Comparing 1D, 2D, and 3D Hydraulic Models in Urban Flooding Applications

Taylor Kesler
Utah State University

Follow this and additional works at: https://digitalcommons.usu.edu/etd

Part of the Civil and Environmental Engineering Commons

Recommended Citation
https://digitalcommons.usu.edu/etd/8691

This Thesis is brought to you for free and open access by the Graduate Studies at DigitalCommons@USU. It has been accepted for inclusion in All Graduate Theses and Dissertations by an authorized administrator of DigitalCommons@USU. For more information, please contact digitalcommons@usu.edu.
COMPARING 1D, 2D, AND 3D HYDRAULIC MODELS IN URBAN FLOODING APPLICATIONS

by

Taylor Kesler

A thesis submitted in partial fulfillment of the requirements for the degree of

MASTER OF SCIENCE

in

Civil and Environmental Engineering

Approved:

Zachary B. Sharp, Ph.D., P.E.  Steven L. Barfuss, M.S.
Major Professor  Committee Member

John D. Rice, Ph.D.  D. Richard Cutler, Ph.D.
Committee Member  Vice Provost for Graduate Studies

UTAH STATE UNIVERSITY
Logan, Utah
2022
ABSTRACT

Comparing 1D, 2D, and 3D Hydraulic Models in Urban Flooding Applications

by

Taylor Kesler, Master of Science

Utah State University, 2022

Major Professor: Dr. Zachary B. Sharp
Department: Civil and Environmental Engineering

This research investigates the differences between 1D, 2D, and 3D numerical hydraulic models in a developed setting. Three models were built and tested of a section of the Logan River. The reach in question has been developed with homes, a park and a school. A broad crested weir serves as a diversion structure for an irrigation canal. It is believed that this diversion structure is the primary cause of flooding in the area.

The 1D and 2D models were built using HEC-RAS. The 3D model was built using Star-CCM+. The terrain for each model comes from open source 0.5m LiDAR of the area. The 1D model was found to be not ideal for this study. The complex terrain and poor modeling of lateral flows gave results with low confidence. The 2D model better shows water leaving the channel and entering the floodplain, highlighting the areas where flooding first occurs. The flooding occurred upstream of the diversion structure. This implies that flooding occurs primarily due to rising stream levels, not the diversion structure. The 2D model also shows the flow of water across the floodplain. The 3D CFD model shows similar results to the 2D model in the river channel and floodplain. The 3D
model however, gives detailed results of the weir itself. The 3D model shows inconsistencies in the terrain model which cause unrealistic flow paths. The 3D model shows the behavior of the water at and around the weir in great detail.

The second portion of the study investigates the effect of changing the weir coefficient in the 2D model. The model showed an increase in weir efficiency as the weir coefficient increased, but no significant changes to the flooding behavior overall. Further pointing to the weir not being the primary source of flooding.
Comparing 1D, 2D, and 3D Hydraulic Models in Urban Flooding Applications

Taylor Kesler

Floods have been a hazard to people since people first started building near rivers. Predicting floods can be very beneficial to save lives and property. Computers have made it possible to solve fluid dynamics equations in a fast and efficient manner. Computer programs have been designed to solve these equations and create digital models of floods.

This study compares three different methods of computer modeling and explores their advantages and disadvantages. One-dimensional models solve fluid equations by setting up a series of cross sections. Two-dimensional models use a grid-like mesh to solve fluid equations from one cell to the next. Three-dimensional or computational fluid dynamics models use a three-dimensional mesh to solve complex fluid equations.

The study focuses on a stretch of the Logan river, located in Logan, Utah USA. This is a stretch of river that is prone to flooding during high runoff due to snowmelt. Determining the primary causes of flooding can help create plans to reduce the risk of loss of life, and property damage.
ACKNOWLEDGMENTS

I would like to thank my major professor Dr. Zac Sharp for his help through this whole process for his support and instruction. I would also like to thank Steve Barfuss and Dr. John Rice for serving on my committee.

I would also like to thank my fellow students in the department and at the UWRL who were going through the same process. They were always there to be a listening ear or to give welcomed suggestions.

Thanks to my family for their continued support over the years. I could not have made it to this point without them. Special thanks to my biggest supporters, my wife Kenna for being at my side through the long days and nights, and for not giving up, and our sweet infant daughter who, very kindly, learned to sleep through most of the night.

Finally, I would like to thank God and His Son Jesus Christ. All good things in my life have been blessings from God and I would not be capable of anything without their love and help.

Taylor Kesler
# CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>ABSTRACT</td>
<td>iii</td>
</tr>
<tr>
<td>ACKNOWLEDGMENTS</td>
<td>vi</td>
</tr>
<tr>
<td>LIST OF TABLES</td>
<td>viii</td>
</tr>
<tr>
<td>LIST OF FIGURES</td>
<td>ix</td>
</tr>
<tr>
<td>CHAPTER I: Introduction</td>
<td>1</td>
</tr>
<tr>
<td>Purpose</td>
<td>1</td>
</tr>
<tr>
<td>CHAPTER II: Literature Review</td>
<td>3</td>
</tr>
<tr>
<td>CHAPETER III: Experimental Methods</td>
<td>8</td>
</tr>
<tr>
<td>Theoretical Background</td>
<td>8</td>
</tr>
<tr>
<td>Area of Study</td>
<td>15</td>
</tr>
<tr>
<td>Terrain Data</td>
<td>18</td>
</tr>
<tr>
<td>Channel and Floodplain Friction</td>
<td>19</td>
</tr>
<tr>
<td>Weir</td>
<td>20</td>
</tr>
<tr>
<td>Flow Data</td>
<td>23</td>
</tr>
<tr>
<td>1D Model Setup</td>
<td>24</td>
</tr>
<tr>
<td>2D Model Setup</td>
<td>27</td>
</tr>
<tr>
<td>3D Model Setup</td>
<td>30</td>
</tr>
<tr>
<td>Chapter IV: Results and Discussion</td>
<td>36</td>
</tr>
<tr>
<td>1000 CFS Results</td>
<td>37</td>
</tr>
<tr>
<td>1670 CFS Results</td>
<td>44</td>
</tr>
<tr>
<td>2500 CFS Results</td>
<td>50</td>
</tr>
<tr>
<td>FEMA Flood Map</td>
<td>57</td>
</tr>
<tr>
<td>Rating Curves</td>
<td>58</td>
</tr>
<tr>
<td>Channel Friction Sensitivity</td>
<td>67</td>
</tr>
<tr>
<td>Pros and Cons of Modeling</td>
<td>68</td>
</tr>
<tr>
<td>Chapter V: Conclusions</td>
<td>71</td>
</tr>
</tbody>
</table>
LIST OF TABLES

