D grid cell/mesh node elevations within the channel. For example, if the original LiDAR data has a 0.5 m horizontal resolution and it needs to be converted to a 2D raster format in order to define channel bathymetry, then the raster file should be set to a 0.5 m grid size. Furthermore, if the LiDAR data includes roadway surface elevations at channel crossing locations, the roadways may need to be removed depending on the methodology used to represent the culverts or bridges.

If the LiDAR data is going to be used to define the channel geometry then it should also be noted that current LiDAR technology does not provide an accurate representation of the bathymetry of surface water bodies including rivers, creeks, ponds and lakes. The LiDAR data will typically represent the water surface elevation as the land surface elevation. Depending on the time of year when the LiDAR data is generated, the characteristics of the channel, and the magnitude of flows considered in the study, this may or may not be a significant consideration. However, some efforts should be made to justify and document whether or not it will have a meaningful impact on the results of the 2D modelling study. This is discussed further in the following section.

3.4 Channel Bathymetry

The channel bathymetry and bed characterization is required in order to provide an accurate representation of the conveyance in the river channel. Although channel cross-section survey data may be available at a few locations within a study area, the distance between the cross-sections is usually too large to be used as the only source of channel bathymetry data. The problem with large distances between cross-sections is that it assumes a straight-line interpolation between cross-sections. This is not suitable for a 2D model or a coupled 1D-2D model because of the potential discrepancies between the bank elevations represented in the detailed 2D topography data and the bank elevations interpolated between the cross-sections.

In most cases, the channel bathymetry for a coupled 1D-2D model will need to be extracted from the detailed 2D topography data at distance intervals that are between 5 to 20 times the size of the 2D model grid cells/mesh elements. As such, the channel cross-section survey data is not usually detailed enough for this purpose. However, the channel survey data can be useful to compare it against the cross-section extracted from the detailed 2D topography to determining whether it provides an accurate enough representation of the channel for the purposes of the study. For example, if LiDAR topography data is collected at a time when water levels in the channel are high then the channel bathymetry will not be accurately reflected in LiDAR data. The significance of this difference can then be assessed against the overall objectives of the study to determine if additional survey data is required.

If the channel survey data shows the channel has significantly more conveyance than what is extracted from the detailed 2D topography data then some additional processing of the 2D topography data may be necessary in order to better represent the conveyance in the channel. Depending on the shape of the channel, the magnitude of the difference, and the consistency of the differences along the channel, the correction of the 2D topography data could be approached in a number of different ways. The following outlines two examples of methods that could be implemented to adjust the 2D topography data to better represent channel bathymetry:

- If the channel is wide and relatively shallow and the difference is relatively uniform along the channel then the existing channel topography can be adjusted by subtracting the average difference from all of the data points within portion of the channel covered by surface water. This effectively lowers a portion of the channel by a uniform amount along the entire channel. This is a relatively easy to implement solution.
- If the difference between the survey data and 2D topography data is inconsistent along the channel then a cross-section of difference values can be created at each survey location and these values can be interpolated along the channel between survey locations to create a raster of difference values along the channel. This raster can then be subtracted from the original 2D topography data.

3.5 Hydraulic Structures

Hydraulic structures typically refer to dams, bridges, culverts, gates, and weirs – essentially any structure that acts to influence flow in the channel where the structure itself is not represented in the channel geometry. These structures can be represented in either a 1D riverine model or a 2D overland flow model, or a combination of both depending on the setting and objectives of the study. While there are many options and data requirements for setting up and defining structures in MIKE HYDRO River and MIKE 21, the basic physical data requirements for the different structures is consistent regardless of the approach. Table 1 provides a list of the basic physical data requirements for each structure type.

Table 1: Summary of Hydraulic Structure data requirements

Dams	Bridges*	Culverts	Weirs	Gates	Pumps
Location	Location	Location	Location	Location	Location
Elevation of the crest	Dimensions of the opening(s)	Dimensions of culvert	Elevation of the crest	Gate type (overflow, underflow, radial, sluice)	Water level to initiate pumping
Breach geometry	Dimensions of pier(s) (if any)	Roughness of culvert	Width of the crest	Width of gate	Water level to stop pumping
Breach timing	Elevation of the deck	Length		Elevation of sill	Pumping rate
	Length (in the direction of flow)	Upstream and downstream invert elevation		Elevation of maximum gate level	
				Maximum speed	

*The number of combinations of methods for representing bridges and data requirements for each method is too numerous to tabulate in this document but most of the geometric data requirements can be derived from the data requirements list in this table.

If the structures contain any operational components that are relevant for flooding conditions then the operational rules for the structure(s) could be coded into the model as well.

3.6 Channel Inflows and Outflows

Channel inflows may consist of flow coming from upstream of the modelled channel, small tributary channels that are not explicitly represented in the model, storm sewer outfalls along the channel, or catchment drainage points from a hydrology model. Channel outflows may consist of the downstream end of the channel as well as locations where water is removed from the channel for water supply, irrigation or industrial uses at rates that are meaningful in the context of the study.

Channel inflow and outflows are typically represented in the model as boundary conditions and it is important to know the location where the inflows and extractions are taking place, the rate at which water is being added or removed, and whether it is added or removed at a single location (point source) or uniformly along a length of the channel (distributed source).

3.7 Land Use and Vegetation Maps

A map of land use and vegetation for the study area is typically needed to establish estimates for the surface roughness of the 2D overland flow model. TRCA has established a standard set of Manning's n Roughness values for a variety of land use categories to be used for hydraulic modelling purposes (see Table 2).

Table 2: Standard TRCA Manning's roughness values

Surface Description	Manning's n	Manning's M
Paved Surface	0.025	40
Urban Pervious	0.050	20
Natural Areas	0.080	12
Natural Channel	0.035	28

The map of land use and vegetation types is typically provided in a polygon shape file format. If the Manning's n roughness values are not already associated with the land use types then the shape file should be edited to add a field for both the Manning's n and the Manning's M (the inverse of Manning's n). The reason for adding a field for Manning's M is because this is the value used by MIKE 21.

Once you have a shape file of land use types with associated Manning's M values it can be used to generate the required surface roughness values in DHI dfs2 format for use in the MIKE 21 2D overland flow model.

3.8 Building Footprints

The building footprints are usually provided in an ArcGIS shapefile format and are used to represent the building as impermeable barriers in the 2D overland flow model. If the MIKE 21 Flexible Mesh model is being used for 2D overland flow then the building footprint shapefile should be 'simplified' to remove small structures like sheds, to smooth the jaggedness in the shapes of the buildings, and to merge buildings that have very little space between them. If the building shape file is not simplified then the mesh generator may fail as it tries to generate very small mesh elements around small jagged shapes, or it may generate many very small elements that will slow down the simulation solution. TRCA has developed an in-house methodology to automate the processing of building structures (see Appendix A).

3.9 Significant Surface Drainage Control Features

Significant surface drainage control features include roadways, drainage ditches, berms, walls any other surficial feature that may significantly influence the overland flow direction. The locations of these engineered drainage features are important to ensure they are properly represented in the model. With the MIKE 21 Flexible Mesh model, the area within and around the roadways, ditches and berms should be delineated using a mesh size that is small enough accurately characterize the hydraulic influence of these features (see Figure 5).

3.10 Imagery

Aeriel imagery is typically used as background map for visual reference and also to verify ground feature locations such as channel central line, river bank alignments, buildings, roads etc. The most commonly used image files are JPG, BMP, TIF, GIF, PNG and SID.

4 Model Setup

The purpose of this section is to provide some guidance on setting up a MIKE FLOOD model. Since there are many different combinations of models that can be used, this section covers each of the different models individually and includes some discussion of considerations and recommendations for the different modelling approaches.

4.1 1D Model Setup – MIKE HYDRO River

The construction of the 1D river model is typically performed in the following steps:

- Define channel center-line
- Define channel cross-section geometry
- Insert hydraulic structures
- Assign boundary conditions
- Define simulation settings

This section describes the methodology used to complete each of these steps.

4.1.1 River Network

The river network represents the centre-line of 1D channel representing the rivers, creeks, streams or drainage ditches being modelled in the study area. In many cases the river network for the study area can be imported from an existing shape file or a previous 1D model.

When manually digitizing or importing a river network from a polyline shape file it is important to understand how the digitized points along the lines are used in a MIKE FLOOD model. When the 1D MIKE HYDRO River model is coupled to a 2D MIKE 21 model via lateral links the algorithm used to generate the lateral link lines uses the bank markers on the cross-sections as well as the curvature of the line. This is done by interpolating the user-defined cross-sections to the digitized point locations to help in representing the curvature (see Figure 7). However, if the digitized points are too close together it will result in some overlap of the interpolated cross-sections and resultant 'looping' of the lateral link line.

In order to avoid the looping of the lateral link line the 1D channel centerline should be simplified or smoothed before importing it into MIKE HYDRO River. After it is imported it should still be examined closely to ensure that any points that are too close together while still maintaining the critical points and overall shape of the line.

It is also important to remember that a river network in MIKE HYDRO River is defined such that the chainage along the river network increases in the downstream direction. This is different than HEC-RAS where the chainage increases in the upstream direction.

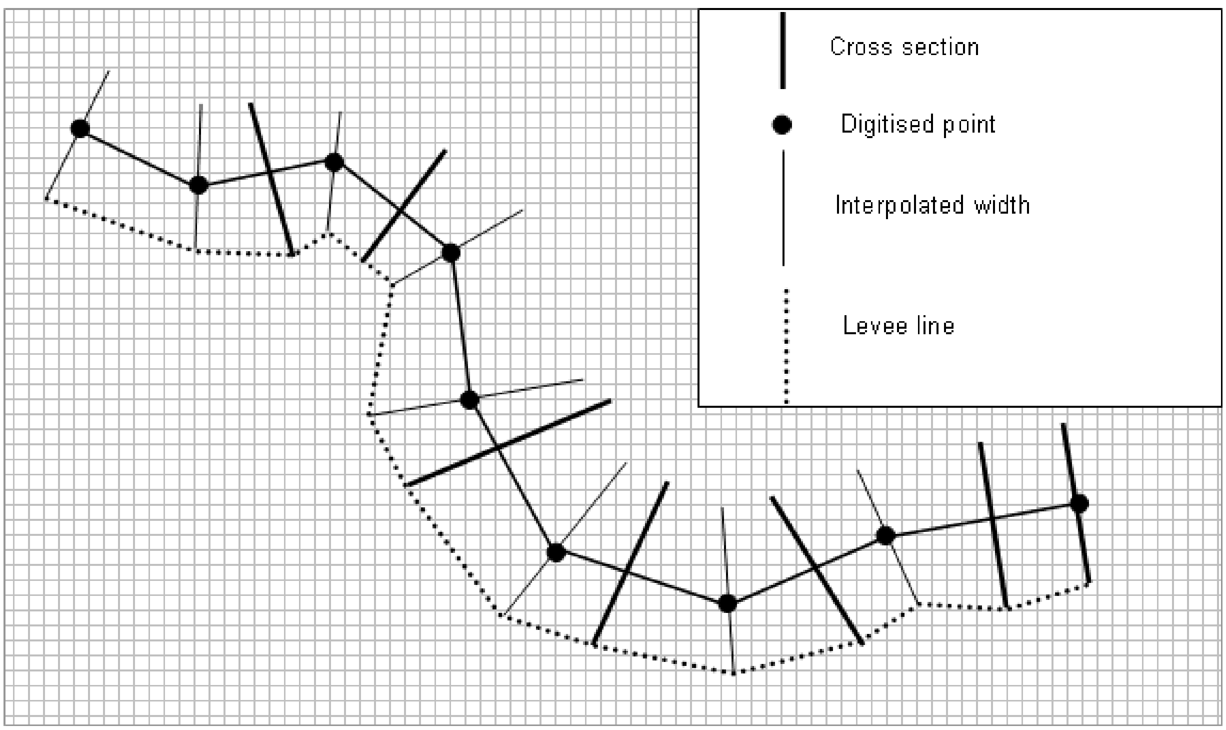


Figure 7 Definition of levee line used for lateral linkage DHI MIKE Flood

4.1.2 Channel Cross-Sections

Channel cross-sections are one of most important inputs in a 1D river model because this information defines the channel capacity, slope and bank elevations. For MIKE HYDRO River, any description of cross-sections is done from the perspective of being positioned upstream of the cross-section and looking in the downstream direction.

Cross-sections are typically defined perpendicular to the direction of flow. Cross-sections are specified by a number of x-z co-ordinates, where x is the transverse distance from a fixed point (often the top of the left bank) and z is the corresponding bed elevation.

4.1.2.1 Cross-Section Alignment and Extent

When developing a 1D river model for the purpose of coupling it with a 2D model, the cross-sections are normally defined such that they only cover the main channelized portion of the network where flow can reasonably be described by a 1D flow model. In cases where a low flow channel is bounded by a well-confined river valley corridor the decision of whether or not to extend the cross-sections to the banks of the river valley corridor should consider whether the low flow channel will have a meaningful influence on the flow during the events of interest. If the conveyance in the low flow channel is inconsequential during the events of interest then the 1D model cross-sections should be extended to the banks of the river valley corridor.

In cases where the floodplain is not well confined by a river valley corridor, the cross-sections should not extend into the overbank floodplain area because the flow in this part of the study area will be handled by the 2D model.

4.1.2.2 Cross-Section Spacing

If the 1D model is going to be coupled to a 2D overland flow model using lateral links along the banks of the channel, then the distance between the cross-sections should typically be between 5 to 20 times the length of the sides of the grid cells (or mesh elements) defined along the edge of the 1D channel model (see Section 3.4). In an urban floodplain setting this usually requires a spacing of 10-50 m between cross-sections (see Figure 2).

The spacing will require some adjustment around structures like culverts and bridges where it is important to locate the cross-sections immediately upstream and downstream of the structure in order to properly represent the contraction and expansion losses.

The spacing may also need to be adjusted at sharp bends or meanders in the channel where close spacing may result in some overlapping of cross-section lines.

4.1.2.3 Cross-Section Data

Typically, channel cross-section data is available from four different sources:

1) Existing 1D hydraulic models
2) Surveyed cross-sections
3) LiDAR topography data
4) As built drawings

If existing 1D models are available then the cross-section data from these models may be used for comparison purposes to determine where the LiDAR data is not able to properly represent the channel bathymetry. Otherwise, the existing 1D model cross-sections are typically spaced too far apart to be used in a coupled 1D-2D model.

If cross-section survey data is available then it can also be imported to the MIKE HYDRO River model. Cross section surveys are normally provided in xyz points in a text format or GIS point shape file format. MIKE Hydro River has a function to read xyz point survey data and generate cross sections from it. In order for MIKE Hydro River to read xyz survey data, the raw survey data need to be pre-processed to a GIS point shape file where each point has an attribute for the cross-section ID and the elevation of the point. The ID connects the individual survey points to a specific cross section, and all survey points with the same ID field will be created as one cross section.