Table 1. Differences between 1D and 2D modeling ................................................................. 5
Table 2. Roughness heights. ..................................................................................................... 20
Table 3. Tested Flow Rates..................................................................................................... 24
Table 4. 1000 cfs inundated area comparison ....................................................................... 41
Table 5. 1000 cfs weir flow summary ...................................................................................... 43
Table 6. 1670 cfs inundated areas comparison ...................................................................... 48
Table 7. 1670 cfs weir flow summary ..................................................................................... 50
Table 8. 2500 cfs inundated areas comparison ...................................................................... 54
Table 9. 2500 cfs weir flow summary ..................................................................................... 56
Table 10. 1670 cfs Inundated area comparison ...................................................................... 57
Table 11. 2500 cfs Inundated area comparison ...................................................................... 57
Table 12. Weir coefficients .................................................................................................... 60
Table 13. Weir coefficient comparison ................................................................................... 65
Table 14. Friction sensitivity summary .................................................................................... 67
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Energy equation representation.</td>
<td>10</td>
</tr>
<tr>
<td>2</td>
<td>HEC-RAS subdivision method</td>
<td>11</td>
</tr>
<tr>
<td>3</td>
<td>2D HEC-RAS mesh</td>
<td>12</td>
</tr>
<tr>
<td>4</td>
<td>Visual representation of various models</td>
<td>15</td>
</tr>
<tr>
<td>5</td>
<td>Area of study with extents of CFD model outlined</td>
<td>17</td>
</tr>
<tr>
<td>6</td>
<td>DEM in HEC-RAS</td>
<td>19</td>
</tr>
<tr>
<td>7</td>
<td>Crockett Canal diversion weir</td>
<td>21</td>
</tr>
<tr>
<td>8</td>
<td>Weir cross section in HEC-RAS</td>
<td>22</td>
</tr>
<tr>
<td>9</td>
<td>3D Weir model</td>
<td>23</td>
</tr>
<tr>
<td>10</td>
<td>Example cross section</td>
<td>25</td>
</tr>
<tr>
<td>11</td>
<td>1D Geometry</td>
<td>27</td>
</tr>
<tr>
<td>12</td>
<td>2D Mesh example</td>
<td>28</td>
</tr>
<tr>
<td>13</td>
<td>2D Mesh detail at weir location</td>
<td>30</td>
</tr>
<tr>
<td>14</td>
<td>3D Study area</td>
<td>32</td>
</tr>
<tr>
<td>15</td>
<td>3D Weir joined with the surface</td>
<td>32</td>
</tr>
<tr>
<td>16</td>
<td>Surface mesh of weir structure and river channel</td>
<td>34</td>
</tr>
<tr>
<td>17</td>
<td>Model volume mesh</td>
<td>34</td>
</tr>
<tr>
<td>18</td>
<td>Water surface area with dividing measurement line</td>
<td>37</td>
</tr>
<tr>
<td>19</td>
<td>1D 1000 cfs flood extents</td>
<td>38</td>
</tr>
<tr>
<td>20</td>
<td>2D 1000 cfs flood extents</td>
<td>39</td>
</tr>
<tr>
<td>21</td>
<td>3D 1000 cfs flood extents</td>
<td>40</td>
</tr>
<tr>
<td>22</td>
<td>1D 1000 cfs weir profile</td>
<td>42</td>
</tr>
<tr>
<td>23</td>
<td>2D 1000 cfs weir profile</td>
<td>42</td>
</tr>
<tr>
<td>24</td>
<td>3D 1000 cfs weir profile</td>
<td>42</td>
</tr>
<tr>
<td>25</td>
<td>HGL profile lines, 1000 cfs</td>
<td>44</td>
</tr>
<tr>
<td>26</td>
<td>1D 1670 cfs flood extents</td>
<td>45</td>
</tr>
<tr>
<td>27</td>
<td>2D 1670 cfs flood extents</td>
<td>46</td>
</tr>
<tr>
<td>28</td>
<td>3D 1670 cfs flood extents</td>
<td>47</td>
</tr>
<tr>
<td>29</td>
<td>1D 1670 cfs weir profile</td>
<td>49</td>
</tr>
</tbody>
</table>
Figure 30. 2D 1670 cfs weir profile ................................................................. 49
Figure 31. 3D 1670 cfs weir profile ................................................................. 49
Figure 32. 1670 cfs hydraulic grade lines ....................................................... 50
Figure 33. 1D 2500 cfs flood extents ............................................................. 51
Figure 34. 2D 2500 cfs flood extents ............................................................. 52
Figure 35. 3D 2500 cfs flood extents ............................................................. 53
Figure 36. 1D 2500 cfs weir profile ............................................................... 55
Figure 37. 2D 2500 cfs weir profile ............................................................... 55
Figure 38. 3D 2500 cfs weir profile ............................................................... 55
Figure 39. 2500 cfs hydraulic grade lines ..................................................... 56
Figure 40. FEMA flood map ........................................................................... 58
Figure 41. Rating curve comparison ............................................................... 59
Figure 42. Weir coefficient 2.6 inundation boundary ...................................... 61
Figure 43. Weir coefficient 3.1 inundation boundary ...................................... 62
Figure 44. Weir coefficient 3.6 inundation area ............................................ 63
Figure 45. Weir coefficient 4.1 inundation area ............................................ 64
Figure 46. Rating curves for weir coefficient comparison ................................ 66
Figure 47. 3D flow at the weir ...................................................................... 69
CHAPTER I

INTRODUCTION

River flooding has been a problem to people for thousands of years. Floods can cause serious economic damage as well as loss of life. In 2020 there were 59 fatalities directly caused by flooding in the United States (NWS 2021). In 2019 flooding caused 3.750 billion dollars’ worth of damage in the United States (“Economic damage caused by floods in the U.S. 2019” n.d.). Mapping the extents of flood waters has the potential to save lives as well as millions of dollars in damages each year.

Flooding in urban areas poses challenges not found in rural, or undeveloped areas. The addition of vegetation, buildings, and man-made topography change the way water flows in a flood. Since urban areas are densely populated, there is a greater probability of economic damages, and potential loss of life in a major flooding event.

Flood modeling is a tool that engineers, city planners, developers, and emergency crews can use to minimize damages caused by flooding. The Federal Emergency Management Agency (FEMA) is responsible for mapping floods and providing flood insurance to people who live in vulnerable areas. These flood maps provide vital information to determine the risk associated with a flood of a given level.

Purpose

The basis of this study came from Logan City and their desire to better understand the river flooding. It is common knowledge that the river reach in question is the first area to flood during a high runoff event. Logan City wanted to know more about the causes of this flooding, and what factors have the most influence. Performing tests using different model types helps determine the biggest contributors to the flooding. In
particular, Logan City wanted to know about a particular canal diversion structure and if it caused flooding due to water being backed up.

The purpose of this study is to compare some of the common types of flood modeling to determine the difference in results between the methods. The methods used will be a one-dimensional (1D) and two-dimensional (2D) model done in the United States Army Corp of Engineers (USACE) program from the Hydrologic Engineering Center (HEC) called the River Analysis System (HEC-RAS). A three-dimensional (3D) Computational Fluid Dynamics (CFD) model was also used with the CFD code Star-CCM+.

3D hydraulic models have higher fidelity and need less assumptions than 1D and 2D models, but come at a higher cost due the complexity and heavy computational requirements. This study will look at the differences in detail among the three types of models to determine if a full 3D study is necessary in flood modeling cases. Comparing the three different model types and their outputs will help determine the best model type for similar studies. There must be a balance between required detail and computational effort. A simple river channel may be sufficiently modeled using a 1D model. A large floodplain may only require a 2D model to determine flood depths. A CFD model can show the effects of flow over a highly detailed hydraulic structure. Determining the complexity will help determine the necessary model.

Logan City was interested in knowing the locations where flooding first occurred. The model comparisons allow the modeler to compare and choose what type of model is best suited for a particular scenario.
CHAPTER II
LITERATURE REVIEW

Engineers have been attempting to predict the extents of floods for years. The use of computer models in recent decades has helped improve the feasibility of mapping large and small flood events. Due to the complex nature of fluid dynamics, computer models have difficulty accurately representing flood events at times. For this reason, physical models have been used to model fluid flow. In 1943 the USACE constructed a model of the Mississippi river basin. The model spread over 200 acres and was used to predict areas vulnerable to flooding (“America’s Last Top Model” n.d.).

Physical models are still widely used to model fluid flow and hydraulic structures. However, computer models can be built and tested much faster in many cases. Computer models allow a user to create and modify a model without the need for a lengthy labor process, and costly materials. In some cases, this can save time and costs.

1D computer models are the simplest type of computer model. 1D models evaluate fluid flow in one direction by simplifying more complex equations with assumptions. The channel geometry is represented by a series of consecutive cross sections cut through the channel. The flow is assumed to be normal to these cross sections. Steady state 1D models use the backwater equations, derived from the energy equation, to solve for the depth at a given flow rate (Chanson 2004). Unsteady flow can be solved using the Saint-Venant equations, derived from the momentum equations (Chanson 2004). Gradually varied flow can also be modeled. Gradually varied flow occurs most commonly from the backwater created by culverts and channel restrictions (Colorado Water Conservation Board 2006). 1D modeling is generally considered
accurate and has been used by the USACE and FEMA to perform floodplain analyses for nearly 50 years now (USACE 1988).

Some limitations to 1D steady flow modeling are; the flow condition is steady and gradually varied, velocity is only accounted for in one direction, flow rates are constant, the channel slope is relatively flat (less than one percent), and they cannot model the effects of flow attenuation due to storage (Colorado Water Conservation Board 2006).

1D models have been verified to be accurate after being tested against physical models, and actual flood events. The USACE conducted tests to validate HEC-RAS results, these tests compared models to hand calculations, laboratory tests and real-world scenarios. Tests were performed for an open trapezoidal channel, flow over a bump, and flow over a drop structure. HEC-RAS provided adequate results for each of these tests (USACE 2018). The tests compared the calculated water surface with an observed water surface. The USACE conducted large scale tests by building a model of a river basin and comparing flood results with historic flooding data (USACE 2018).

2D models are often used to model floodplains and other areas where the flow path is poorly defined. 2D models calculate water depth and velocity across a grid or mesh that defines the topography (Toombes and Chanson 2011). These depths and velocities are usually calculated using the Saint-Venant equations, a derivation of the Navier-Stokes equations (Toombes and Chanson 2011). The accuracy of 2D modeling is limited to some degree by the assumptions of hydrostatic pressure in areas of adverse pressure gradients, such as separation zones (Bates et al. 2005). In an adverse pressure gradient flow separation occurs

2D meshes are defined by the user to cover the area of interest. The cells inside
the mesh are squares in most areas, however cells can have up to eight sides in HEC-RAS (USACE 2016a). These cells are solved using a finite element method to determine properties such as depth and water velocity. Each cell face creates a cross section based on the underlying topography. This ensures that water flows from one cell to the next based on elevations and volumes of the cell.

2D HEC-RAS has many capabilities, however 2D HEC-RAS is still limited in some areas such as modeling bridges (USACE 2016a). In cases where a strictly 1D or 2D model is not viable, a combined 1D-2D model can be created. A 1D-2D model will use a standard 1D model on the main river channel, the flood plain, where flow paths are less defined, can be modeled with a 2D mesh connected to the 1D modeled river channel.

The differences between 1D and 2D are important to take in to consideration when modeling. A brief list of these differences is summarized in Table 1.

Table 1. Differences between 1D and 2D modeling (Colorado Water Conservation Board 2006).

<table>
<thead>
<tr>
<th>Property or Factor</th>
<th>One-Dimensional Modeling</th>
<th>Two-Dimensional Modeling</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flow Direction</td>
<td>Prescribed (streamwise)</td>
<td>Computed</td>
</tr>
<tr>
<td>Transverse Velocity and Momentum</td>
<td>Neglected</td>
<td>Computed</td>
</tr>
<tr>
<td>Vertical Velocity and Momentum</td>
<td>Neglected</td>
<td>Neglected</td>
</tr>
<tr>
<td>Velocity Averaged Over…</td>
<td>Cross Sectional Area</td>
<td>Depth at a Point</td>
</tr>
<tr>
<td>Transverse Velocity Distribution</td>
<td>Assumed Proportional to Conveyance</td>
<td>Computed</td>
</tr>
<tr>
<td>Transverse Variations in Water Surface</td>
<td>Neglected</td>
<td>Computed</td>
</tr>
<tr>
<td>Vertical Variations</td>
<td>Neglected</td>
<td>Neglected</td>
</tr>
<tr>
<td>Unsteady Flow Routing</td>
<td>Can Be Included</td>
<td>Can Be Included</td>
</tr>
</tbody>
</table>
3D CFD is used across various industries to solve fluid dynamics problems. Aerospace and automotive companies use 3D CFD for aerodynamics modeling. In the past, engineers were constrained to physical models, or completed prototypes to determine aerodynamic capabilities. Incorporating 3D CFD models into the design process can improve design from the beginning ("What is Computational Fluid Dynamics (CFD)?" n.d.).

3D CFD is useful in hydraulic numerical modeling as well. 3D CFD can be used to model flow in pipelines, water meters, elbows, and valves. These applications are helpful in determining properties such as friction losses, and minor losses (Barth 2021). Analytical solutions exist for laminar flows, but not for turbulent flows (Beck 2021). It is believed (but not proven) that the Navier-Stokes equations can model any kind of flow, both laminar and turbulent ("What is Turbulent Flow?" n.d.). 3D CFD can be used in open channel flow scenarios as well. Ryan (2016) used 3D CFD to model fish passage structures. Spillways, weirs, and gated structures are a few examples of useful 3D CFD modeling.

As implied by the name, 3D CFD is capable of modeling fluid flow in three dimensions. Unlike 2D models, 3D models account for vertical momentum, velocities, and pressure changes. 3D CFD uses the Navier-Stokes equations to model the flow in three dimensions (Robb and Vasquez 2015). The Navier-Stokes equations can only be used directly to solve simple flows. For complex flows the Navier-Stokes equations are simplified as Reynolds-Averaged Navier-Stokes (RANS) equations ("turbulence.pdf" n.d.), which are solved in a CFD software. Roughness is calculated differently in different types of modeling. When compared to 1D models, 3D models are less sensitive
to change in the model roughness (Morvan et al. 2008). When compared to 2D models, 3D models have also been found to be less sensitive to roughness, this is due to bottom stresses causing recirculation which leads to shear between fluid layers (Lane et al. 1999). Surface roughness plays an important role in determining flood extents and water velocities. Luo et al. (2018) found that comparison ratios between 3D/2D and 3D/1D decrease with depth due to the different calculation approaches based on roughness definitions.
CHAPTER III
EXPERIMENTAL METHODS

Theoretical Background

The objective of numerical modeling is to simplify and solve complex physics equations using a computer. Fluid dynamics equations become more complex as they reach a higher order. These higher order equations stem from basic fluid flow equations. The continuity equation is one of the most fundamental open channel flow equations.

\[ Q = VA \]  

(1)

Where \( Q \) = flow rate, \( V \) = velocity, and \( A \) = the area of the cross section. In a closed system (such as a control volume in a pipe) with a steady solution the flow rate will remain constant even though the velocity and cross section can change. For example, a stream may experience contractions and expansions. As long as there is no inflow or outflow, the flow rate will remain constant.

The Bernoulli equation is used to describe the energy of a fluid. The Bernoulli equation is derived from the Navier-Stokes equation for a frictionless steady flow state (Chanson 2004).

\[ \frac{v^2}{2g} + z + \frac{P}{\gamma} = Total \ Head \]  

(2)

Where \( v \) = velocity, \( z \) = elevation above datum, \( P \) = pressure, \( \gamma \) = specific weight, and \( g \) = gravity constant. In open channel flow the energy is often measured as an elevation known as pressure head. The pressure head is expressed in units of length and is calculated using a variation of the Bernoulli equation.

\[ \frac{v^2}{2g} + z + \frac{P}{\rho g} = H \]  

(3)
Where $\rho = \text{fluid density}$. In open channel hydraulics the pressure $P$ is often expressed as the local atmospheric pressure (Chanson 2004). The energy equation is derived from the Bernoulli equation and is a fundamental open channel flow equation.

$$z + y + \frac{v^2}{2g} = H \tag{4}$$

1D HEC-RAS generates water surface profiles for individual cross sections by solving the energy equation in an iterative process called the standard step process (USCAE 2016). The energy equation is as follows:

$$Z_2 + Y_2 + \frac{a_2 V_2^2}{2g} = Z_1 + Y_1 + \frac{a_1 V_1^2}{2g} + h_e \tag{5}$$

Where $Z_1, Z_2 = \text{elevation of the main channel inverts}$, $Y_1, Y_2 = \text{depth of water at cross section}$, $V_1, V_2 = \text{average velocities (total discharge/ total flow area)}$, $a_1, a_2 = \text{velocity weighting coefficients}$, $g = \text{gravitational constant}$, and $h_e = \text{energy head loss}$. Figure 1 shows the energy equation in a profile view of a channel.
The cross sections defined in 1D HEC-RAS are further subdivided into the main channel area and overbanks. A variation of Manning’s equation is used to calculate the flow of each individual area. Each of these areas is defined by a new Manning’s n-value in the respective area. The Manning equation for each subdivision is:

\[ Q = KS_f^{1/2} \]  

(6)

\[ K = \frac{1.486}{n}AR^{2/3} \]  

(7)

Where \( K \) = conveyance for subdivision, \( n \) = Manning’s roughness coefficient for subdivision, \( A \) = flow area for subdivision, \( R \) = hydraulic radius for subdivision (area/wetted perimeter), \( S_f \) = slope of the energy gradeline (USCAE 2016). Figure 2 shows an example of a channel cross section divided in HEC-RAS.
The Manning’s $n$ coefficient is an empirical coefficient used to describe the roughness of a given surface. A higher coefficient indicates a greater roughness for example finished concrete has a Manning’s $n$ coefficient of 0.012 and a flood plain with trees has a Manning’s coefficient of 0.15 (Chanson 2004).

Another fundamental equation of fluid dynamics is the momentum equation. The energy equation is useful for calculating depths but is not applicable when flow passes through critical depth. For cases of flow passing through critical depth, such as at a hydraulic jump or a control structure, it is necessary to use the momentum equation. The momentum equation is derived from Newton’s second law of motion. The momentum equation is shown below:

$$\sum \vec{F} = \frac{D}{Dt} (M + \vec{V})$$

(7)

Where $\Sigma \vec{F} = \text{the resultant of all external forces acting on a system}$, $M = \text{mass of a system}$, and $\vec{V} = \text{velocity of the center of mass of a system}$.