In the case when a 1D riverine model is being developed for the purpose of coupling it to a 2D model, the required spacing of the cross-sections usually dictates that high-resolution LiDAR topography data (or a similar high-resolution data format) is necessary in order to generate cross-section data at the required intervals. For purposes of flood modeling a horizontal resolution of 0.5m – 1m is recommended with a vertical accuracy of 0.1m to capture the meaningful variations of the ground surface. Depending on the time of year when the LiDAR was captured and the hydraulic conditions of the river system at that time, the raw LiDAR data may not provide a sufficiently accurate representation of the channel bottom if the water levels were too high at that time. In this case the LiDAR data may need to be pre-processed prior to generating the cross-sections in order to manually 'burn' the riverbed into the topography (see Section 3.4).

MIKE Hydro River provides a tool for extracting cross-sections from raster data formats according to pre-defined locations or based on a user-defined spacing and width. The auto-generated cross-sections can then be trimmed or extended according a user-defined alignment lines in order to better suit the dimensions of the channel.

In cases where the channel has very steep side slopes (e.g. engineered rectangular channel) the LiDAR data may not provide a suitably accurate representation of the channel dimensions. In the absence of channel survey data, as-built drawings may also provide the information needed to supplement or replace the LiDAR generated cross-sections.

4.1.3 Hydraulic Structures

MIKE HYDRO River provides the opportunity to describe hydraulic structures in the 1D riverine model using any of the following structure types:

- Weirs
- Culverts
- Bridges
- Pumps
- Gates
- Direct discharges
- Dambreak
- Energy losses
- Tabulated

If the 1D model is being prepared for use in a coupled 1D-2D MIKE FLOOD model the setup of the hydraulic structures is no different than it would be standard 1D riverine model. A description of the data requirements and options for each of these structure types is available in the MIKE HYDRO River user manual and/or on-line help.

When a 1D riverine model is coupled to a 2D overland flow model, the road crossings are typically handled entirely by the 1D model such that any overtopping of a structure (i.e. a bridge deck or a roadway surface) within the channel is handled entirely by the 1D model. That is to say, the deck of the bridge or the surface of the roadway is omitted from the 2D model surface where it overlaps with the channel.

4.1.4 Boundary Conditions

Boundary conditions need to be defined at the open upstream ends of the 1D river network, at the open downstream ends of the network, and at appropriate inflow locations throughout the network.

4.1.4.1 Upstream Boundary Conditions

If the upstream open end of a branch extends to the very beginning of the channel then it can be defined as a Closed boundary condition type with no direct upstream contributions.

If the upstream open end of a branch is located in a channel with meaningful upstream flows then the boundary condition should be specified as Discharge boundary condition type. The inflow can be either a constant value or a time-series depending on the desired conditions. In most cases, the upstream ends of the channel(s) will be chosen at flow node locations from the existing hydrology model such that the design storm inflow at the upstream end of the 1D riverine model can be readily obtained from the hydrology model

> **Note:** If the model is being run for steady flow conditions then it is common for the upstream inflow to be defined using the peak flow for the next downstream flow node of the hydrology model. This is typically done to be consistent with the approach used for 1D, steady-state modelling using HEC-RAS and is considered to be a conservative approach. However, in some circumstances the peak flow calculated by the hydrology model at the flow nodes may decrease in the downstream direction due to the simplified routing and peak flow attenuation methods used by the hydrology models. In this case, using the downstream node peak flow is not a conservative approach since the hydraulic model should be used to determine the peak flow attenuation.

If the upstream open end of a branch is located at a large surface water body then the boundary condition should be specified as Water Level boundary condition type. The water level can be either a constant value or a time-series depending on the desired conditions.

4.1.4.2 Downstream Boundary Conditions

If the downstream end of a 1D model network branch is terminated at a location where the flow in the river continues beyond the end of the network branch, then a boundary condition needs to be defined at this location to allow the flow to exit the model. The boundary condition should be defined such that it is able to represent the flow across the boundary under natural conditions. In most cases this can be accomplished using a Q-H boundary condition type. The Q-H relationship is best derived from the measured flow and water level data at that location or, if that data is not available, it can either be estimated by assuming critical or normal flow conditions, or it can be derived from an existing 1D hydraulic model.

If the downstream open end of a branch is located at a large surface water body then the boundary condition should be specified as Water Level boundary condition type. The water level can be either a constant value or a time-series depending on the desired conditions.

4.1.4.3 Internal Boundary Conditions

The internal boundary conditions are typically used to describe the rainfall runoff inflows from hydrologic subcatchments defined within the model domain or small tributaries connecting to the main channel. In rare circumstances the internal boundary conditions could also be used to describe significant extractions from a riverine system. With a coupled 1D-2D model it is important to note that these internal boundary condition inflows are applied as incremental flows between the flow nodes and not the total flows at the nodes.

If the internal boundary condition is describing inflow from a minor tributary or a subcatchment that drains to a point along the main channel, then a Discharge boundary condition type is used with a Point Source location type. The inflow values are typically obtained from the hydrology model as either a time-series (for unsteady flow scenarios) or a peak value (for steady flow scenarios).

If the internal boundary condition is describing rainfall runoff from a subcatchment that drains along a significant length of the channel, then a Discharge boundary condition type is used with a Distributed location type. The inflow from the subcatchment will then be distributed along the channel between two user-defined chainages.

If the model is being run for steady flow conditions then the selection of internal boundary conditions needs to be carefully considered so as not to over-estimate the steady peak flows. If the upstream inflow boundary condition is defined using the peak flow from the next downstream flow node, then the addition of subcatchment inflows need to consider whether these are already accounted for as part of the upstream inflow.

4.1.5 Initial conditions

The initial conditions define the depth of flow in the channel or the discharge in the channel at the beginning of the simulation period. The specification of initial conditions is often critical for establishing the numerical stability at the beginning of the simulation, particularly if there are specified water level boundary conditions in the model. In the case of downstream boundaries spilling into permanent water bodies, the water level of the water body should be specified for chainages where the bottom of the channel is lower than the downstream water level.

If the 1D channel model is being coupled to a 2D overland flow model then it is common to run the 1D model to the point where the 1D channel it is close to full but is not yet flooding over the banks. The almost full 1D channel model results can then be used as the initial conditions for the coupled 1D-2D model. This approach to defining initial conditions helps to reduce the simulation period for the coupled 1D-2D model and, thus, reduces the time required to run the coupled 1D-2D model.

4.1.6 Simulation Period and Time Step

MIKE HYDRO River does not have a steady-state solver so it always runs a fully hydrodynamic simulation. As such, it always requires the specification of a simulation period (i.e. a start date and time and an end date and time). A dynamic 'steady-state' condition is achieved by specifying constant flows and/or water levels as boundary conditions and then running the model for a sufficiently long simulation period to achieve 'steady-state' calculated flow and water levels in the model.

When coupled with a 2D model, the length of simulation period for running steady-state flows should be selected based on the flow travel time from upstream to downstream of river and also based on time taken for system to reach steady flow conditions.

The time step used for the calculations is one of the most important parameters to consider for achieving numerical stability in the model. Choosing this value should be done with care and consideration as to how it will affect the simulation. The time step should be based on several factors such as grid spacing, structures, complexity of network system etc.

As a general rule of thumb the Courant criteria can be used to estimate time step. The Courant number (Cr) describes the relation between the speed of physical disturbances in the system and the speed at which disturbances travel in the numerical model. The Courant number is calculated as shown in the following equation and the time step should be selected such that Cr number of 1.0 or less is maintained throughout the model.

$$Cr = \frac{(\sqrt{g \cdot D} + v)}{\frac{\Delta x}{\Delta t}}$$

Where:

g – gravity

D – water depth

V – velocity

Δx – grid spacing

Δt – time step

4.2 Output Settings

The major settings for output are the type of results and saving intervals. The default for a MIKE HYDRO River model is for the output to be written at each computational point unless otherwise instructed. The basic outputs from 1D river models are discharge, velocity and water level, but additional information including Froude number, Energy grade line etc. can also be generated.

The Saving interval is used to define the time interval at which the results will be written. The saving interval should consider the level of detail needed in the results, storage space, and runtime. The saving interval should be selected to give an adequate number of points to reflect the influence of the temporal inputs to the model (e.g. the shape of the computed hydrographs).

4.3 2D Model Setup – MIKE 21 HD Classic

This section describes the steps for setting up a MIKE 21 HD Classic model for 2D overland flooding. MIKE 21 divides the model setup into two sections of model inputs for a hydraulic model; Basic Parameters and Hydrodynamic Parameters. The Basic Parameters define the model setup and settings that will be used regardless of which processes are being modelled (e.g. hydrodynamics, sediment transport, waves, etc.). The Hydrodynamic Parameters describe the model inputs relevant only for the hydrodynamic model.

For the purposes of this general guidelines document, the description of the 2D model setup will include only those inputs that are relevant for a typical 2D flooding project and they will be described in a generalized way with relevant considerations and data preparation processes.

4.3.1 Model Domain

The model domain defines the horizontal extent of the 2D model. For MIKE 21 HD Classic the grid is always a rectangular shape regardless of the shape of the study area but inactive cells can be used to reduce the number of computational points in areas where overland flow does not need to be considered and, thus, the time required to run the simulation.

Ideally, in most inland flooding studies, the extent of the 2D model should be chosen such that no significant overland flooding reaches the edge of the model domain unless there is a large open water body to which it is draining. The reason for this is to capture and characterize as much of the 2D overland flooding as possible within the 2D model.

If the 2D model is being coupled to a 1D model the extents of these models do not necessary need to be the same, i.e. the 1D model can extend far outside of the 2D model domain, or it could be much smaller than the 2D model domain. The most important consideration is whether the 2D model domain is sufficiently large to contain the overland flooding from the 1D model. In many cases the flooding will spill into an urbanized floodplain and then eventually spill back into the channel at downstream locations. If this happens then the ideal situation is to make the 2D model large enough to capture the return of the flooding to the channel. In other cases, the overland flooding from the 1D model will cross over into another watershed and in this case the 2D model domain should be large enough that the flooding crossing the external boundary of the model is far enough downstream of the area of interest that it will have no meaningful impact on the results in that area.

In general, 2D model domain should be chosen such that it is sufficient larger enough to accommodate the most extreme flood event considered by the project and to ensure floodwater is not touching the edge of model domain unless above mentioned conditions are applied.

It is recommended that an initial test run should be performed with the most extreme flood event to help identify if external boundary conditions are needed and, if so, where they need to be defined.

4.3.2 Grid Cell Size and Topography

MIKE 21 HD Classic requests the user to choose a 'Bathymetry' file rather than a topography file but, for the purposes of these guidelines, the term 'topography' will be used because the majority of applications are for inland flooding studies. The Topography is defined in the Basic Parameters section of the model input.

The MIKE 21 HD Classic model does not specifically require the user to define the model grid cell size but rather it is required to define the grid cell size when generating the required 2D topography as a .dfs2 file. The .dfs2 file used for the model topography defines the domain of the model, the grid cell size and the topography of each grid cell.

As discussed in Section 2.3, the most appropriate size of the grid cells is a judgement call based on the resolution of the available topography data, the complexity of the terrain, and the time available to complete the model.

The easiest way of generating the .dfs2 file of the topography is to process the detailed 2D topography data sets using ArcGIS to merge/interpolate the data to a single raster data set, then clip the data to the desired model domain, and then export the clipped data to an ArcGIS ASCII Grid format with the desire grid cell size. Once the data is available in an ASCII grid format it can be easily converted to a .dfs2 file format using MIKE Zero Toolbox (GIS -> Grd2Mike).

Since the conversion of the ASCII Grid file does not have units, the resulting .dfs2 file also does not have units so it needs to be opened in the MIKE Zero Grid Editor where the data type and units are modified using Edit-Items. The topography data type is "Bathymetry", the unit is "meter", and the Land value is the grid cell elevation above which the grid cell will be considered inactive.

Defining Building Footprints

As mentioned in the previous paragraph, the Land Value of the topography file is used to define the grid cells that will automatically be omitted from the calculation of overland flow in the 2D model. As such any grid cell where the building polygon covers the majority of the grid cell should be assigned an elevation equal or higher than the defined Land Value of the topography file. This can be done using the Shape2Mike Tool available for download from the DHI website. The tool will use the building footprint shape file to automatically modify the topography file and assign the Land Value to each grid cell where the majority of the grid cell is overlapped by a building polygon.

4.3.3 Surface Roughness

MIKE 21 HD Classic requires the user to choose a file defining the 'Resistance' values in units of Manning's M but, for the purposes of these guidelines, the term 'surface roughness' will be used because it is a more common terminology for inland flooding studies. The Surface Roughness is defined in the Hydrodynamic Parameters section of the model input.

As described previously, Manning's M is the inverse of the more common Manning's n values used to describe riverbed roughness in hydraulic models (e.g. a Manning's n value of .05 corresponds to a Manning's M value of 20). As with the topography, MIKE 21 HD Classic requires the surface roughness values to be defined as a .dfs2 file having exactly the same extents and grid size as the topography file.

The easiest way of generating the .dfs2 file of the surface roughness is to process the land use shape file using ArcGIS to generate a raster data set, then clip the raster data to the same extents as the topography data, and then export the clipped data to an ArcGIS ASCII Grid format with the same grid cell size as the topography file. Once the data is available in an ASCII grid format it can be easily converted to a .dfs2 file format using MIKE Zero Toolbox (GIS -> Grd2Mike).

Since the conversion of the ASCII Grid file does not have units, the resulting .dfs2 file also does not have units so it needs to be opened in the MIKE Zero Grid Editor where the data type and units are modified using Edit-Items. The topography data type is "Manning's M" and the unit is "meter^(1/3)/s".

4.3.4 Eddy Viscosity

Eddy viscosity describes the effects of turbulence in 2D flow field. For inland flooding models it is usually recommended to use a constant eddy viscosity. As a rule of thumb, constant Eddy viscosity can be estimated using the following equation:

$Eddy * \Delta t/\Delta x^2 \leq 0.5$

Where:

Eddy – Eddy viscosity (m^2/s)
Δt – time step (s)
Δx^2 – cell/mesh area size (m^2)

Table 3 provides guidelines on ranges and commonly used values of Eddy viscosity for different range of computational spacing when selecting constant Eddy viscosity for MIKE 21 HD Classic model and MIKE 21 Flexible Mesh.

Table 3: Guideline for Eddy viscosity values

Dx^2 (m^2)	**Range (m^2/s)**	**Recommended (m^2/s)**
0 - 1	0 – 0.1	~ 0.05
1 - 100	0.1 - 5	~ 1
100 – 10,000	1 - 10	~ 5

4.3.5 Boundary Conditions

For MIKE 21 HD Classic the term ‘boundary conditions’ is meant to describe transfer of water across the edges of the rectangular model domain. The location of the boundary conditions are described in the Basic Parameters section of the input while the type of boundary and associated values are described in the Hydrodynamic Parameter section of the input.

The location of the boundary condition along the edge of the model is described using the j,k coordinates of the grid cells describing the starting point and ending point of the line along which the boundary condition is located. This has to be done for each side of the model domain where a boundary condition is located. If no boundary condition is defined it is assumed to be a no-flow boundary (i.e. no water can be exchanged across the boundary).