2D modeling is capable of modeling flow in two directions across a mesh. These
meshes consist of polygons which are solved using a finite element analysis. 2D modeling is useful for modeling a floodplains and other areas where flow the flow path is poorly defined (Toombes and Chanson 2011). The cells in a 2D mesh can vary in size depending on the level of detail required from a certain area. For example, smaller cells will give more detail at a weir or control structure, but larger cells provide sufficient detail in a floodplain with simple topography. Figure 3 shows an example of cell size.

![2D HEC-RAS mesh](image)

**Figure 3.** 2D HEC-RAS mesh (USACE 2016a).

2D modeling uses a simplified version of the Navier-Stokes equations called the Shallow water equations to calculate 2D fluid flow. Incompressible flow, uniform density, and hydrostatic pressure are assumed and the equations are Reynolds averaged so that turbulent motion is approximated using eddy viscosity. It is also assumed that the vertical length scale is much smaller than the horizontal length scale. As a result the
vertical velocity is small and pressure is hydrostatic (USCAE 2016). The shallow water equation is shown below:

\[
\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -g \frac{\partial H}{\partial x} + v_t \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right) - c_f u + f v \tag{8}
\]

\[
\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = -g \frac{\partial H}{\partial y} + v_t \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right) - c_f v + f u \tag{9}
\]

Where \(u\) and \(v\) are the velocities in the cartesian directions, \(g\) is the gravitational acceleration, \(v_t\) is the horizontal eddy viscosity coefficient, \(c_f\) is the bottom friction coefficient, and \(f\) is the Coriolis parameter. The left-hand side of the equation contains the acceleration terms. The right-hand side represents the internal or external forces acting on the fluid (USCAE 2016).

3D modeling is the most complex of the three modeling methods discussed here. 3D modeling or computational fluid dynamics (CFD) is capable of modeling complex properties of fluid flow which are only assumed in 1D and 2D modeling, such as hydrostatic pressure distribution or shear forces to name a few (Toombes and Chanson 2011). CFD modeling divides an area of study into small 3D control volumes. The model then solves for forces acting on these control volumes using the Navier-Stokes equations shown in equation 10. The Navier-Stokes equations are a differential form of the linear momentum principle(Finnemore and Franzini 2011).

\[
-\frac{\partial p}{\partial x} + \mu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) = \rho \left[ \frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} \right] \tag{10a}
\]

\[
-\frac{\partial p}{\partial y} + \mu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) = \rho \left[ \frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} \right] \tag{10b}
\]

\[
-\rho g - \frac{\partial p}{\partial z} + \mu \left( \frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right) = \rho \left[ \frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} \right] \tag{10c}
\]
CFD is useful in modeling applications where a large physical model was once
the only means of modeling flow. In some cases a CFD model can replace a physical
model, saving time and resources (Andersson 2012). Since CFD models break down, and
solve many individual cells in an area, they can become very complex and
computationally intensive. A model can be simplified to save on computational energy.
One simplification is performing a large-eddy simulation (LES). A large-eddy simulation
filters out fine-scale turbulence, and model these small scales as flow-dependent effective
viscosity (Andersson 2012). Another simplified method is a Reynolds-averaged Navier-
Stokes (RANS) method. In these models the turbulent fluctuations are time averaged, yet
reasonable velocity averages can be simulated in these models. Anything smaller than the
grid size is not included in these models, such as break-up and coalescence of bubbles
(Andersson 2012). Figure 4 shows how fluid dynamics calculations are set up.
Area of Study

The area of study for this project was a stretch of the Logan River in an area locally known as “the island”. This river is in Logan, Utah in the United States. The area is focused on a canal diversion called the Crockett Canal Diversion. According to Logan City, during high runoff events the river is most likely to flood upstream of this particular canal diversion. For that reason, it is assumed that this diversion structure could be the reason for the flooding.

This study has high potential benefits for Logan City. Prior to the project start coordination with Logan City was key to determining a starting location and objective. According to Logan City the Crockett Canal Diversion is a potential cause of flooding during periods of high spring runoff. The obstruction caused by the diversion can cause the river to become backed up and flood. Debris being caught in the diversion has also been a cause of flooding in the past, but will not be looked at in this study.
An objective of this study is to determine the effect of the diversion structure on upstream flooding. Results could be used to find the best solution to the flooding. Changes in the weir type, or weir design may reduce flooding. Hydraulic modeling allows for changes to be made more easily than in the field. For example, the weir coefficient can be adjusted in HEC-RAS to try and improve flood conveyance in the river channel.

The area along this stretch of river is considerably developed with single family homes making up a majority of the area. There is also a park and a school in the area. A major flood would cause significant damage, therefore, knowing flooding extents is important for reducing risk to lives and properties. Figure 5 shows the area of study.
Figure 5. Area of study with extents of CFD model outlined. Flow moves right to left (Google Earth).

The topography of the area consists of a higher north bank and a lower south bank. This leads to the majority of flooding occurring on the south side of the river. Flood waters exit the river upstream of the diversion structure and enter again to the south west. In this study the Crockett Canal is neglected as a flow path. This is due to the fact that
flooding occurs in the spring before water is being diverted into the canal for irrigation. In this study all of the flow in the river is being diverted down the main river channel.

**Terrain Data**

All three tests used the same set of terrain data. Light Detection and Ranging (LiDAR) elevation data was use from the Utah Geospatial Resource Center (UGRC). The data forms a digital elevation model (DEM) which can be used in HEC-RAS projects. The terrain data has not been modified in any way for a hydraulic analysis. In some areas there are sinks in the terrain which can cause inconsistencies in the flood mapping as will be seen later. Figure 6 shows the DEM in HEC-RAS with contours, and elevation shading.

HEC-RAS and Star-CCM+ display the terrain in two different ways. HEC-RAS imports a raster file and displays the terrain as seen in Figure 6. The model can generate contours and the display can be adjusted to show relief. Star-CCM+ does not use any type of terrain model. Instead, Star-CCM+ uses a 3D model solid. The terrain data was processed in AutoCAD Civil 3D to achieve compatibility. The DEM was changed in to a triangular irregular network (TIN), then exported as a 3D solid to be used in Star-CCM+. The two programs ultimately use the same base terrain and displays them differently. Upon searching there is no obvious evidence of loss of terrain quality from this process, nor does that fall within the scope of this project.
Friction plays an important role in all hydraulics modeling. For the HEC-RAS portion of this study Manning’s $n$ coefficient is used to quantify friction loss. Manning’s $n$ is an empirical coefficient, found to be characteristic of the surface roughness alone. Such an approximation might be reasonable as long as the water is not too shallow or the channel too narrow (Chanson 2004).

For the area of study Manning’s $n$ was determined using empirically derived values. These values were selected based on recommended values from Chow (Chow 1959). In the channel a $n = 0.05$, in the overbanks $n = 0.07$, in the flood plain $n = 0.08$. These coefficients are used in equation 7 to calculate conveyance.

The CFD model uses a roughness height assigned to each surface to account for
friction and the losses associated with said friction. Roughness heights were assigned based on visual analysis of the Logan River and surround areas. The roughness heights assigned to each surface are given in Table 2.

**Table 2.** Roughness heights.

<table>
<thead>
<tr>
<th>Surface</th>
<th>Roughness Height (ft)</th>
</tr>
</thead>
<tbody>
<tr>
<td>River Bottom</td>
<td>0.35</td>
</tr>
<tr>
<td>River Bank</td>
<td>0.5</td>
</tr>
<tr>
<td>Flood Plain</td>
<td>0.25</td>
</tr>
</tbody>
</table>

**Weir**

The weir at the Crockett Canal diversion is the primary variable of this study. The weir is a basic broad crested weir with four slots for stop logs. In this study the weir is modeled with no stop logs. This is because the stop logs are used to divert water into the Crockett Canal. Peak flows are most likely to occur in the spring, before water is being diverted for irrigation in mid to late summer. The flow does not approach the weir perpendicularly, but at an angle. All three models are capable of modeling a weir, but do so in different ways. A picture of the weir is shown in Figure 7.
1D HEC-RAS models a weir as a modified cross section. Weir dimensions are entered as (X, Y) coordinates on a grid and the weir is added as a cross section. HEC-RAS uses the standard weir equation to calculate flow over the weir (USCAE 2016):

$$Q = C L H^\frac{3}{2}$$  \hspace{1cm} (10)

where $Q =$ Total flow over the weir, $C =$ discharge coefficient for weir flow, $L =$ effective length of weir, $H =$ weir energy head.

2D HEC-RAS uses a similar process to 1D HEC-RAS to model weirs. A single cross section is established and (X, Y) coordinates are entered to establish the weir dimensions. 2D HEC-RAS also uses the standard weir equation (equation 10) to solve for
flow over a weir. The weir geometry is shown in Figure 8.

The CFD model does not use a weir coefficient. The CFD model provides the needed variables to calculate a weir coefficient by back calculating the weir equation.

![Weir cross section in HEC-RAS.](image)

**Figure 8.** Weir cross section in HEC-RAS.

Both HEC-RAS models require the user to input a weir coefficient. This is a shortcoming of 1D and 2D modeling. The exact coefficient of the weir in this study is unknown due to the unique nature of this weir. As an initial estimate, a value of 2.6 was used for both weir coefficients. This value comes from the low end of what the HEC-RAS Hydraulic Reference Manual recommends for a broad crested weir.

Additionally, the model was run with weir coefficients of 3.1, 3.6, and 4.1 to investigate model sensitivity. While these numbers do not reflect the weir coefficient of a broad crested weir. They represent a more efficient weir which could improve flooding conditions. The weir length was not changed, and the weir energy head is automatically calculated in HEC-RAS.

The 3D CFD model uses 3D objects to build a geometry. This allows the user to
build 3D objects using a CAD program and import the objects into STAR-CCM+. Using a CAD program ensures the desired level of precision can be achieved. The author drew a 3D model in AutoCAD based on design drawings, and field measurements. The weir model also includes wing walls and a bridge deck that spans the length of the weir, for adding and removing stop logs. A weir coefficient is not used in the 3D model. The 3D model does not solve the weir equation. The 3D weir model is shown in Figure 9.

![3D weir model](image)

**Figure 9.** 3D weir model.

**Flow Data**

The flow of the Logan River is primarily controlled by melting snowpack in the Bear River mountain range. The portion of the river located up Logan Canyon is a steep mountain stream with low sinuosity. Below Logan Canyon the river enters the valley bottom where it begins to meander before entering Cutler Reservoir. The area of study lies approximately two miles from the mouth of Logan Canyon and has characteristics more similar to the steep mountain stream.

A USGS stream gauging station is located at the mouth of Logan Canyon. At this location the Logan River watershed is approximately 214 square miles (USGS 2021). A
2011 FEMA flood insurance study (FIS) was used to determine flow rates and return intervals near the area of study. Table 3 shows the flow rates that were tested in this study.

**Table 3.** Tested Flow Rates.

<table>
<thead>
<tr>
<th>Flow (cfs)</th>
<th>Approximate Return Interval (years)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1000</td>
<td>5</td>
</tr>
<tr>
<td>1670</td>
<td>10</td>
</tr>
<tr>
<td>2500</td>
<td>100</td>
</tr>
</tbody>
</table>

**1D Model Setup**

The 1D model was built and tested using HEC-RAS 5.0.7. A river channel is established and delineated and then cross sections are drawn using the DEM. Cross section spacing varies depending on the desired level of detail. In a straight reach of channel, the cross sections can be spaced further apart than at a bridge or control structure where more detail is necessary. Water surface profiles are calculated at each of these cross sections. A cross section is shown in Figure 10.
The 1D model was solved using steady state parameters. A flow rate was established and a water surface elevation (WSE) is calculated for each cross section at the given flow rate. The tested flow rates correlate to possible flow rates on the Logan River.

The banks along a majority of the river are higher than the floodplain, creating an almost levee like structure. This caused significant problems in preliminary 1D testing. When a WSE reached the height of the bank, the WSE of the remaining floodplain instantly filled to the same level. 1D modeling does not allow for any lateral flow. To mitigate this error the river and floodplain were modeled as two separate “channels” with a second set of cross sections drawn across the floodplain. HEC-RAS allows a lateral flow structure to pass flow from one reach to another. Lateral flow structures can be used to model levees. In this study the lateral flow structure allowed for water to pass from the main channel to the floodplain incrementally. These are some of the shortcomings of 1D modeling which will be seen in the results later.

1D models use various assumptions to simplify their process. Flow is calculated...
based only on cross section properties. 1D modeling is only appropriate for well-defined constant flow paths (Toombes and Chanson 2011). Areas with complex flow paths, such as flood plains, are modeled better with 2D or 3D methods. 1D models assume the channel slope is small, less than 1:10. When the channel bottom is steeper than 1:10 HEC-RAS uses the momentum equation, or other empirical equations to account for rapidly varying flow in these sections (USCAE 2016). 1D modeling also uses Manning’s n to account for friction. Manning’s n is dependent on the hydraulic radius and will therefore change with discharge, even though the model has one fixed Manning’s n for the flow area (Toombes and Chanson 2011).

Boundary conditions are required for any stretch of river not connected to another stretch of river (USCAE 2016). In the 1D models a flow rate is used as the upstream boundary condition, downstream a normal depth is used as a boundary condition. The 1D geometry is shown in Figure 11.

In the 1D model cross sections are spaced approximately 100 ft apart, with shorter spacing at the weir. The FEMA map cross sections are spaced at intervals of approximately 1000 ft. The author used the shorter distance to capture more terrain detail in the study area. Various empirical equations exist for choosing cross section spacing (USCAE 2016).
2D Model Setup

The 2D portion of the study was also conducted using HEC-RAS. 2D models calculate water depths and velocities across a grid or mesh that defines the topographic information (Toombes and Chanson 2011). The Two-dimensional momentum equations solve for fluid flow across the boundaries of the grid. A 2D geometry and mesh is shown in Figure 12.

Figure 11. 1D Geometry.
The computational mesh represents the underlying topography. In the case of this study the same DEM from LiDAR data was used to define topography. A major advantage of 2D modeling over 1D modeling is the ability to model lateral flows. Where a 1D model only models flow in a channel perpendicular to a cross section, 2D modeling is capable of modeling flows in multiple directions.
2D models are susceptible to problems when calculating viscous shear and friction. Viscosity calculations can be affected by water column effects, when the depth of the vertical scale of the water column approaches the horizontal scale (Toombes and Chanson 2011). Roughness is also calculated different for 2D models than 1D models. Roughness is not included in the walls of the 2D model. Manning’s roughness coefficients were developed for 1D flow (Toombes and Chanson 2011).

Because computational cells are dependent on the topography the model will fill low lying areas first. If a sink or a small channel is contained inside a cell the water will fill these features first and then proceed to spill into adjacent cells, or continue filling the current cell.

The mesh is divided into polygons which are then solved as a finite element analysis (Toombes and Chanson 2011). The polygons do not need to be square, and are allowed up to eight sides. The computer is capable of solving polygons with more sides, but to save on computational complexity HEC-RAS has limited the polygons to eight sides (USCAE 2016). Water velocities are calculated at cell faces, water depths are calculated at cell centers (USCAE 2016). The 2D flow area in this study contains 19504 cells in the mesh.

The mesh was refined by incrementally decreasing the mesh size. As the mesh was refined the extents of the water surface increased also. The mesh size was decreased until the water surface extents were within 1% of two consecutive mesh sizes.

The cell size should generally be based on the slope and detail of the topography. Steeper areas, or areas where more detail is required (weirs, levees, embankments etc.) should have smaller cell areas. Open or flat areas should have a larger cell size (USACE
In this study the base cell size was set at 10 ft across and 10 ft high. Not all cells were square, along breaklines and mesh boundaries cells are polygons, the smallest cell size was 22.36 ft$^2$ and the largest cell was 198.35 ft$^2$. HEC-RAS literature recommends making sure cell accurately capture the terrain. The cells must be sized such that the change in water surface is not abrupt. In flat areas with a gradual change in water surface, larger cells are acceptable, in steep areas, smaller cells may be necessary to capture changes in water surface (USCAE 2016).

In this study the diversion structure, a broad crested weir, was modeled as a 2D boundary area. The weir geometry was defined and a break line was used to better align the mesh with the weir. The boundary conditions for the model are a flow hydrograph at the upstream end of the model and a normal depth at the downstream end of the model. A detailed mesh is shown in Figure 13.