If a boundary condition is needed in a 2D flood model to convey overland flow across the boundary and out of the 2D model domain, then it is recommended to use a water level boundary condition. Although a Q-H boundary condition is available it is usually difficult to define a Q-H rating curve that is suitable and numerically stable along the entire length of the boundary condition line.

The water level along the boundary should be chosen such that it is lower than the topography of the lowest grid cell along the boundary line. Once the water level value is determined, the topography file needs to be modified to set the elevation of all the grid cells along this boundary line equal to value slightly less than the boundary condition water level. The reason for doing this is to ensure the grid cells along the boundary line remain ‘wet’ for the entire simulation period. If the cell elevations are higher than the associated boundary condition water level then the boundary condition is ignored and the cell is considered a no flow boundary.

4.3.6 Sources and Sinks

Sources and sinks are used to describe inflows and outflow within the model domain. While these are not commonly used in typical 2D flood modelling projects there may be some cases where they are needed. For example, if a fully 2D model was being used to simulate flows in a channel and inflow from a catchment or outfall needed to be accounted for then a Source could be defined at the required location with an associated discharge value or time-series.

Similar to boundary conditions, the locations of the sources and sinks are defined in the Basic Parameters section of the data tree while the type and associated values of sources and sinks are defined in the Hydrodynamic Parameters section of the data tree. The location of a source or sink is defined according to the j, k coordinate of the grid cell where it is located such that any number of sources and/or sinks can be defined but they each need to be defined individually for each grid cell.

The Source and sink description on the Hydrodynamic Parameters section can also be used to define precipitation (Source) and/or evapotranspiration (Sink) across the entire model domain. This is how a rain-on-grid model is prepared where direct rainfall on the model is used instead of hydrologic model inputs.

4.3.7 Hydraulic Structures

Hydraulic structures in a 2D model are designed to be able to simulate the hydraulic influence of structures that are typically much smaller than the model grid cells. However, if used properly and with a clear understanding of the limitations, 2D hydraulic structures can also be useful for representing the hydraulic influence of culverts, weirs, dikes and bridges in a detailed 2D overland flow model.

The most important limitation of 2D hydraulic structures is that it is only capable of describing the hydraulic influence of a structure across the grid cell faces along which it is defined. This concept is relatively easy to understand for linear structures like dikes and weirs where the flow crossing the alignment of the structure is influenced. In these cases the alignment of the physical structure is very similar to how it is defined in the model. However, it is more difficult to understand how it applies to culverts because the alignment of a physical culvert is in the direction of flow, while the alignment of the culvert structure in the model is perpendicular to the direction of flow (see Figure 8).

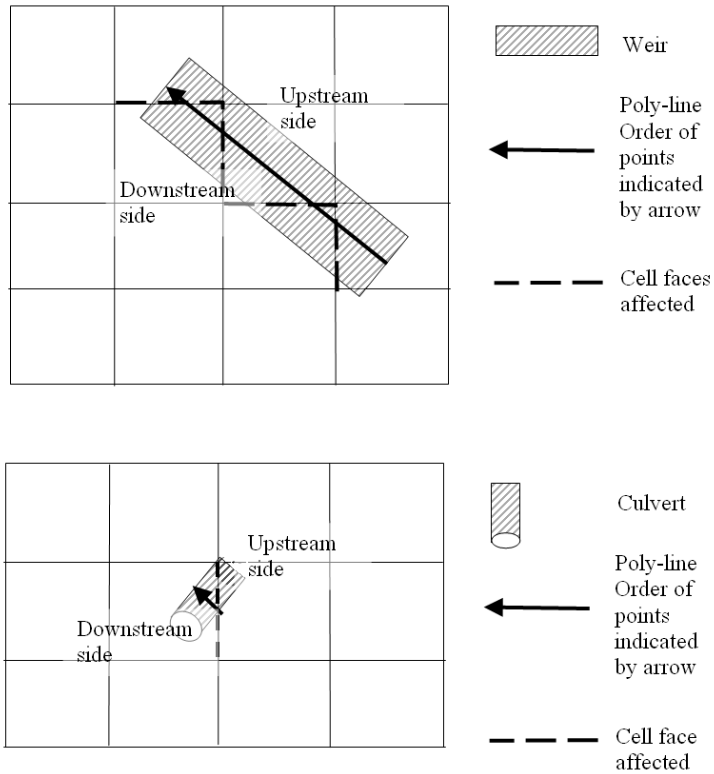


Figure 8: Schematic of 2D structure descriptions in MIKE 21 HD Classic

While the actual length of a culvert may cross many grid cells, the hydraulic influence of the culvert can only be represented as an energy loss from one grid cell to the immediately adjacent downstream grid cells. As such, it is not possible to define a 2D model culvert where flow enters at one grid cells and discharges to a point that is a distance of multiple grid cells away. While this does pose some limitations to the application of 2D model structures for long buried culverts, it is generally suitable for representing roadways crossings in a 2D channel or even representing bridge using a composite culvert and weir structure.

4.3.8 Initial Conditions

In many flood modelling studies the initial surface water elevation can be set to 0.0 (i.e. a value that is lower than the lowest topographic elevation) such that there is no water on the surface at the beginning of the simulation. However, in some situations it is appropriate to use initial surface water elevations to represent permanent water bodies, initial water levels in a channel, or water levels consistent with any water level boundary conditions.

A spatial map of initial water levels can be easily created by copying the topography .dfs2 file to an initial water level file, and then the MIKE Zero Grid Editor to modify the data type and units (Water Level, meter) and then modifying the values in the relevant areas accordingly. It is also possible to use model results from the previous run as initial conditions in a 2D model.

Note that it is important to make sure initial conditions are consistent with the outer boundary conditions in order to avoid potential numerical instabilities at the beginning of the simulation.

4.3.9 Flooding and Drying

Flooding and drying values are used to determine how the model solves for flow across grid cells. When the depth of water in a grid cell is greater than the Flooding depth, the model uses the full hydrodynamic formulation to solve for flow across the grid cells, and when the depth of water on a grid cell is less than the Drying depth there is no flow across the cell but the accumulation of water in the dry cell from adjacent wet cells is still accounted for.

For inland flooding studies in urbanized areas the typical range of values are 0.005 to 0.01 m for Drying depth, and 0.01 to 0.02 m for Flooding depth.

4.3.10 Simulation Period and Time Step

The Simulation Period is defined in the Basic Parameters section using a Starting Date and Time to define the starting time of the simulation period. The user defined values for the time step size and the number of time steps is then used to determine the End Data and Time of the simulation period. The simulation period should be defined such that it covers a period that is inclusive of all time-series files associated with the model inputs (e.g. boundary conditions, sources and sinks, time-varying topography, time-varying structure geometry).

The selection of time step should be based on the cell size whereby smaller grid cells will generally require smaller time steps, and the selection of the time step should also maintain stability criteria, i.e. Courant (Cr) Number. The Courant Number describes the relationship between the speed of physical disturbance in the system and the speed at which disturbance travels in the numerical model. In practice the Courant value should be maintained less than 1, and it is defined as following:

$$Cr = \sqrt{gh}\,\frac{\Delta t}{\Delta x} \text{ (with Cr} \leq 1)$$

Where:

g – gravity h – water depth

Δt – time step Δx – spacing in x-direction

4.3.11 Model Outputs

The MIKE 21 HD Classic model can output a variety of result data types and format. The output options include:

- Output file types: 2D map, Time series at a point, Time series along a profile
- Basic output items: water surface elevation, water depth, velocity
- Statistics: maximum values of water surface elevation, water depth, velocity and time when maximum values occur during simulation
- Saving time steps: the time step intervals at which the results will be written to the result file. The selection of saving time step should consider level of details of results needed, storage space, and runtime. Writing the results too frequently can slow down the simulation.

Some of the post-processing tools (e.g. MIKE 21 Discharge Calculator, Flood Modelling Toolbox) require the result file to be in a strict HPQ format, meaning it only contains the water depth (H), the P flux and the Q flux. As such, it is good practice for final model runs to always generate one file containing only the HPQ values, and another containing all of the result types needed for mapping.

4.4 2D Model Setup – MIKE 21 Flexible Mesh

This section describes the steps for setting up a MIKE 21 Flexible Mesh model for 2D overland flooding. The model setup is structures such that there is are global settings applied to the model regardless of what type of processes are being simulated (e.g. hydrodynamics, waves, sediment transport, water quality, etc.) and then modules containing inputs and settings associated with each process. For an overland flooding model the Hydrodynamic Module contains all of the hydrodynamics related model inputs and settings.

For the purposes of this general guidelines document, the description of the 2D model setup will include only those inputs that are relevant for a typical 2D flooding project and they will be described in a generalized way with relevant considerations and data preparation processes.

4.4.1 Model Domain

The model Domain is a global setting for MIKE 21 Flexible Mesh and the main input for the model Domain is the Mesh file (.mesh). The Mesh file defines the horizontal extent of the 2D model and it contains the configuration of the flexible mesh, the elevations of each mesh node, and the locations of boundary conditions.

As the 'flexible mesh' name alludes to, the model domain can be any shape and there can be multiple disconnected model areas within the same model. Ideally, in most inland flooding studies, the extent of the 2D model should be chosen such that no significant overland flooding reaches the edge of the model domain unless there is a large open water body to which it is draining. The reason for this is to capture and characterize as much of the 2D overland flooding as possible within the 2D model.

If the 2D model is being coupled to a 1D model the extents of the models do not necessarily need to be the same, i.e. the 1D model can extend far outside of the 2D model domain, or it could be much smaller than the 2D model domain. If the 1D model extends outside of the 2D model domain then the location where the 2D model starts and ends should, ideally, be chosen at a location where the flow in the 1D model is entirely contained in the channel for all flooding conditions. The reason for this is that it avoids the complexity of defining a 2D model boundary condition to convey upstream overland flooding into the 2D model domain.

Another important consideration is whether the 2D model domain is sufficiently large to contain the overland flooding from the 1D model. In many cases the flooding will spill into an urbanized floodplain and then eventually spill back into the channel at downstream locations. If this happens then the ideal situation is to make the 2D model large enough to capture the return of the flooding to the channel rather than spilling across an otherwise dry model boundary.

In other cases the overland flooding from the 1D model will cross over into another watershed and in this case the 2D model domain should be large enough that the flooding crossing the external boundary of the model is far enough downstream of the area of interest that it will have no meaningful impact on the results in the area of interest.

In general, regardless of whether it is a fully 2D model or a coupled 1D-2D model, for inland flooding studies the 2D model domain should be chosen such that it is sufficient larger enough to accommodate the most extreme flood event considered by the project such that flooding does not reach the boundary of the model unless that boundary is a permanent water body.

It is recommended that an initial test run should be performed with the most extreme flood event to identify if external boundary conditions are needed and, if so, where they need to be defined.

4.4.1.1 Flexible Mesh Design

The development of the flexible mesh requires the consideration of the model domain, the location and extents of significant flow channels within the model domain, the outer model boundary condition locations, and the topography – including the location of significant surface drainage features such as roadways, ditches and berms.

The flexible mesh allows the model domain to be discretized into a mixture of triangular and quadrangular elements of varying sizes whereby some areas of the model can have very small element sizes where a high level of details is required in the results and other areas of the model can have much large mesh elements where less detailed data is available or the results are not as spatially variable.

The flexible mesh is developed using the MIKE Zero – Mesh Generator tool for defining the model domain(s), defining the mesh type and sizes throughout the sub areas of the model domain (defined by individual polygons), assigning the boundary condition locations (using Code values of 2 or greater), and interpolating multiple topography data sources to the mesh nodes. The resulting mesh file (*.mesh) created by the Mesh Generator tool contains a description of the mesh configuration, boundary condition locations, and the elevation at each mesh node.

In Mesh Generator accepts the following elevation data formats:

- XYZ file (*.xyz) – a text file containing X, Y coordinates and associated elevation values (Z)
- Dfs2 file (*.dfs2) – a binary, gridded DHI file structure containing 'Elevation' or 'Bathymetry' values for each grid cell
- Dfsu file (*.dfsu) – a binary DHI file for an 'unstructured' mesh containing 'Elevation' or 'Bathymetry' values
- Mesh file (*.mesh) – an ascii DHI file containing the mesh configuration, boundary condition locations, and elevation values at each mesh node.

Channels

If the MIKE 21 Flexible Mesh model is being coupled to a 1D MIKE Hydro River model then the mesh should be design such that the area occupied by the 1D channel model is omitted from the 2D flexible mesh. Ideally, the edge of the flexible mesh along the bank of the channel should just slightly extend inside the channel to avoid potential problem with lateral links being coupled to unmeshed areas. Figure 9 shows an example of a flexible mesh where the area occupied by the 1D channel model is not included in the mesh.

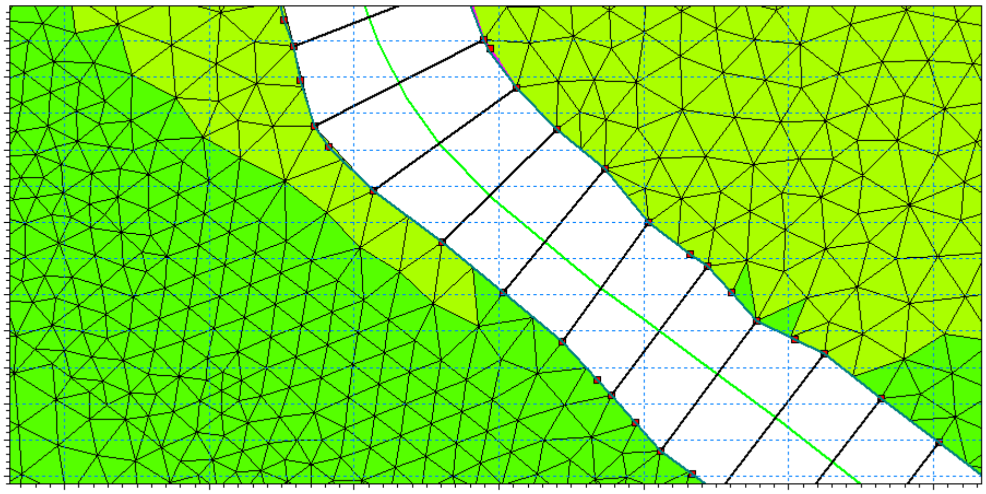


Figure 9: Example of flexible mesh with unmeshed area inside the 1D channel

Alternatively, if the channels will be included as part of the 2D model then consideration should be given to representing the channel using elongated quadrangular elements. These elements are well suited for situations where the direction of flow is well known as the elongated elements allow fewer elements to be required while still providing a detailed representation of the cross-sectional geometry of the channel (see Figure 10).