![Figure 13. 2D Mesh detail at weir location.](image)

**3D Model Setup**

HEC-RAS is not capable of performing a full 3D model. This requires a program
built specifically for 3D fluid mechanics. STAR-CCM+, a commercially available double precision CFD code, was used in this study to perform the CFD modeling.

A 3D CFD model is capable of modeling the vertical components of flow. A 3D model requires a 3D geometry that can be imported in the CFD model as a 3D solid. The DEM used in the HEC-RAS modeling was imported into AutoCAD Civil 3D where it was extruded into a 3D solid. Creating the solid in Civil 3D also allowed for a detailed weir to be drawn and added to the solid so the entire geometry was imported in the CFD model. Figures 14 and 15 show the 3D solid that was used in the CFD model.
Figure 14. 3D study area. Note the thickness of the object.

Figure 15. 3D weir joined with the surface.
After importing the 3D object into STAR-CCM+ the geometry was modified to correct abnormalities that would affect the CFD calculations. The volume was also trimmed to help optimize the needed space and therefore computational demand.

The 3D object was divided into a mesh similar to a 2D model. The mesh divides the entire volume into individual cells. The fluid dynamics equations are solved across these cells. The size of the cells is dependent on the level of detail required. The cells are much finer around the weir than they are out further in the floodplain.

The three meshing tools were used: the prism layer mesh, the surface remesher, and the trimmed mesher. The prism layer mesh generates orthogonal prismatic cells next to wall surfaces or boundaries. The surface remesher retriangulates the surface to optimize the volume mesh. The trimmed mesher generates a grid for mesh generation (Siemens 2020). A good meshing practice is to have 6 to 10 mesh cells around any fluid path or structure in the model (Sharp 2022).

The mesh for this study contained 10,729,847 cells. A mesh convergence was performed similar to the 2D model. To ensure mesh independence the model is run and then the mesh is refined. The model is run again and the results compared. This process is repeated until the results of interest are within the uncertainty of the solution for two different meshes. This ensures the solution is independent of the mesh size. The smaller of the meshes should be used after that to save on computation time (LEAP 2012). The surface mesh at the weir is shown in Figure 16. The Volume mesh is shown in Figure 17.
Since CFD models are capable of modeling a variety of fluids the physics must be properly adjusted for each model. This project primarily models water flow in the river.
and floodplain, however the model also takes the air above the water into account. The
water properties reflected water at 45 degrees Fahrenheit (density of 62.42113 lb/ft\(^3\) and
dynamic viscosity of 1.3998420922773254E-8 atm-s). Air pressure was set at 1 atm. The
Reynolds-Averaged Navier-Stokes method was used to solve for the momentum and
pressure equations. The K-Epsilon Turbulence model was used for turbulence closure.
This turbulence model is sufficient because small scale turbulence is negligible, such as
water and air mixing, air bubbles in the water, and small-scale vortices. Initial conditions
were set as a water level and then a flow rate was established at the model inlet and the
water surface throughout the model was allowed to stabilize.

The boundary conditions for the model were a mass flow rate at the river inlet.
The rate was adjusted with each model version. The surfaces in the model were no-slip
walls. Each surface was assigned a different roughness height based on the material or
ground cover associated with the surface. Those values are shown in Table 2. The other
physics models used were: gradients, gravity, implicit unsteady, K-Epsilon turbulence,
multiphase, multiphase equation of state, multiphase interaction, realizable K-Epsilon
two layer, Reynolds Averaged Navier-Stokes (RANS), segregated flow, solution
interpolation, three dimensional, turbulent, two layer all y+ wall treatment, volume of
fluid (VOF), and wall distance.
CHAPTER IV
RESULTS AND DISCUSSION

This section will present the findings from the various tests and discuss some of the observed results. The section is organized by the flow rates tested. In each section the inundation area upstream of the weir, weir flow, and hydraulic grade line (HGL) are presented for the 1D, 2D and 3D model. Rating curves for the weirs of each test are also presented. In addition to the flow rate results, a section is also included on the results of the weir sensitivity in the 2D model. As well as a channel friction sensitivity analysis for the 2D model.

The measured water surface area only includes the area in the channel and floodplain upstream of the weir. This was done to observe the change in area that is influenced by the weir. The area downstream of the weir is primarily a result of water flowing across the flood plain as a result of the upstream flooding. An example of the area measured is shown in Figure 18. The line drawn across the area represents the dividing line, the area above (on the upper part of the figure) is the area of the water surface measured.
Figure 18. Water surface area with dividing measurement line. The area above the line is the water surface area measured.

**1000 CFS Results**

The 1000 cfs tests across all three model types saw water spilling over the banks and into the floodplain. The water surface areas are shown in Figures 19, 20, and 21 for the 1D, 2D, and 3D model respectively.
Figure 19. 1D 1000 cfs flood extents.
Figure 20. 2D 1000 cfs flood extents.
Figure 21. 3D 1000 cfs flood extents.
Water first leaves the channel in two locations on all three models. The first location near the upstream boundary is 1100 ft upstream of the weir, the second location is 430 ft upstream of the weir. These locations are the only two empty lots along the south bank of the river. These undeveloped lots have lower banks than the adjacent lots with houses. These lower banks allow the flow to leave the channel before any other location. The inundated areas above the weir are shown in Table 4.

**Table 4.** 1000 cfs inundated area comparison.

<table>
<thead>
<tr>
<th>Model Type</th>
<th>Area (ft²)</th>
<th>Area (ac)</th>
<th>% Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>1D</td>
<td>126,331</td>
<td>2.90</td>
<td>-</td>
</tr>
<tr>
<td>2D</td>
<td>105,846</td>
<td>2.43</td>
<td>-16%</td>
</tr>
<tr>
<td>3D</td>
<td>174,551</td>
<td>4.01</td>
<td>38%</td>
</tr>
</tbody>
</table>

The flow depths of water through the weir section for each model are shown in Figures 22, 23, and 24. The flow through the weir increases for the higher fidelity models. The depth of flow increases between the 1D and 2D model, but decreases between the 2D and 3D model. The 3D model is more efficient at passing the flow over the weir. This suggests the weir may have a higher discharge coefficient than the original value assigned which was for a broad crested weir. Any flow not passing through the weir was passing through the floodplain, this was found based on continuity. The profiles are based on the geometry used for each model. The 3D CFD model allows the entire structure to be modeled, not just the weir. Therefore, the 3D profile shows the vertical stop log rails and bridge deck, and how the water interacts with these objects. Water can also be seen passing around the weir. The water profile is indicated by the blue line, in Figure 24 the line can be seen outside of weir extents, this flows around the weir.
Figure 22. 1D 1000 cfs weir profile.

Figure 23. 2D 1000 cfs weir profile.

Figure 24. 3D 1000 cfs weir profile.
The flow through each weir at 1000 cfs is compared in Table 5. Any water not flowing through the weir flows around the weir and through the floodplain.

**Table 5.** 1000 cfs weir flow summary.

<table>
<thead>
<tr>
<th></th>
<th>Q (cfs)</th>
<th>WSE (ft)</th>
<th>Depth (ft)</th>
<th>% Difference-Depth</th>
</tr>
</thead>
<tbody>
<tr>
<td>1D</td>
<td>781.6</td>
<td>4585.4</td>
<td>3.2</td>
<td>-</td>
</tr>
<tr>
<td>2D</td>
<td>999.8</td>
<td>4586.5</td>
<td>4.3</td>
<td>34%</td>
</tr>
<tr>
<td>3D</td>
<td>972.4</td>
<td>4585.3</td>
<td>3.1</td>
<td>-4%</td>
</tr>
</tbody>
</table>

The hydraulic grade lines for the 1000 cfs tests are plotted in Figure 25. The hydraulic grade line shows the elevation of the water in the channel across the length of the river. The hydraulic grade line can show areas of the river that are controlled by the weir. The weir will increase the depth directly up stream of the weir. If the water surface elevation is high enough, the water will over flow the banks and flow around the weir. The hydraulic grade line increased between the 1D and 2D models, but the 3D model yielded a lower hydraulic grade line in most locations. This is due to the 3D model measuring elevations at specific points in the river, there is no smoothing or averaging of the values.
Figure 25. HGL profile lines, 1000 cfs.

1670 CFS Results

The 1670 cfs results are presented in the following section. The inundated areas are shown in Figures 26, 27, and 28.
Figure 26. 1D 1670 cfs flood extents.
Figure 27. 2D 1670 cfs flood extents.
Figure 28. 3D 1670 cfs flood extents.
The inundated areas above the weir are shown in Table 6.