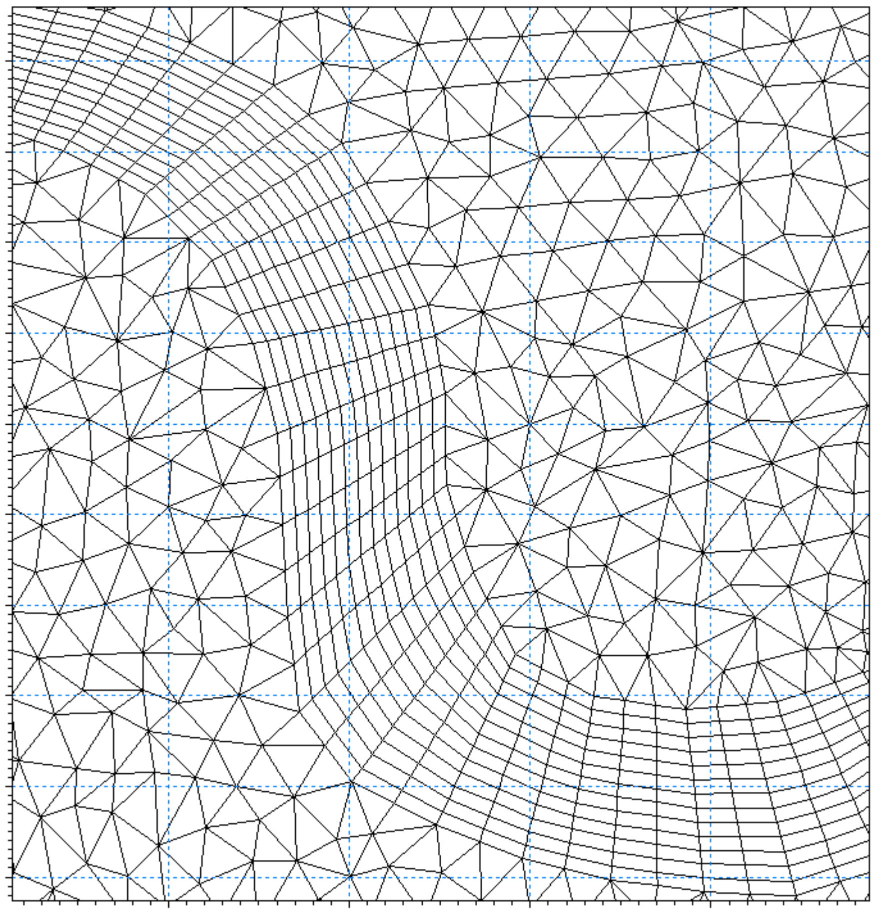


Figure 10: Example of 2D channel represented using quadrangular elements

Buildings and Roadways

Shape files of relevant polygons (e.g. roadways, buildings, berms) and lines (walls, ditches) can be incorporated into the mesh by converting them to the required boundary line file format (.xyz) using the MIKE Zero Toolbox – Shp2Xyz tool. Once the polygons are converted to the .xyz file they can be imported to the Mesh Generator as arcs and polygons.

As mentioned previously, the building footprint shape files should first be processed using ArcGIS tools to simplify the building shapes to avoid the generating very small mesh elements around small irregularities in the shape (see example in Figure 11).

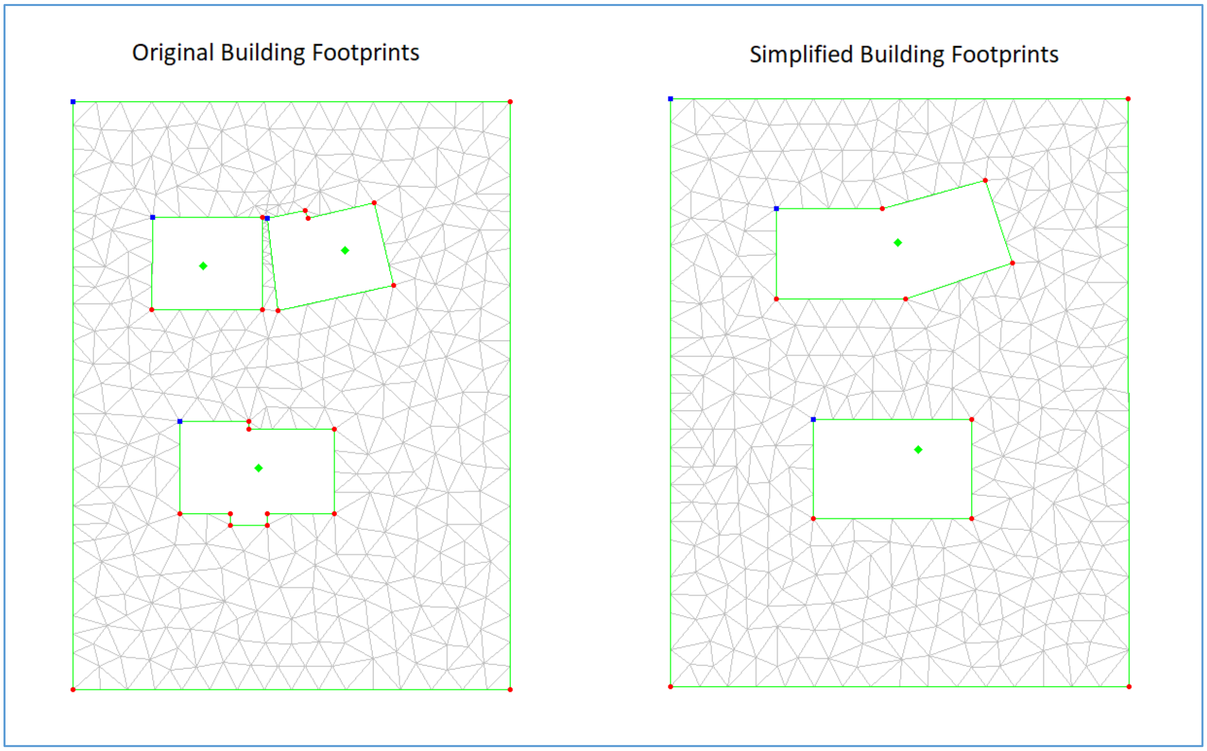


Figure 11: Comparison of mesh with original vs. simplified building shapes

Berms and Drainage Ditches

For flooding studies in urbanized areas the extent of flooding is likely to be significantly influenced by roadways where even relatively subtle changes in grade can have a meaningful impact on the extents of flooding and the direction of overland flow. As such it is desirable to represent the roadways with a small enough mesh resolution to allow the influence of the curbs and the crown of the roadway to be reflected in the results. A maximum mesh area of 10 m^2 has proven to be effective for this purpose.

For representation of flood mitigation berms and/or drainage ditches it is important to understand how the model determines flow from one element to the next in order to properly design the mesh. In the flexible mesh model the flow is calculated across the face of each mesh element and the direction of flow is determined by comparing the topographic elevation plus the depth of water in each element (for the purpose of this explanation the influence of momentum is ignored). The elevation of the element is calculated by taking the average of elevations at each element node, and the depth of water is calculated by taking the average calculated depth at each element node. As such, if the mesh is designed in such a way that the crest of a berm is associated to only a single line of mesh nodes, the model will represent overtopping of the berm before the water level reaches the actual crest elevation of the berm. This concept is illustrated in the schematic presented in Figure 12.

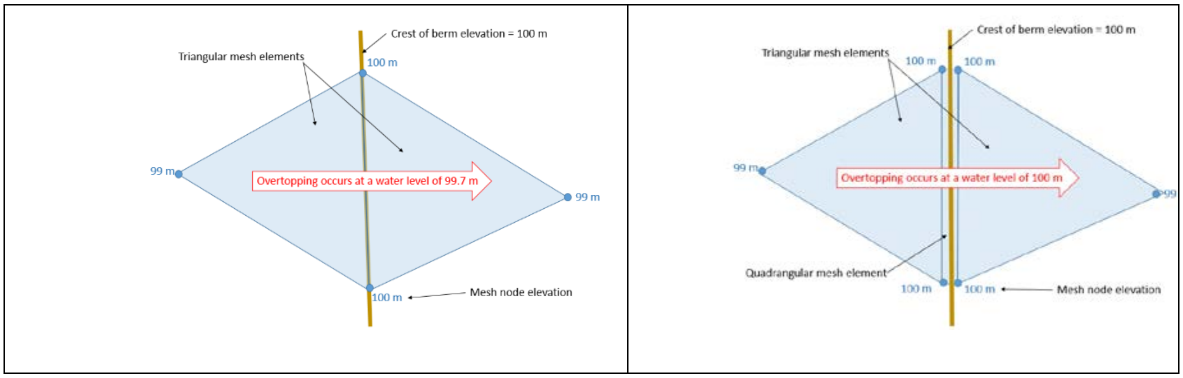


Figure 12: Schematic representation of a mesh representing a berm

In order to ensure the crest of a berm or the trough of a ditch are well represented it is important to ensure the mesh is detailed enough to represent the crest elevation in the elements and not just in the nodes in order to misrepresent overtopping of a berm or under-estimate the conveyance capacity of a ditch. The required resolution is dependent upon the dimensions of the berm or ditch but, in practice, a maximum mesh element area of 5 m^2 in the near vicinity of these features has shown to be effective. Alternatively, a row of narrow quadrangular elements can be defined along the crest of the berm (or trough of the ditch) in order to ensure the crest elevation is represented in the elements (see Figure 12).

Culverts, Bridge, Dikes and Weirs

If 2D model structure are going to be used to represent culvert road crossings, dikes, bridges, weirs or any other structure that will influence flows then some customization of the mesh is likely required at the location of the structure in order to ensure it is properly represented. This will be covered in Section 4.4.9.

4.4.2 Time

The Time of the simulation is a global setting that defines the Starting Date and Time of the simulation and then uses a Time step interval and the Number of time steps to determine the End Date and Time. The simulation period should be defined such that it is inclusive of all time-series files associated with the model inputs (e.g. boundary conditions, sources and sinks, time-varying topography, time-varying structure geometry). That is to say, all contributing time-series in the model setup must, at a minimum, have data covering the entire simulation period.

The time step interval used to define the simulation period is not necessarily the time step used in the solution but, rather, it determines the minimum time interval for which output can be obtained. The solution time step is adaptive whereby the user defines the minimum and maximum time step to be used (see next section on Solution Technique). As such, the maximum solution time step used must be equal to or smaller than the time step interval used to define the simulation period. A time step interval value of 10 seconds is usually a good value to start with as it is small enough to still get a detailed time-series but large enough that it won't unnecessarily restrict the solution time step.

4.4.3 Solution Technique

The Solution Technique is related to the Hydrodynamics Module and it describes the options and settings for modifying the solution used to solve for flow and water levels in the model. The solution technique for inland flood modelling and mapping purposes should always be the Higher Order solution. While it usually takes longer to solve, it provides a more accurate solution.

The selection of the Maximum time step should be based on the mesh size where smaller mesh elements usually requires smaller time steps. The selection of the Maximum time step should also maintain stability criteria, i.e. Courant-Friedrich-Lévy (CFL) number. The CFL number describes the relationship between the speed of physical disturbance in the system and the speed at which disturbance travel in the numerical model, which in practice should be maintained less than 1, and it is defined as following:

$$CFL = \left(\sqrt{gh} + |u|\right)\frac{\Delta t}{\Delta x} + \left(\sqrt{gh} + |v|\right)\frac{\Delta t}{\Delta y} \text{ (with CFL < 1)}$$

Where:

g – gravity	h – water depth	Δt – time step
Δx – spacing in x-direction	Δy – spacing in y-direction	
u – velocity in x-direction	v – velocity in y-direction	

4.4.4 Flooding and Drying

For a MIKE 21 Flexible Mesh model the Wetting, Flooding and Drying values are used to determine how the model solves for flow across the mesh elements. When the water depth in an element is less than the Drying depth the element is removed from the calculation. When the water depth is greater than the Flooding depth but less than the Wetting depth the element is re-entered into the calculation using a reformulated equation where the momentum fluxes are set to zero and only the mass fluxes re taken into consideration.

For inland flooding models it is recommended to use the Advanced Flooding and Drying (Floodplain) option. When "Advanced flood and dry (floodplain)" is selected the momentum equation is suppressed as the water depth tends to the wetting depth. The suppression starts at two times the wetting depth. Additionally, the bed resistance is treated implicitly by calculating the bed resistance source term based on the solution estimated at the new time step. Finally, a correction of the velocities/fluxes is applied when the CFL number, estimated based on the calculated solution at the new time step, becomes larger than 1. In this case the velocities/fluxes are reduced so that the CFL number becomes less than 0.5.

For inland flooding studies in urbanized areas the typical range of values are 0.005 to 0.01 m for Drying depth, 0.01 to 0.02 for Flooding depth, and 0.02 to 0.05 for Wetting depth.

4.4.5 Eddy Viscosity

Eddy viscosity describes the effects of turbulence in 2D flow field. For inland flooding models it is usually recommended to use a constant eddy viscosity. As a rule of thumb, constant Eddy viscosity can be estimated using the following equation:

$Eddy * \Delta t/\Delta x^2 \leq 0.5$

Where:

Eddy – Eddy viscosity (m^2/s)
Δt – time step (s)
Δx^2 – cell/mesh area size (m^2)

Table 4 provides guidelines on ranges and commonly used values of Eddy viscosity for different range of computational spacing when selecting constant Eddy viscosity for MIKE 21 HD Classic model and MIKE 21 Flexible Mesh.

Table 4: Guideline for Eddy viscosity values

Dx^2 (m^2)	**Range (m^2/s)**	**Recommended (m^2/s)**
0 - 1	0 – 0.1	~ 0.05
1 - 100	0.1 - 5	~ 1
100 – 10,000	1 - 10	~ 5

4.4.6 Bed Resistance

MIKE 21 Flexible Mesh requires the user to choose a file defining the 'Resistance' values in units of Manning's M but, for the purposes of these guidelines, the term 'surface roughness' will be used because it is a more common terminology for inland flooding studies. The Surface Roughness is defined in the Hydrodynamic Module of the model input.

As described previously, Manning's M is the inverse of the more common Manning's n values used to describe riverbed roughness in hydraulic models (e.g. a Manning's n value of .05 corresponds to a Manning's M value of 20). MIKE 21 Flexible Mesh allows the surface roughness values to be defined as either a dfs2 file covering the extents of the model domain, or a dfsu file with the exact same mesh configuration as the model domain. Since the model domain mesh configuration is prone to changing frequently during the model setup, the dfs2 file is usually the most convenient format to use.

The easiest way of generating the dfs2 file of the surface roughness is to process the land use shape file using ArcGIS to generate a raster data set, then clip the raster data to the same extents as the topography data, and then export the clipped data to an ArcGIS ASCII Grid format with the same grid cell size as the topography file. Once the data is available in an ASCII grid format it can be easily converted to a dfs2 file format using MIKE Zero Toolbox (GIS -> Grd2Mike).

Since the conversion of the ASCII Grid file does not have units, the resulting dfs2 file also does not have units so it needs to be opened in the MIKE Zero Grid Editor where the data type and units are modified using Edit-Items. The surface roughness data type is "Manning's M" and the unit is "meter^(1/3)/s".

4.4.7 Precipitation and Evaporation

Precipitation can be used to create a 'rain-on-mesh' model where precipitation is applied across the entire surface of the model using either a constant rate, a time-series uniformly applied over the entire model domain, or a spatially and temporally distributed precipitation rate applied over the entire model domain. Similarly, Evaporation can be used to remove water (if available) from the surface of the model.

Niether of these are used in a typical inland flooding study because hydrology models are more commonly used than rain-on-mesh models to generate flows in river channels, while the time periods and extreme flows considered for flood models usually make the influences from Evaporation processes negligible.

4.4.8 Sources

Sources and sinks are used to describe inflows and outflow within the model domain. While these are not commonly used in typical 2D flood modelling projects there may be some cases where they are needed. For example, if a fully 2D model was being used to simulate flow in a channel and inflow from a catchment or outfall needed to be added to the channel then a Source could be defined at the required location with an associated discharge value or time-series.

In MIKE 21 Flexible Mesh model the sources and sinks are added in the Sources section of the Hydrodynamic Module. The Sources require the x, y coordinates of the source location and a specification of the rate. A positive rate adds flow to the model while a negative rate removes flow from the model. Any number of sources can be defined but they each need to be defined individually.

If a source adds or removes flow at a very large rate compared to the size of the mesh elements then it may cause some numerical instabilities due to see hydraulic gradients. In this case it should be considered to split the source into several source items spread over the nearby elements.

4.4.9 Hydraulic Structures

Hydraulic structures in a 2D model are designed to be able to simulate the hydraulic influence of structures that are typically much smaller than the model grid cells. However, if used properly and with a clear understanding of the limitations, 2D hydraulic structures can also be useful for representing the hydraulic influence of culverts, weirs, dikes and bridges in a detailed 2D overland flow model.

The most important limitation of 2D hydraulic structures is that it is only capable of describing the hydraulic influence of a structure across the faces of the mesh elements along which it is defined. This concept is relatively easy to understand for linear structures like dikes and weirs where the alignment of the physical structure is very similar to how it is defined in the model. However, it is more difficult to understand how it applies to culverts because the alignment of a physical culvert is in the direction of flow, while the alignment of the culvert structure in the model is perpendicular to the direction of flow (see Figure 13).

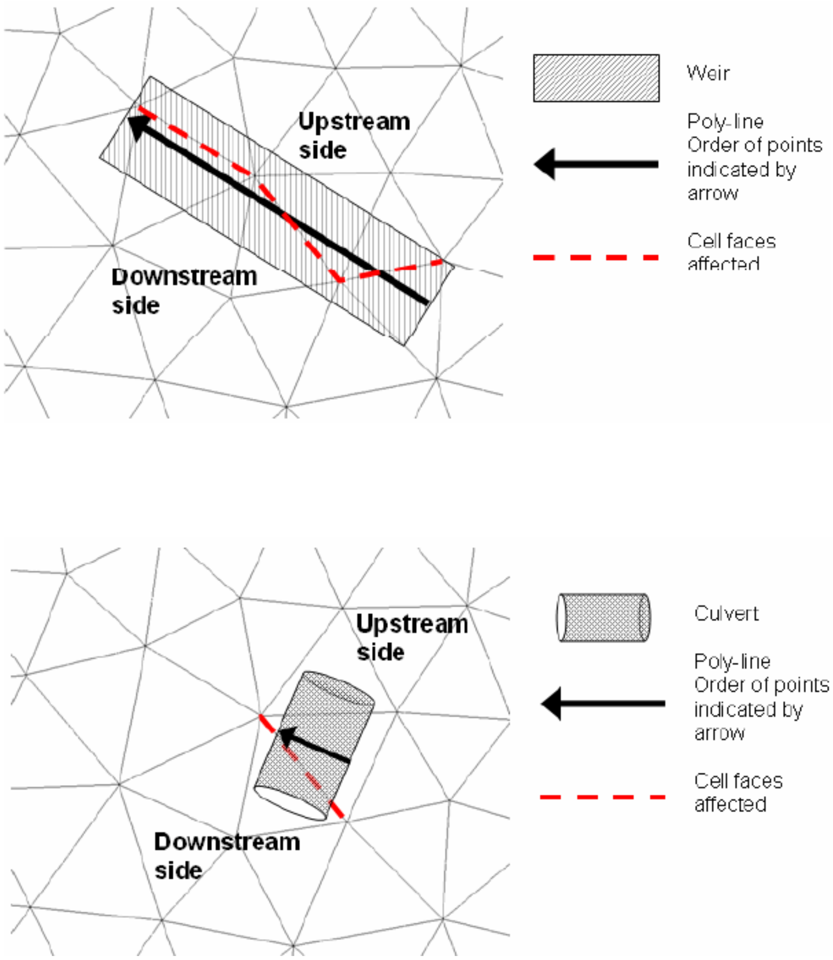


Figure 13: Schematic of 2D structure descriptions in MIKE 21 Flexible Mesh

4.4.9.1 Culverts

While the physical location of a culvert may cross many mesh elements, the hydraulic influence of the culvert can only be represented as an energy loss across the affected element face(s) (see dashed line in lower image of Figure 13). As such, it is not possible to define a 2D model culvert where flow enters at one element and discharges to a point that is a distance of multiple elements away. For the example shown in Figure 13, the culvert will affect flow passing across the affected cell face (dashed red line) whereby the flow through the culvert will be based on the calculated upstream water level in the element on the right side of the affected face and the calculated downstream water level in the element on the left side of the affected face. Another way of visualizing this is provided in Figure 14. While this does pose some limitations to the representation of long, buried culverts, it is still suitable for representing roadways crossings in a 2D channel using a composite culvert and weir structure. However to support 2D culvert road crossing the mesh and topography will need to be modified to accommodate the limitations noted above.

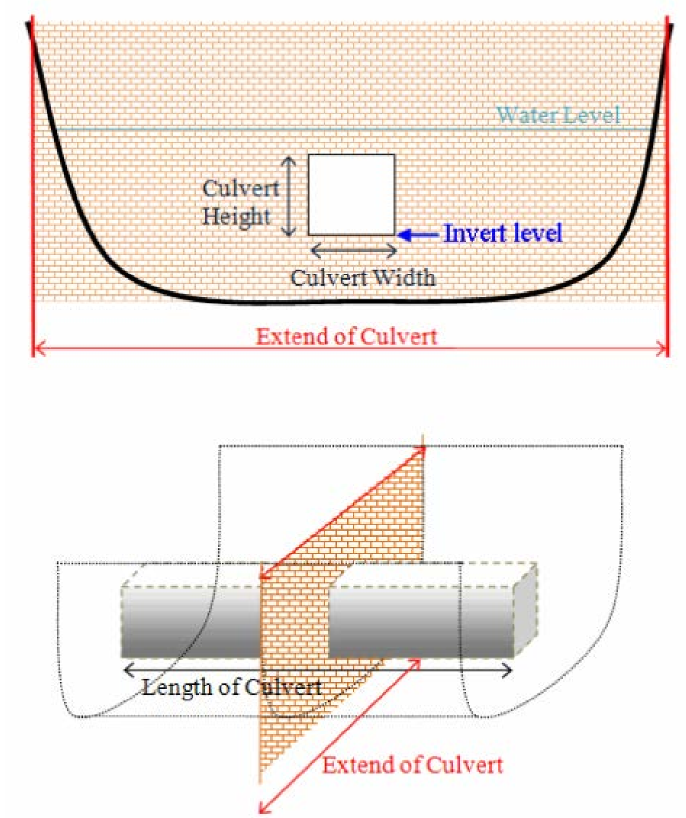


Figure 14: Schematic representation of a 2D model culvert structure

Firstly, a channel needs to be burned through the roadway at an elevation equal to the lowest invert of the culvert in order to allow flow to pass from one side of the culvert to the other. Then the mesh needs to be detailed enough in the burned channel such that there is a row of elements along the channel where each element has an elevation equal to the lowest invert of the culvert. This approach is illustrated in Figure 15 where it shows the original elevated roadway in the image on the left side and the modified topography and mesh in the image on the right side.

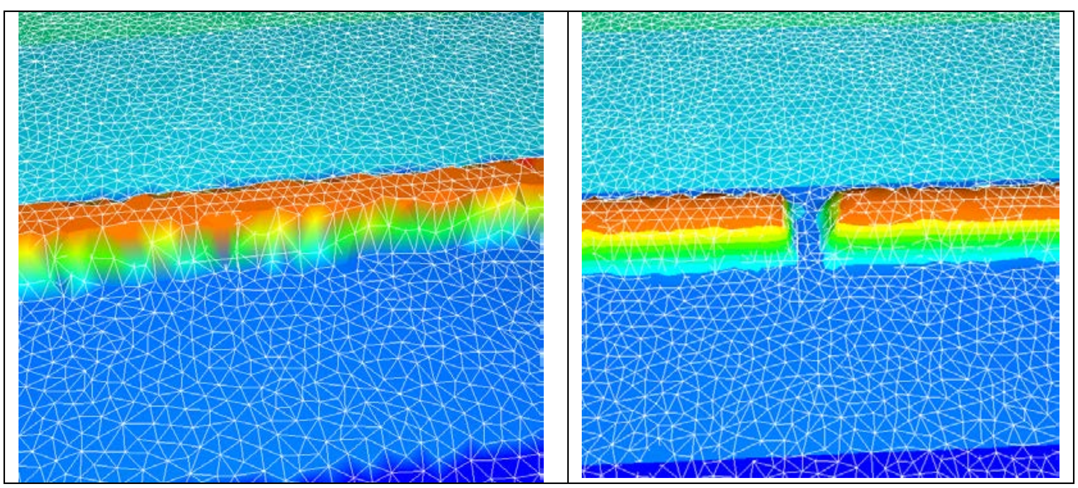


Figure 15: Modified topography and mesh for 2D culverts at road crossings

Once the mesh and topography have been modified, the culvert structure can be defined across the burned channel from the top of the roadway on either side of the channel (see Figure 16). This creates a vertical wall across which flow can only pass via the 2D culvert structure (see also the bottom image in Figure 14).

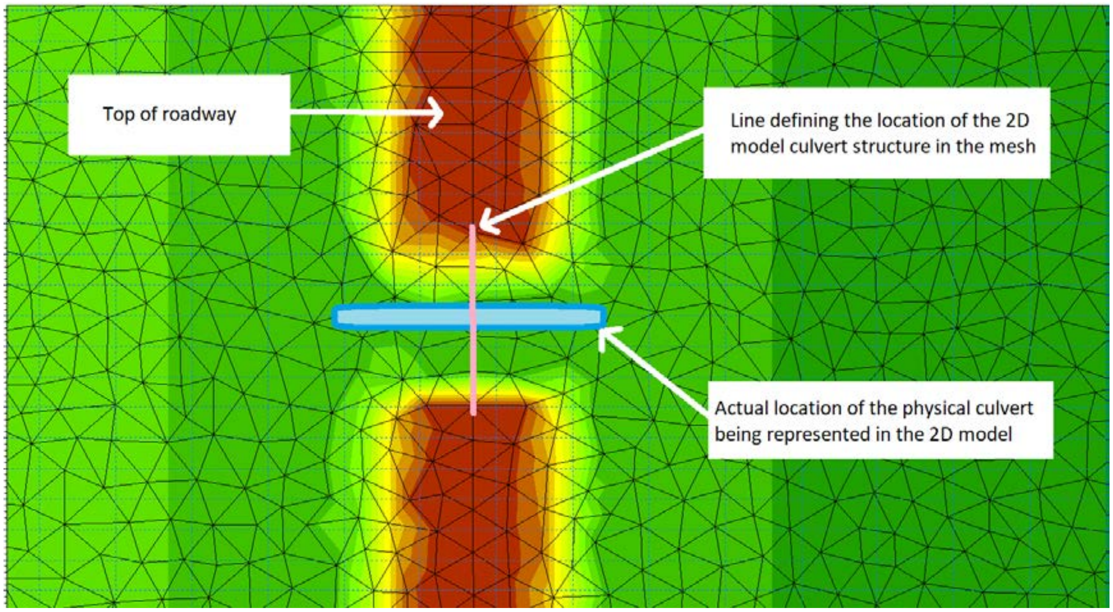


Figure 16: Example of a 2D model culvert structure representing a culvert road crossing

In the example shown above the line defining the 2D culvert structure is drawn across multiple mesh elements so the calculation of flow through the culvert structure is based on the average water level of the elements on the upstream side of the line vs. the average water level of the elements on the downstream side of the line. In addition, if the 'non-uniform' flow distribution option is chosen the flow through the culvert will be distributed to the downstream mesh elements according the topographic elevation of each element (i.e. lower elements receive more flow).

If a culvert roadway cross is represented using only the 2D culvert structure then any potential overtopping of the roadway will have to occur around the edges of the line defining the 2D model culvert structure because the vertical wall extends up infinitely high (see bottom image in Figure 14). In order to allow for overtopping of the roadway at the culvert location a composite culvert and weir structure is required. In this case a weir structure should be defined along exactly the same line as the culvert and the crest level of the weir should represent the top of the roadway. In this case the water will initially flow through the culvert until the water level on the upstream side of the line exceeds the crest level of the weir, at which point it will begin calculating flow over the weir and into the downstream mesh elements.

4.4.9.2 Bridges

Although there is no specific 2D structure type for bridges the hydraulic influence of the bridge structures can be effectively represented in the model using culverts and weirs in a manner that is similar to that described for culvert roadway crossings. However, in the case of a bridge the shape of the culvert structure would be set as Irregular and the shape would be described using a level-width curve. If there are multiple openings then a composite culvert structure can be defined with multiple culverts defined at the same location.

4.4.10 Initial Conditions

In many flood modelling studies the initial surface water elevation can be set to 0.0 (i.e. a value that is lower than the lowest topographic elevation) such that there is no water on the surface at the beginning of the simulation. However, in some situations it is necessary to use initial surface water elevations to represent permanent water bodies, initial water levels in a channel, and/or water levels consistent with any water level boundary conditions.

A spatial map of initial water levels can be easily created by generating a .dfs2 file of the topography and then renaming it to an initial water level file, and then use the MIKE Zero Grid Editor to modify the data type and units ('Water Level', 'meter',) and then modify the initial surface water elevation values accordingly. It is also possible to use model results from the previous run as initial conditions in a 2D model but the mesh configuration needs to be exactly the same in both models.

Note that it is important to make sure initial conditions are consistent with the outer boundary conditions in order to avoid potential numerical instabilities at the beginning of the simulation.

4.4.11 Boundary Conditions

For MIKE 21 Flexible Mesh the term 'boundary conditions' is meant to describe transfer of water across the edges of the model domain. The location of the boundary conditions are described in the Mesh file while the type of boundary condition and associated values are described in the Hydrodynamic Parameter section of the input. If no boundary condition is defined it is assumed to be a no-flow boundary (i.e. no water can be exchanged across the boundary).

If a boundary condition is needed in a 2D flood model to convey overland flow across the boundary and out of the 2D model domain, then it is recommended to use a water level boundary condition. Although a Q-H boundary condition is available it is usually difficult to define a Q-H rating curve that is suitable and numerically stable along the entire length of the boundary condition location.

The water level along the boundary should be chosen such that it is lower than the topography of the lowest mesh element along the boundary condition location. Once the water level value is determined, the Mesh file needs to be modified to set the elevation of all the nodes along this boundary line equal to a value slightly less than the boundary condition water level. The reason for doing this is to ensure the mesh elements along the boundary line remain 'wet' for the entire simulation period. If the mesh elevations are higher than the associated boundary condition water level value then the boundary condition assignment is ignored and it is considered a no-flow boundary.