**Table 6.** 1670 cfs inundated areas comparison.

<table>
<thead>
<tr>
<th>Model Type</th>
<th>Area (ft$^2$)</th>
<th>Area (ac)</th>
<th>% Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>1D</td>
<td>200,785</td>
<td>4.61</td>
<td>-</td>
</tr>
<tr>
<td>2D</td>
<td>265,180</td>
<td>6.09</td>
<td>32%</td>
</tr>
<tr>
<td>3D</td>
<td>277,115</td>
<td>6.36</td>
<td>38%</td>
</tr>
</tbody>
</table>

The weir profiles for the 1670 cfs flow are shown in Figures 29, 30, and 31 below.
Figure 29. 1D 1670 cfs weir profile.

Figure 30. 2D 1670 cfs weir profile.

Figure 31. 3D 1670 cfs weir profile.
The flow through each weir at 1670 cfs is compared in Table 8. Any water not flowing through the weir flows around the weir and through the floodplain.

**Table 7.** 1670 cfs weir flow summary.

<table>
<thead>
<tr>
<th></th>
<th>Q (cfs)</th>
<th>WSE (ft)</th>
<th>Depth (ft)</th>
<th>% Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>1D</td>
<td>1017.8</td>
<td>4586.0</td>
<td>3.8</td>
<td>-</td>
</tr>
<tr>
<td>2D</td>
<td>1557.9</td>
<td>4587.9</td>
<td>5.7</td>
<td>50%</td>
</tr>
<tr>
<td>3D</td>
<td>1345.6</td>
<td>4585.8</td>
<td>3.6</td>
<td>-5%</td>
</tr>
</tbody>
</table>

The hydraulic grade lines for the 1670 cfs tests are plotted in Figure 32.

**Figure 32.** 1670 cfs hydraulic grade lines.

**2500 CFS Results**

The 2500 cfs results are presented in the following section. The inundated areas are shown in Figures 33, 34, and 35.
Figure 33. 1D 2500 cfs flood extents.
Figure 34. 2D 2500 cfs flood extents.
Figure 35. 3D 2500 cfs flood extents.
The inundated areas above the weir are shown in Table 8.

**Table 8.** 2500 cfs inundated areas comparison.

<table>
<thead>
<tr>
<th>Model Type</th>
<th>Area (ft²)</th>
<th>Area (ac)</th>
<th>% Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>1D</td>
<td>260,663</td>
<td>5.98</td>
<td>-</td>
</tr>
<tr>
<td>2D</td>
<td>382,149</td>
<td>8.77</td>
<td>47%</td>
</tr>
<tr>
<td>3D</td>
<td>386,100</td>
<td>8.86</td>
<td>48%</td>
</tr>
</tbody>
</table>

The weir profiles for the 2500 cfs flow are shown in Figures 36, 37, and 38 below.
Figure 36. 1D 2500 cfs weir profile.

Figure 37. 2D 2500 cfs weir profile.

Figure 38. 3D 2500 cfs weir profile.
The flow through each weir at 2500 cfs is compared in Table 9. Any water not flowing through the weir flows around the weir and through the floodplain.

**Table 9.** 2500 cfs weir flow summary.

<table>
<thead>
<tr>
<th></th>
<th>Q (cfs)</th>
<th>WSE (ft)</th>
<th>Depth (ft)</th>
<th>% Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>1D</td>
<td>1205.9</td>
<td>4586.5</td>
<td>4.3</td>
<td>-</td>
</tr>
<tr>
<td>2D</td>
<td>1847.6</td>
<td>4588.6</td>
<td>6.4</td>
<td>48%</td>
</tr>
<tr>
<td>3D</td>
<td>1658.1</td>
<td>4586.2</td>
<td>4.0</td>
<td>-7%</td>
</tr>
</tbody>
</table>

The hydraulic grade lines for the 2500 cfs tests are plotted in Figure 39.

**Figure 39.** 2500 cfs hydraulic grade lines.
FEMA Flood Map

The current flood map developed by FEMA is shown in Figure 40. This map was published in 2011 and is the current standard used to determine flood insurance needs. The map was created using a 1D model. The light blue color represents the extents of the 1670 cfs flood, and the brown represents the extents of the 2500 cfs flood.

The approximate inundated areas above the weir for the two mapped floods are 10.76 acres and 11.85 acres for the 1670 cfs and 2500 cfs floods respectively as shown in Table 10 and 11. These values are larger than the values calculated in this study suggesting that FEMA uses conservative modeling methods. It is in FEMA’s best interest to be conservative in their assessment to ensure more people are protected under flood insurance.

**Table 10.** 1670 cfs Inundated area comparison.

<table>
<thead>
<tr>
<th>Model Type</th>
<th>Area (ft²)</th>
<th>Area (ac)</th>
<th>% Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>1D</td>
<td>200,785</td>
<td>4.61</td>
<td>-</td>
</tr>
<tr>
<td>2D</td>
<td>265,180</td>
<td>6.09</td>
<td>32%</td>
</tr>
<tr>
<td>3D</td>
<td>277,115</td>
<td>6.36</td>
<td>38%</td>
</tr>
<tr>
<td>FEMA</td>
<td>468,915</td>
<td>10.76</td>
<td>134%</td>
</tr>
</tbody>
</table>

**Table 11.** 2500 cfs Inundated area comparison.

<table>
<thead>
<tr>
<th>Model Type</th>
<th>Area (ft²)</th>
<th>Area (ac)</th>
<th>% Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>1D</td>
<td>260,663</td>
<td>5.98</td>
<td>-</td>
</tr>
<tr>
<td>2D</td>
<td>382,149</td>
<td>8.77</td>
<td>47%</td>
</tr>
<tr>
<td>3D</td>
<td>386,100</td>
<td>8.86</td>
<td>48%</td>
</tr>
<tr>
<td>FEMA</td>
<td>516,334</td>
<td>11.85</td>
<td>98%</td>
</tr>
</tbody>
</table>
Figure 40. FEMA flood map.

Rating Curves

The rating curves represent the efficiency of the weir. Pre-determined weir coefficients of 2.6 were used for the HEC-RAS models. Using the results from the 3D model a weir coefficient was calculated for each of the flow rates. The rating curves for
the three models are shown in Figure 41. The 3D rating curve is smaller due to the model only being run with the three different flow rates.

![Rating Curves](image)

**Figure 41.** Rating curve comparison.

**2D Weir Coefficient Sensitivity**

In addition to the various model types, tests were also done on the effect of changing the weir coefficient in the 2D HEC-RAS model. HEC-RAS requires a weir coefficient to solve the weir equation. Using a more efficient weir coefficient would help determine if a different type of weir will help improve flow conveyance and therefore decrease the flooding upstream.

Adjusting the weir coefficient will further show if the weir is the main cause of flooding or if the problem lies elsewhere.

To assess the change in flooding a different model was built using the same flow
rates and base geometry. The only variable that changed was the weir coefficient. The various weir coefficients are shown below in Table 12.

**Table 12.** Weir coefficients.

<table>
<thead>
<tr>
<th>Weir Coefficients</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>2.6 (base)</td>
<td>3.1</td>
</tr>
<tr>
<td></td>
<td>3.6</td>
</tr>
<tr>
<td></td>
<td>4.1</td>
</tr>
</tbody>
</table>

To compare the differences in weir coefficients, the maximum inundation boundary was plotted for each condition at a flow rate of 2500 cfs. An increase in weir efficiency should show a decrease in water surface area. The rating curve data for each weir coefficient were also plotted on a chart.
For the base condition with a weir coefficient of 2.6, the maximum inundation area was 942,836 ft$^2$ shown in Figure 42 below.

**Figure 42.** Weir coefficient 2.6 inundation boundary.
For the weir coefficient of 3.1, the maximum inundation area was 938,864 ft² shown in Figure 43 below.

**Figure 43.** Weir coefficient 3.1 inundation boundary.
For the weir coefficient of 3.6, the maximum inundation area was 937,474 ft$^2$

shown in Figure 44 below.

**Figure 44.** Weir coefficient 3.6 inundation area.
For the weir coefficient of 4.1, the maximum inundation area was 936,655 ft$^2$ shown in Figure 45 below.

**Figure 45.** Weir coefficient 4.1 inundation area.
The inundation extents showed very little variance in area between the different weir coefficients. The results are summarized in Table 13 below.