4.4.12 Model Outputs

MIKE 21 Flexible Mesh can output a variety of result data types and formats. The output options include:

- 2D horizontal: Produces result files in a variety of formats)
  - Formats:
    - Point series (time series of values at a point)
    - Line series (time series of values along a line)
    - Area series (time series of values within an area)
  - Items (Variables):
    - Surface elevation
    - Water depth (Total, Still)
    - Velocity (U-velocity, V-velocity, Current Speed)
    - Flux (P-flux, Q-flux)
    - Others
  - Time steps:
    - First (time step when model starts saving results to a file)
    - Last (time step when model stops saving results to a file)
    - Frequency (number of ***time steps*** between saving of results)
- Mass Balance: Generates a time series of mass balance for the hydrodynamic simulation
- Discharge: Produces a time-series of discharge across a line defined by two x, y coordinates
- Inundation: Produces a 2D map of result statistics including:
  - Maximum water depth
  - Time of maximum water depth
  - Maximum current speed
  - Time of maximum current speed
  - Duration of depth above threshold

When determining how often to save the results the resultant size of the result file should be considered along with the fact that saving results more often will slow down the simulation. Similarly, if Inundation statistics are needed, they should only be run for the very final production run of the model since it also considerably impacts the speed of the solution.

5 1D-2D Coupled Model

In order to enable the exchange of flows between the 1D model and the 2D model the two models need to be coupled together. The coupling describes the locations in each model where the flow will be exchanged as well as how it will be exchanged. MIKE FLOOD provides a variety of coupling options as described in Section 2.2.4 and listed again below.

1. Lateral Link: Describes the coupling along the banks of the channel to the 2D model.

2. Standard Link: Describes the coupling at the upstream or downstream end of the 1D river model to the 2D model.
3. Structure Link: Describes the coupling of a 1D structure element (e.g. culvert) to the 2D model.

The approach and methodology used to exchange flows between the models is described in Section 2.2.4, so this section will be somewhat repetitive but will focus mainly on the steps for setting up the coupled model.

5.1 Configuring MIKE FLOOD

MIKE FLOOD is a tool used to define the coupling locations where the 1D and 2D models will exchange flow, and settings describing how the flow will be exchanged at each coupling location. In order to start, the 1D river model and the 2D overland flow model must be loaded into a MIKE FLOOD project by defining the Linkage Files. For the purposes of this document, only the 1D River Model and the 2D Surface Model will be discussed. MIKE FLOOD can support either a MIKE HYDRO River model setup file or a MIKE 11 model setup file for the 1D River Model. Similarly, MIKE FLOOD supports either a MIKE 21 HD Classic model or a MIKE 21 Flexible Mesh model for the 2D Surface Model.

Once the 1D River Model and 2D Surface Model are loaded in MIKE FLOOD the linkages (coupling locations) can be defined using the Link Definitions editor where a map view will display the 2D model bathymetry and 1D model network and cross-sections. Linkages can be defined by right-clicking on the map and choosing one of the applicable options (e.g. “Link river branch to MIKE 21”) to load an editor window where the specific location of the linkage is defined (see example in Figure 17).

Link Types and associated settings are described in the user manual so this section of the guidelines will focus on providing suggestions on how to approach the coupling definitions for different situations.

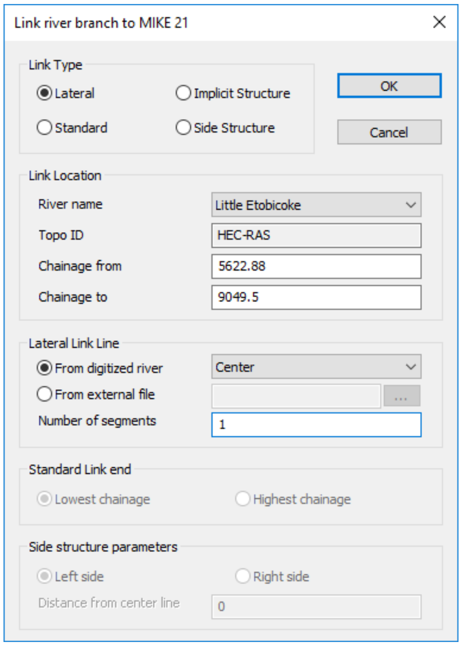


Figure 17: 1D River to 2D Surface Model Linkage Options editor

5.2 Lateral Links

The Lateral Link defined a line of 2D grid cells/mesh elements that are linked to a branch, or a section of a branch, in the 1D River model. When coupling to a MIKE 21 HD Classic model the lateral links are defined by X,Y coordinates of the Lateral Link line vertices as well as the j, k coordinates of the linked grid cells. When coupling to a MIKE 21 Flexible Mesh model the lateral links are defined by only the X,Y coordinates of the lateral link line vertices.

Although it was described previously, it is important to understand how MIKE FLOOD automatically generates the Lateral Link line. The steps used by MIKE FLOOD are as follows:

- Linearly interpolates cross-section data to generate a cross-section at each digitized vertice of the line defining the selected branch
- Digitizes a line along the left / right bank of the channel connecting all Marker 1/Marker 3 locations on each cross-section, including the interpolated cross-sections.
- Intersects the line with the 2D surface model to define the grid cells that intersect the Lateral Link line(s).

Alternatively, there is an option to define link lines using an external file that can be either a polyline shape file (containing one single continuous polyline) or a xyz ASCII file.

It is important to understand the mechanics for how the Lateral Link line is generated because it helps to understand some anomalies that may appear when the Lateral Link line is closely examined. Figure 18 shows and example of a situation where the river centerline bends sharply in a few locations because the line is defined by closely spaced vertices along the bends. An interpolated cross-section is created at each vertex of the line and this results in some of the interpolated cross-sections overlapping each other. Since the automatically generated Lateral Link line simply joins the bank marker locations on each cross-section (including interpolated cross-sections) in sequential order according to the chainage location, the Lateral Link line ends up looping back on itself in a few locations. This issue can be resolved by either reducing the detail represented in the centreline (i.e. using fewer vertices and spacing them further apart) or, if that is not possible, manually modifying the Lateral Link line using the tools available in MIKE FLOOD to:

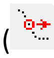
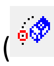
- Insert points ()
- Move points ()
- Delete points ()
- Select points ()

Figure 19 shows an example of the Lateral Link lines after being manually fixed.

> **Note:** If the 2D Surface Model is a MIKE 21 HD Classic model then the cell indices need to be remapped after the Lateral Link line has been adjusted.

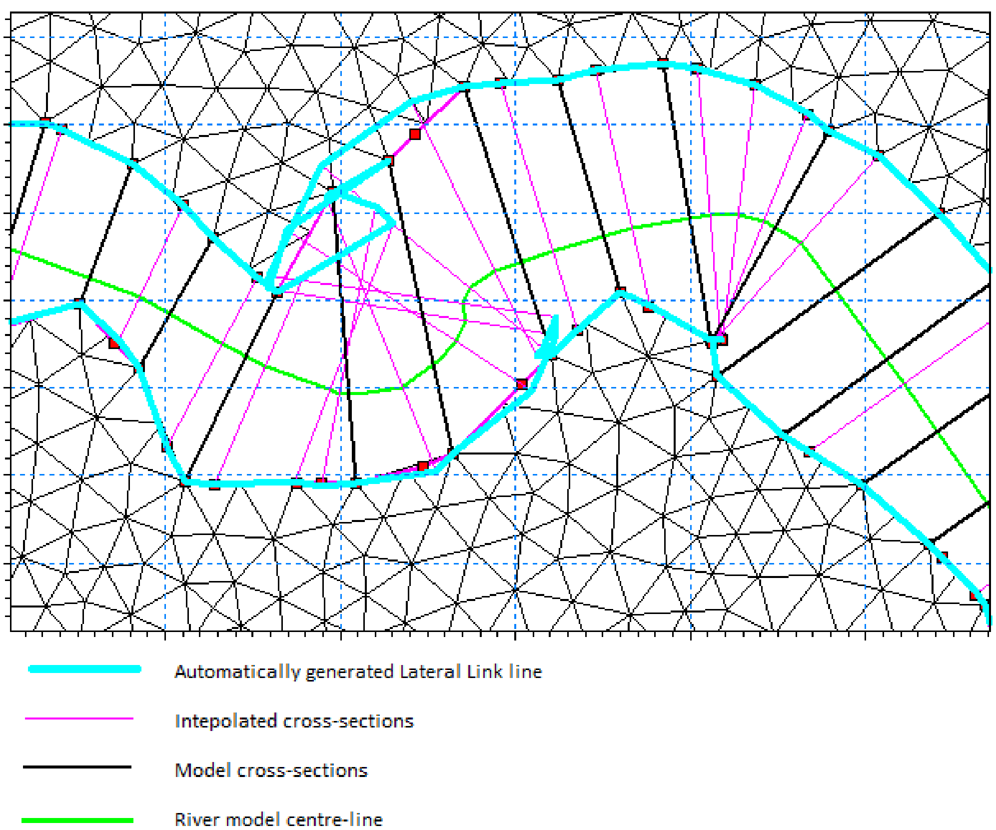


Figure 18: Example of Poorly Formed Lateral Link Lines

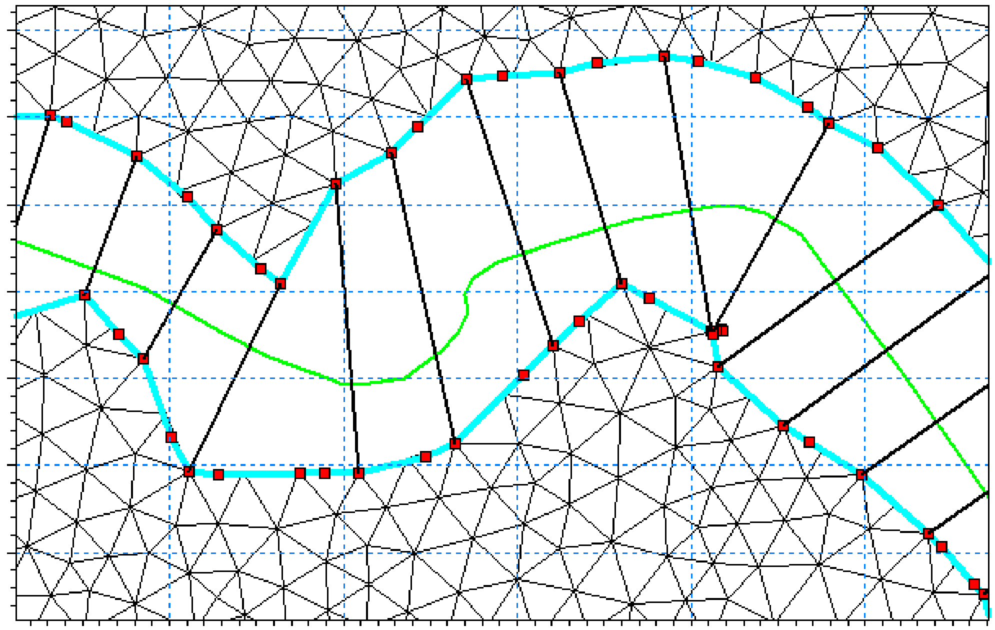


Figure 19: Example of Manually Fixed Lateral Link Lines

5.2.1 Linking Across Tributaries

When defining the chainage interval of the Lateral Link, as well as selecting which side of the channel to generate the link (e.g. left, right, center, or left and right), it is important to consider whether there are any tributaries along the length of the channel that will be coupled.

If the branch being coupled is intersected by a tributary that is also being coupled, then the Lateral Link line should not cross the tributary since the 2D model will not exist within the tributary river channel. In this case the left bank and right bank Lateral Link lines should be defined separately whereby one side is continuous along the entire branch while the side with the tributary should be divided into two separate Lateral Link lines such that it does not intersect the 1D channel of the tributary branch (see Figure 20).

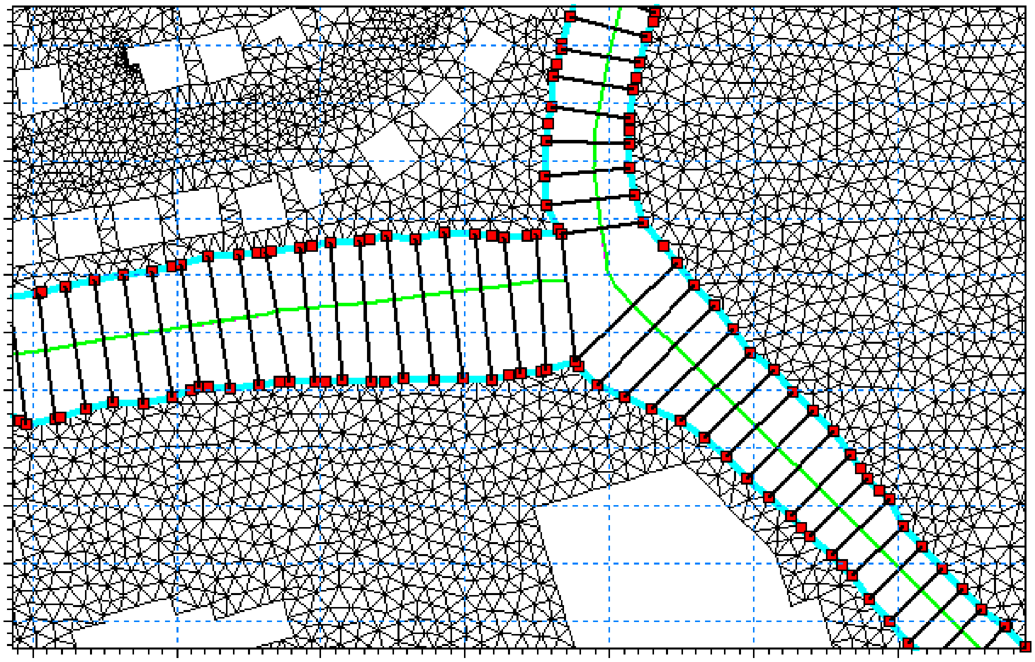


Figure 20: Example of Lateral Link Lines at the Confluence of Two Channels

5.2.2 Linking Across Structures

The handling of Lateral Links across structures is a topic of some debate because there are a few different ways to approach it and each one has it's merits and drawbacks. To-date, there has not been a detailed study to evaluate each option, compare the advantages and disadvantages, and make a recommendation. Until this has been done, it is important to understand the different options available and the potential implications of each. The available options include:

1. Represent the entire crossing as a 1D channel with Lateral Links defined along both sides of the channel.
2. Represent the entire crossing as a 1D channel with no Lateral Links defined adjacent to the structure.
3. Represent the crossing surface in the 2D model with no Lateral Links defined adjacent to the structure.

For the purposes of this discussion, the crossing structure can either be represented in the 1D model as a bridge, or as a composite culvert and weir structure.