**Table 13. Weir coefficient comparison.**

<table>
<thead>
<tr>
<th>Weir Coefficient</th>
<th>Inundation area (ft²)</th>
<th>% Different from base</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.6</td>
<td>942,836</td>
<td>-</td>
</tr>
<tr>
<td>3.1</td>
<td>938,864</td>
<td>0.42</td>
</tr>
<tr>
<td>3.6</td>
<td>937,474</td>
<td>0.57</td>
</tr>
<tr>
<td>4.1</td>
<td>936,655</td>
<td>0.66</td>
</tr>
</tbody>
</table>

The change in inundation area was less than 1% for each weir coefficient when compared to the base condition. Only minor changes in flooding occurred upstream of the weir. The flooding occurs in an almost identical manner with each weir coefficient. Water leaves the channel at low areas upstream of the weir. The weir does not appear to have an effect on the river level rising. The areas where water first exits the channel are approximately 8-9 ft higher in elevation than the weir. The crest of the weir sits at elevation 4582.2. Water is first leaving the channel at a location with an elevation of 4590. Water levels at the weir never reach this elevation, suggesting the weir is not causing the flooding at this location.

The HEC-RAS generated rating curves show improved weir efficiency as the weir coefficient increases, but this improvement is not significant enough to affect the flooding. Further indicating that the rise in channel level affects the flooding more than the presence of the weir. The rating curves are compared in Figure 46 below.
Figure 46. Rating curves for weir coefficient comparison.
**Channel Friction Sensitivity**

In order to observe the effects of friction on the model tests were performed using various Manning’s N values. The base N values were selected based on the characteristics of the channel. Additional models were created using Manning’s N values that are both higher and lower than the base values. Since the main focus was on channel friction, the N value of the floodplain remained the same for each test.

The results measured the total inundated area for the corresponding flow rate. Lower N values had consistently lower inundated areas. Higher N values had larger inundated areas.

Lower flow rates are generally affected by the change in friction than the higher flow rates. This is due to the fact that a deeper water column is influenced less by the surface friction than a shallow water column. The largest outlier was the difference between the base and low friction model for the 1670 cfs flow rate. The lower N values allow more flow to pass through the channel before spilling over the banks. The higher N values cause a faster rise in water surface elevation and the water leaves the channel sooner. The friction values, flow rates, and inundated areas are summarized in Table 14.

**Table 14.** Friction sensitivity summary.

<table>
<thead>
<tr>
<th></th>
<th>Manning’s N</th>
<th>Inundated Area</th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>1000 cfs</td>
<td>1670 cfs</td>
<td>2500 cfs</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Low Channel</td>
<td>0.04</td>
<td>87088 % -18%</td>
<td>124491 % -53%</td>
<td>279270 % -27%</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Low Banks</td>
<td>0.05</td>
<td>105846 % -4%</td>
<td>260612 % -2%</td>
<td>378921 % -1%</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Base Channel</td>
<td>0.05</td>
<td>105846 % -4%</td>
<td>265180 % -2%</td>
<td>382149 % -2%</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Base Banks</td>
<td>0.07</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>High Channel</td>
<td>0.06</td>
<td>101530 % -4%</td>
<td>260612 % -2%</td>
<td>378921 % -1%</td>
<td></td>
<td></td>
</tr>
<tr>
<td>High Banks</td>
<td>0.08</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Pros and Cons of Modeling

As modeling capabilities have improved over time each model type has shown various advantages and disadvantages.

1D modeling is more simple than higher order models. The equations used in 1D modeling allow for low computational demand and can be solved on most basic desktop computers.

The simplicity of 1D modeling comes at a cost of capabilities. 1D modeling has been the standard for many years in flood planning. However, 1D models are most accurate when there is a well-defined channel and flow paths normal to the cross sections. In areas where there is no defined channel and flow paths, such as floodplains, and alluvial fans, 1D models do not capture lateral flows that may spread across the area.

2D models do a better job at modeling lateral flows, and areas with undefined flow paths. These models are useful in predicting flood extents in areas with irregular topography.

Because of the increased capabilities of 2D models there is also an increased complexity. A 2D model of a dam breach and subsequent flood may take multiple days to run on a basic desktop computer. 2D models also require a grid convergence to ensure optimal output. 2D models are relatively new compared to 1D modeling. For this reason there has been less 2D modeling used for flood planning, but has started to be accepted and implemented by state agencies.

The added complexity of 3D CFD modeling comes with advantages and disadvantages. Greater detail comes at the cost of more time and computational resources. In this study, the different model types showed some differences and
similarities in various areas.

The case with the most obvious differences is the weir modeling. 1D and 2D models only model the weir as a cross section in the channel. A simple geometry is entered and weir properties are assigned. The 3D model allows for a detailed 3D model of the weir to be imported and tested. The 3D model uses the same meshing system and equations for the weir as the rest of the model. The weir is treated as a piece of the terrain, rather than an object with different conditions from the river channel.

The differences in weir flow are noticeable when comparing the weir profiles. The 3D model shows the entire structure and how the water flows around the structure’s components. This is obvious where the water flows past the vertical supports of the bridge deck. The 1D and 2D models do not allow for the added detail of the supports and bridge. 3D modeling shows the flow through and around the weir as seen in Figure 47.

![Figure 47. 3D flow at the weir.](image)

In a detailed analysis of a weir or other hydraulic structure 3D modeling is capable of showing vortices and recirculation’s that may occur. In this particular reach of
river 3D modeling could be used to observe the effects of changing the type of weir.

The flooding extents were noticeably different between the 2D and 3D models. 3D modeling will, by nature, provide more detailed results than the 2D model. However, in many cases a well refined 2D model will be capable of providing sufficient detail in flooded areas. 2D models are capable of generating flood areas and depth versus velocity plots. These are both valuable in disaster planning and prevention. A 2D model may provide sufficient detail making a 3D model unnecessary for some cases.
CHAPTER V
CONCLUSIONS

Flood modeling can be done in a variety of ways. What type of model is used depends on factors such as detail required, terrain complexity, and computational resources to name a few. All modeling is based on the basic fluid dynamics equations. More powerful models are capable of solving increasingly complex equations and make models with greater detail.

In this study 1D, 2D, and 3D models were developed for a specific stretch of the Logan River. Each of the models used the same base topography. Each model uses different governing equations to solve for flow. 1D models are relatively simple and can be performed quickly, while a 3D CFD model may take days to run due to its complexity. These are dependent on computational power available. 1D models can be solved easily on a standard desktop computer. To make 3D models economical a supercomputer sometimes with hundreds of processors is necessary to complete a model. Even this can take multiple days. This extra computational power comes at a greater cost.

The objective of the study was to observe the differences in each of these three models. Another objective given by Logan City was to determine the effect of the irrigation diversion structure on flooding upstream. Finally, a second 2D study was performed to observe the sensitivity of the weir coefficient.

This study found the 1D HEC-RAS model to not be ideal for the river reach being studied. Lateral flows are not performed well in 1D models and are better suited for 2D modeling. The 2D HEC-RAS models gave better results. The 2D model showed water leaving the channel upstream of the diversion structure. The location where flooding
begins was far enough upstream that there was no indication of the diversion structure causing the flooding. Hydraulic grade lines and water surface elevations do not show significant water accumulating behind the weir. If the weir were the primary factor of flooding the water surface would rise behind the weir and flow over the banks. Additional evidence is seen in the 1000 cfs condition. The weir profiles show the water level below the top of the weir walls, but there is still water in the floodplain. In higher flow cases the water is above the top of the weir walls. Water that flowed around the diversion structure primarily came from upstream, and not immediately before the structure. The 3D Star-CCM+ model showed the water leaving the channel at the same locations. The 3D model also gave very detailed results of the weir itself. Water can be seen flowing around the weir in a low point in the topography. This inconsistency was not visible in a less detailed model. The 3D model provided valuable insight to the performance of the structure, these results were not possible in the 1D and 2D models.

The sensitivity to the weir coefficient was investigated using 2D HEC-RAS models. All of the model parameters were kept the same as the original 2D models except for the weir coefficient. Increasing the weir coefficient increased the weir efficiency, but ultimately did not significantly improve the flooding results.
REFERENCES


Cook, A. C. 2008. “Comparison of one-dimensional HEC-RAS with two-dimensional FESWMS model in flood inundation mapping.” Graduate School, Purdue University, West Lafayette.


FEMA. 2011. “FEMA FIS.”


Sharp, Z. 2022.

Siemens. 2020. “Simcenter STAR-CCM+.”


USACE. 2016b. “HEC-RAS River Analysis System.”