5.2.2.1 Option 1

Figure 21 shows an example of the Option 1 approach where the entire crossing is represented as a 1D channel with Lateral Links defined along both sides of the channel. This approach is typically the easiest option in terms of setting up the Lateral Links between the models because the entire branch can be linked without breaking it up into multiple segments, and it produces fewer linkage items to manage. While there is nothing that is technically wrong about this approach, it is important to understand how the exchange of flow is handled.

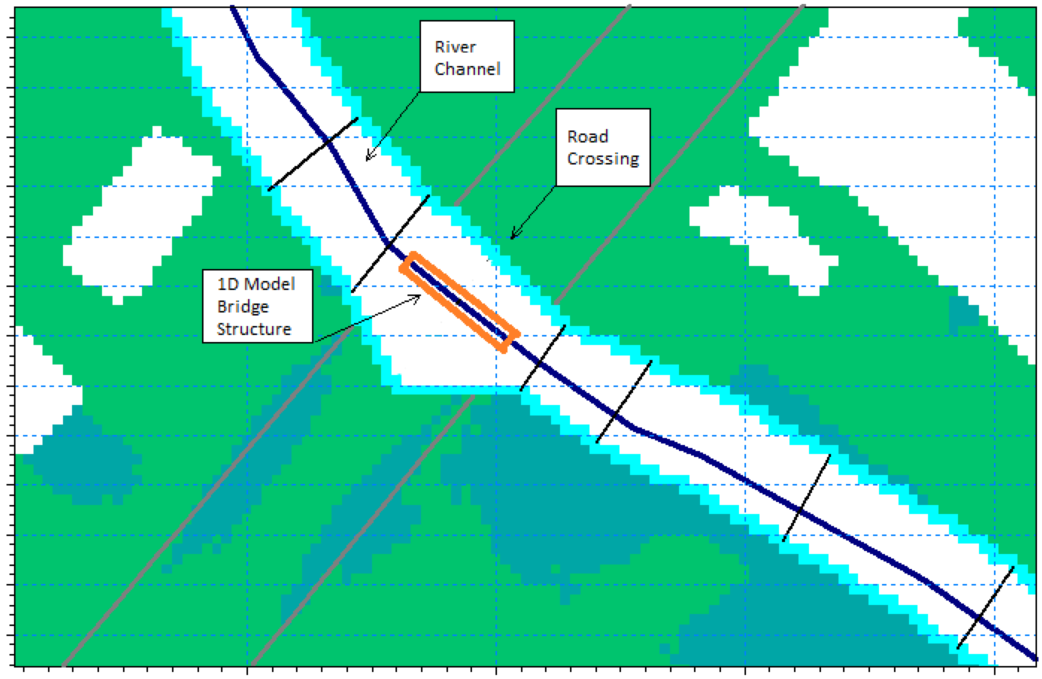


Figure 21: Example of Lateral Links Defined Across a Structure

In the above example, the cyan color shows the 2D model grid cells that are linked to the 1D model. These are the 2D model grid cells where the flow will be exchanged with the 1D model. The determination of whether flow is exchanged between the models via the Lateral Links is based on a comparison of the water levels in each model. In the case where the cells are crossing the bridge, the water level comparison with the 1D model must be done with the cross-sections immediately upstream and downstream of the bridge structure because the structure equation only calculated flow through the structure. As such, the determination of flow exchanges between the 1D and 2D model along the crossing is based on an average of the upstream and downstream water levels. Depending on the flow conditions this may or may not be a significant consideration. This approach also ignores any structural components of a bridge that may block the water in the channel from spilling into the roadway.

5.2.2.2 Option 2

Figure 22 shows an example of the Option 2 approach where the entire crossing is represented as a 1D channel with no Lateral Links defined adjacent to the structure crossing. In this example the 2D surface flow can only cross the channel via the 1D channel model so it assumes that there is no overland flow on the surface of the structure. While this approach may omit the transfer of overland flooding across the surface of the structure, the water level in the 1D channel is typically driving the majority of overland flooding to either side of the channel. As a result, the flow overtopping the structure is usually flowing in the downstream direction rather than across the structure.

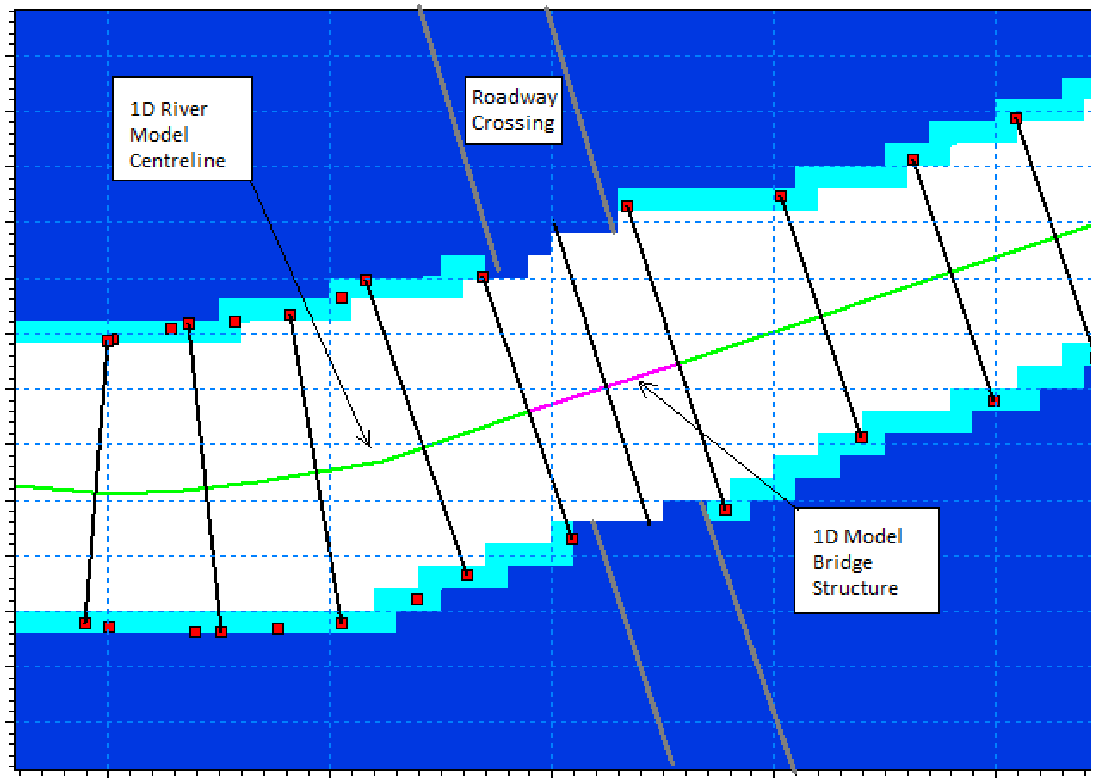


Figure 22: Example of Lateral Links Omitted Across a Structure

5.2.2.3 Option 3

Figure 23 shows and example of the Option 3 approach where the 2D surface of the crossing is represented in the 2D model and the Lateral Links are not defined through the crossing. In this example the flow in the channel passes under the 2D surface via the 1D model structure (culvert or bridge) and passes over the 2D surface if the structure (weir or bridge) gets overtopped. The advantage of this approach is that overland flooding from the 2D model can flow on the surface and across the structure entirely as 2D overland flow. The disadvantage of this approach is that if the 1D model structure is overtopped, the overflow does not spill onto the 2D surface but, rather, it goes directly into the 1D section of channel immediately downstream of the structure.

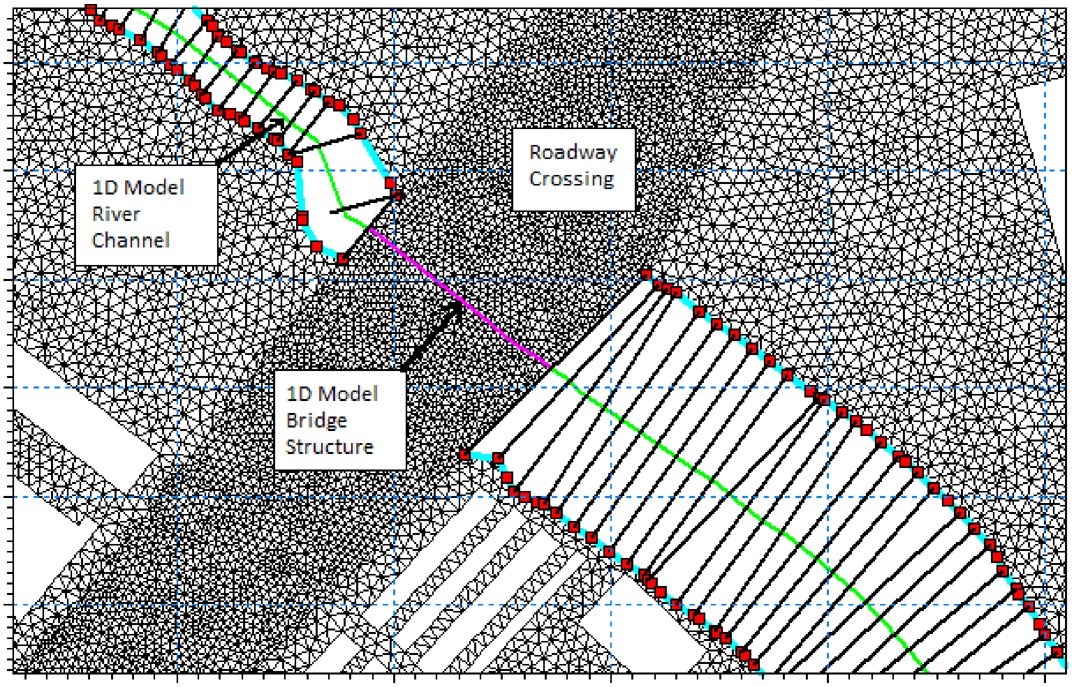


Figure 23: Example of the Structure Surface Represented in the 2D Model with no Lateral Links

5.3 Standard Link

The Standard Link allows one or more 2D model grid cells/mesh elements to link to the end of a 1D channel. In practical applications, a Standard Link is normally used under, but not limited to, following situations (see Figure 24):

1. A river reach discharges into an open water body (e.g. lake, estuary, bay, sea etc.) or an open area (e.g. wetland, floodplain etc.). In this instance the Standard Link is defined at the chainage location of the last cross-section of the 1D river model branch and it is linked to a line of grid cells/mesh elements intersecting the cross-section.
2. An open area (e.g. wetland, floodplain etc.) drains into a river reach. In this instance the Standard Link is defined at the chainage location of first cross-section of the 1D River model branch and it is linked to a line of grid cells/mesh elements intersecting the cross-section.
3. Representing a long, buried culvert in an otherwise 2D surface model. In this instance an isolated 1D model is prepared representing the upstream and downstream cross-sections of the channel immediately upstream and downstream of the culvert, with a culvert structure representing the culvert. The Standard Link is then used to link the 1D model to the 2D model at both the upstream and downstream cross-section locations such that a line of grid cells/mesh elements linked at the upstream and downstream cross-section locations. The exchange of flow depends on the calculated water surface elevations in connected 2D grid cells/mesh element along the Standard Link lines, so this type of situation requires the 2D model to provide a very accurate representation of

the topography along the linkage in order to get an accurate representation of the flows being exchanged.

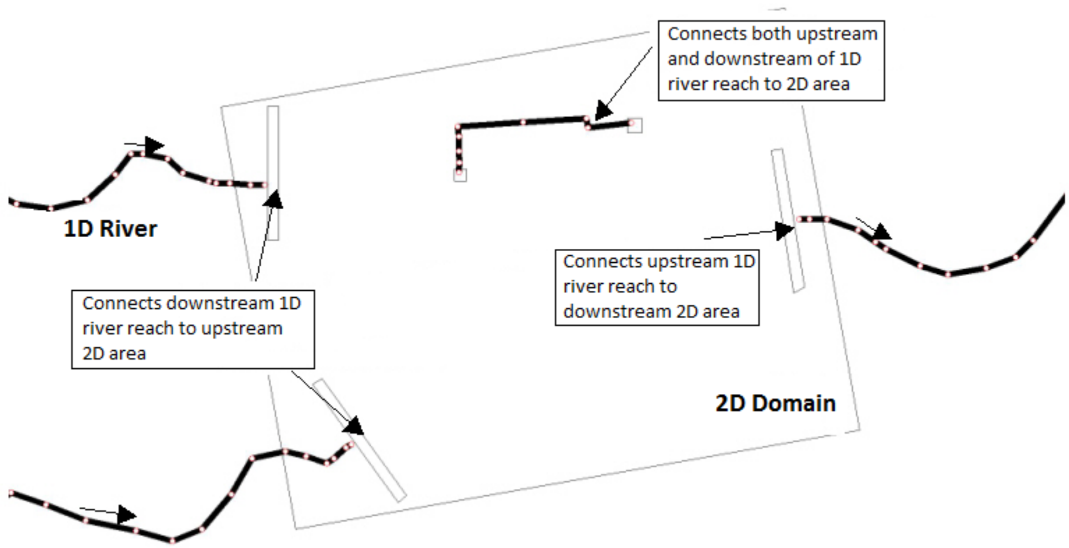


Figure 24: Examples of Typical Settings for Standard Links

6 Simulation Troubleshooting

When working with hydro-dynamic models, maintaining stable model is always challenging, not mention to maintain two hydro-dynamic models (i.e. 1D river model and 2D model) at the same time when running coupled model. Sometimes, both 1D model and 2D model can run by themselves, but when running coupled model it crashes. Here are some tips to trouble-shoot stability issue:

6.1 Initial conditions

Initial conditions are representing the conditions when the model initially starts. Most of time, 2D model can be initially dry unless there are known water surfaces.

If the model crashes right in the beginning of simulation, this suggests there might be issue with the setup of initial conditions. Initial conditions should be matching with the boundary conditions at first time step, in order to avoid a ‘numerical shock’ in the beginning of simulation.

It is sometimes very useful to generate a hot-start file, and use it as initial condition. For instance, running 1D model with a small constant flow for few time steps until it reaches dynamic equilibrium, and then use it as initial condition.

When using 1D channel ends link to connect 1D river model with 2D model, make sure the initial water levels specified in 1D river model and 2D model are the same.

6.2 Mesh

Instabilities in the MIKE 21 Flexible Mesh model can often times be caused by the mesh configuration. While the Mesh Generator tool attempts to generate a well formed mesh, it does sometimes produce mesh elements that can be susceptible to numerical instabilities. In particular, triangular mesh elements with obtuse angles and/or long narrow triangles tend to be problematic. If the MIKE 21 Flexible Mesh model is 'blowing up' in a particular area, evaluate the mesh in the problem area to look for ill-formed triangles. The ill-formed mesh elements may not be exactly in the same location as where the model blows-up, but a poorly formed mesh may impact results in nearby areas as well.

The mesh can be corrected manually by importing the mesh to the Mesh Generator and using the mesh editing tools, or the Mesh Generator file (.mdf) can be opened and the Mesh Smoothing tool can be used.

6.3 Topography

In general, numerical models do not like sharp and sudden transitions of any model parameter, but with 2D hydraulic modelling, the topography is obviously the most influential model input in determining the direction and depth of flow. As such, the model will often experience numerical instabilities in areas where there are sharp transitions in the topography, particularly near boundary conditions. Smoothing the topography near open boundaries and making a smooth transition between the boundary and the interior of the model is a way to help improve the numerical stability and robustness of the model.

6.4 Time step

After checking initial conditions, boundary conditions and topography data, adjusting time step could be the most effective way to improve the model stability. As discussed in time step sections in both 1D model and 2D model, when choosing time steps Courant criteria should be maintained all the time during simulation. If numerical stability issues persist then evaluate the results and look for maximum velocities in result file. If the velocities are much higher than what was assumed when initially setting the time step then adjust the time step value accordingly

For coupled 1D – 2D flood modelling in urban areas the time steps normally range from 1 – 2 seconds, but in many cases a time step of less than 1 s is required to maintain a Courant value less than 1.0 and/or achieve numerical stability in the model.

6.5 Flooding and drying depths

2D flood models are typically being prepared in areas where flooding impacts urban developments where there is a significant area of the model covered by paved surfaces. As a result, the drying depths of the 2D models are often very small (e.g. less than 5 mm) in order to better capture the spreading front of the flooding. However, very small drying depths can sometimes cause unrealistic high flow velocities resulting in some numerical stability problems. Increasing the drying depth could improve the model stability in these cases.

6.6 Computational parameters

In 1D river model, the Delta (implicit weighting factor) variable can be used to try to achieve numerical stability in the finite difference solution of the unsteady flow equations. The value of this parameter ranges between 0.5 and 1.0, where a value of 0.5 - 0.6 will give the most accurate solution of the equations, but is more susceptible to instabilities. A value of 0.8 – 1.0 provides the most stable solution but may be less accurate. Since a coupled 1D-2D flood model normally uses small time steps, a value of 0.8 – 0.85 generally provides accurate solution as well as stable computation.

6.7 Manning's Roughness

If numerical stability problems in the 2D model persist then adjusting Manning's roughness in the problem areas can be used to alleviate the problems. The numerical stability is often caused by very fast moving overland flow, so increasing the roughness to slow the water down can help to mitigate numerical instabilities. However, since this approach may introduce artificially higher roughness value it should be used only locally in the problem area and only as a last resort to the stability problem.

Note: This section refers to increasing the roughness of the surface because surface roughness is normally described using Manning's n value. However, for a MIKE 21 model this actually means decreasing the specified Manning's M value.

7 Conclusions

The purpose of this document is to provide a technical guideline for the use of MIKE FLOOD and the associated models and software to prepare coupled 1D-2D flood models and/or standalone 2D flood models. It is intended to deliver the current state of best practices to follow in considering how to configure and construct a reliable flood model using 2D flow modelling tools. Considering that 2D flood modelling is still a relatively new approach, that professional experience with this approach is quickly growing, and that the technology itself is constantly improving, it is expected that these guidelines should be reviewed and updated annually to ensure they remain relevant.

8 Recommendations

While this document covers best practices for configuring and constructing a MIKE FLOOD model it does not cover the reporting of the model setup and results. In order to facilitate an effective process of reviewing the 2D models being developed and submitted to TRCA, it is recommended to extend this document to include a standard reporting format with specific instructions on required model setup information, analysis of model coupling, and presentation and analysis of model results.

Appendix A

TRCA Methodology for Processing of Building Footprints

Toronto and Region Conservation for The Living City

MEMORANDUM

TO:	Nick Lorrain	DATE:	April 23, 2018
FROM:	Qiao Ying, Mike Todd	CFN:	
RE:	2D Modeling Guidance – Building Simplification		
CC:			

1.0 Introduction

The following presents the buildings simplification methodology developed by TRCA to assist with mesh file preparation for use in 2D modelling using the MIKE 21 Flexible Mesh (FM). When developing a mesh for 2D modelling purposes it is always a balance between accuracy and solution stability. High resolution is typically required to represent detailed topographic features (such as buildings, paved roads, flood walls/berms etc.), at the same time it is also important to minimize the number of mesh elements to reduce computation time and to avoid overly small meshes and angles that may cause model instability.

Building simplification is an important step before generating mesh, this is to avoid small meshes and small angles around buildings during the mesh construction process. The process of adjusting building footprints is done in ArcGIS using Aggregate Polygons and a Building Simplify functions.

2.0 Building Simplification Process

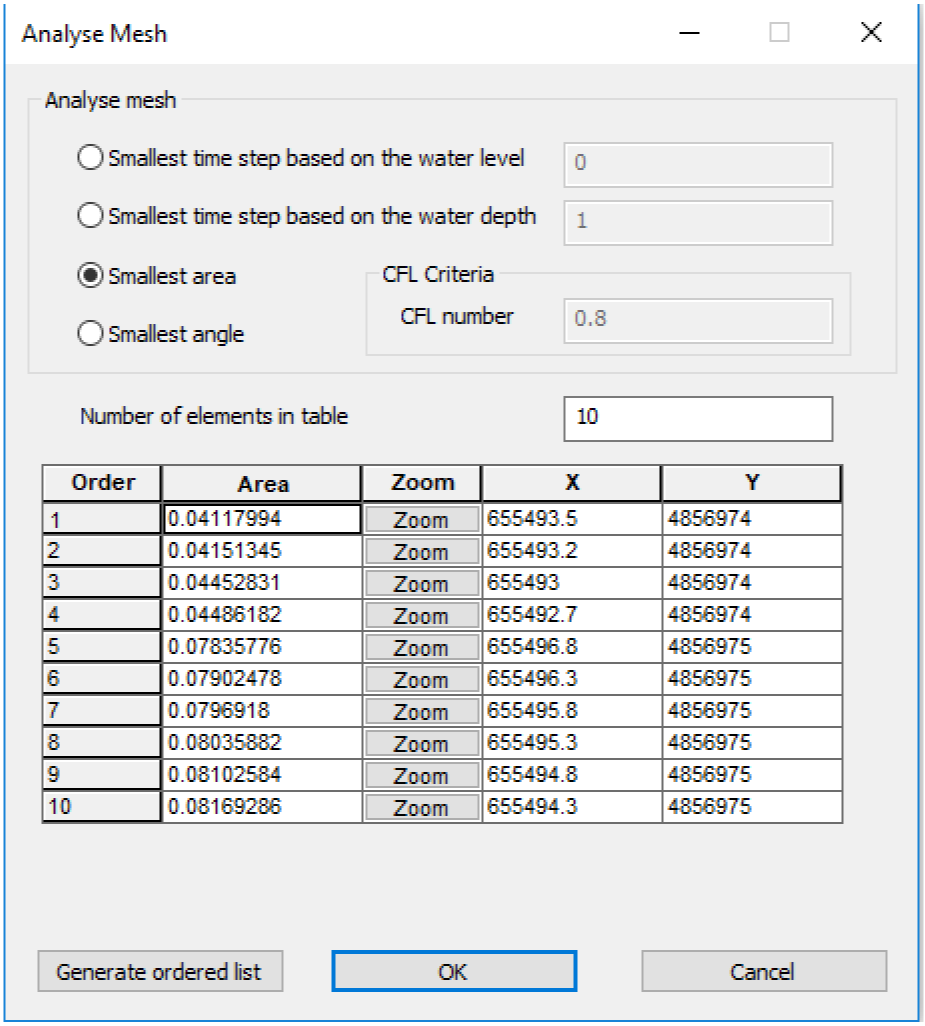
Building simplification process includes three steps:

1. Aggregate Polygons
2. Simplify Building
3. Further Adjustment

2.1 Aggregate Polygons

Gaps less than 1m between buildings are insignificant to the major flow path, it was determined to use 1m as the criterion to merge the buildings. Aggregate Polygons function in ArcGIS is used to merge buildings with gaps less than and equal to 1m. The major parameters in Aggregate Polygons function which need input are Aggregation_distance and output_table. Aggregation_distance should be set to 1m; name of out_table should be given, and this table is a one-to-many relationship table that links the aggregated polygons to their source polygon features.

Figure 1 gives a comparison of meshes around the buildings with small gap (0.3m) and with buildings aggregated. It can be seen that the small gap between two buildings cause very small meshes, and the size of mesh elements within the gap is less than 0.1m^2 (see list of elements with smallest area in screen capture below)



Order	Area	Zoom	X	Y
1	0.04117994	Zoom	655493.5	4856974
2	0.04151345	Zoom	655493.2	4856974
3	0.04452831	Zoom	655493	4856974
4	0.04486182	Zoom	655492.7	4856974
5	0.07835776	Zoom	655496.8	4856975
6	0.07902478	Zoom	655496.3	4856975
7	0.0796918	Zoom	655495.8	4856975
8	0.08035882	Zoom	655495.3	4856975
9	0.08102584	Zoom	655494.8	4856975
10	0.08169286	Zoom	655494.3	4856975

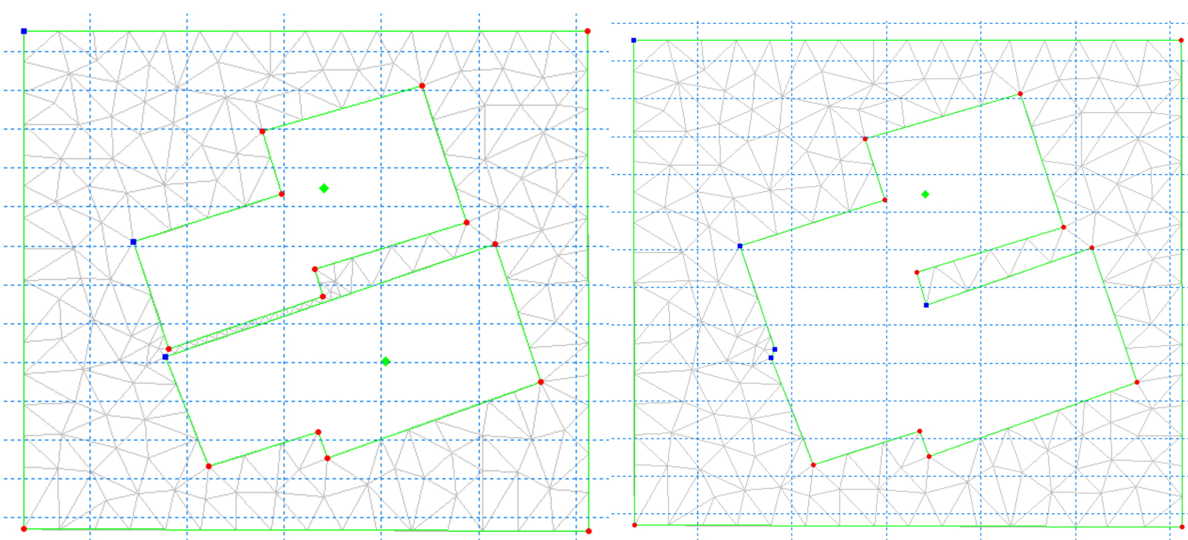


Figure 1 Comparison of mesh with original (left) vs. aggregated polygons (right)

2.2 Simplify Building

Once buildings with small gaps were merged, next step is to remove points that are very close to each other along the building outline while maintaining their essential shape and size.

Buildings don't always come with regular shapes like square or rectangle, and a lot of time they are irregular shaped with round corners, short sides or are small and narrow, etc. and these irregular features may cause very small meshes and small angles within a developed mesh.

The Simplify Building function in ArcGIS is used to remove points that are close to each other. Major parameters for consideration in the Simplify Building function are:

- simplification_tolerance: 3 values have been tested, 0.5m, 1m and 2m, and based on the test 1m is recommended.
- minimum_area: keep default value of 0, that is, to keep all buildings.
- conflict_option: choose CHECK_CONFLICTS, that is, to check for potential conflicts; the conflicting buildings will be flagged.

Figure 2 below shows the comparison of mesh with original building and simplified building, it can be seen after simplification building outline was maintained and short sides were removed, this process has removed small meshes and small angles presented with original building outline.

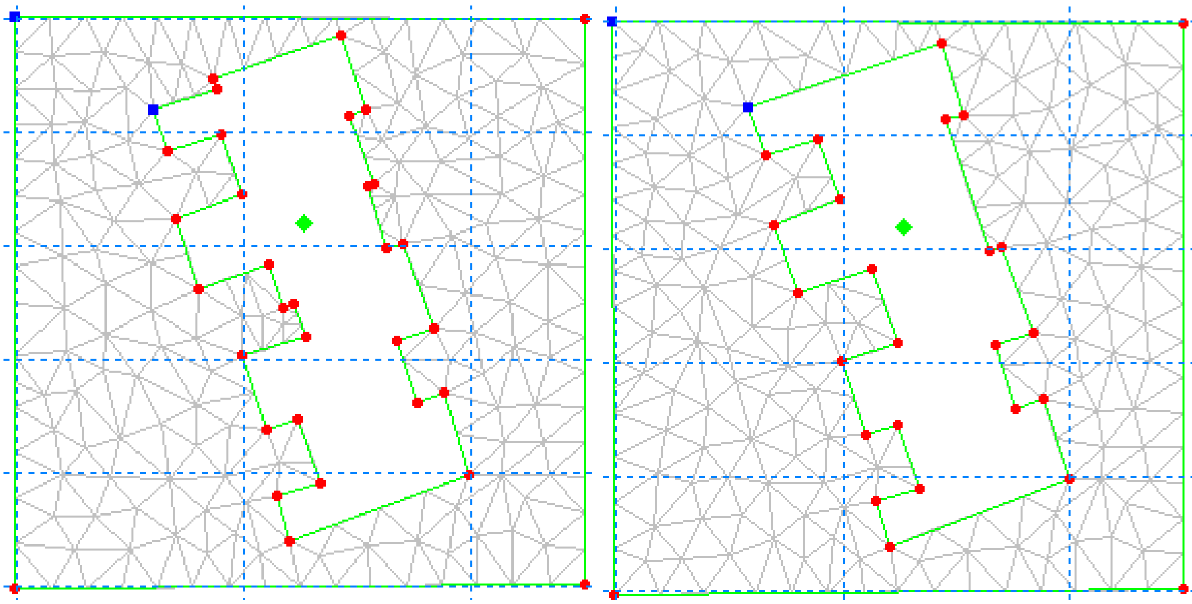


Figure 2 Comparison of mesh with original (left) vs. building simplify (right)

2.3 Further Adjustment

Further adjustment may be required after first two steps. For instance, after the Simplify Building step, if the conflict_option field shows flags for some of buildings, this means those buildings may be overlapped after simplification, then the Aggregation Polygons function could be used again to merge those overlapped buildings. Sometimes, manual adjustment may be required for a final touch.

3.0 Conclusions and Recommendations

When preparing mesh files used for 2D MIKE 21 FM modelling, prior to generating a mesh it is necessary to simplify building footprints to avoid small meshes and small angles around the building outline, and this is done to improve model stability during the computation process.

A GIS approach was developed to simplify building footprints, which involves three steps 1) Merge buildings using Aggregate Polygons function; 2) Simplify building using Simplify Building function; 3) Further adjustment using Aggregate Polygons or Manual adjustment. This approach is efficient and effective when working with bulk building footprints.