

MODELING OF RIVER FLOW AND FLOW DYNAMICS NEAR JUNCTIONS

A THESIS SUBMITTED TO
THE GRADUATE SCHOOL OF NATURAL AND APPLIED SCIENCES
OF
MIDDLE EAST TECHNICAL UNIVERSITY



BY
TUTKU EZGİ YÖNTER

IN PARTIAL FULFILLMENT OF THE REQUIREMENTS
FOR
THE DEGREE OF MASTER OF SCIENCE
IN
CIVIL ENGINEERING

APRIL 2023

Approval of the thesis:

**MODELING OF RIVER FLOW AND FLOW DYNAMICS NEAR
JUNCTIONS**

submitted by **TUTKU EZGİ YÖNTER** in partial fulfillment of the requirements
for the degree of **Master of Science in Civil Engineering, Middle East Technical
University** by,

Prof. Dr. Halil Kalıpçılar
Dean, Graduate School of **Natural and Applied Sciences**

Prof. Dr. Erdem Canbay
Head of the Department, **Civil Engineering**

Assoc. Prof. Dr. Ali Ercan
Supervisor, **Civil Engineering, METU**

Examining Committee Members:

Prof. Dr. Zafer Bozkuş
Civil Engineering, METU

Assoc. Prof. Dr. Ali Ercan
Civil Engineering, METU

Prof. Dr. A. Burcu Altan Sakarya
Civil Engineering, METU

Assoc. Prof. Dr. Elif Oğuz
Civil Engineering, METU

Asst. Prof. Dr. Meriç Yılmaz
Civil Engineering, Atılım University

Date: 27.04.2023



I hereby declare that all information in this document has been obtained and presented in accordance with academic rules and ethical conduct. I also declare that, as required by these rules and conduct, I have fully cited and referenced all material and results that are not original to this work.

Name Last name : Tutku Ezgi Yöner

Signature :

ABSTRACT

MODELING OF RIVER FLOW AND FLOW DYNAMICS NEAR JUNCTIONS

Yönter, Tutku Ezgi
Master of Science, Civil Engineering
Supervisor: Assoc. Prof. Dr. Ali Ercan

April 2023, 132 pages

In river networks, confluences are critical components where their hydrological, geomorphological, and ecological properties substantially impact the downstream characteristics of a river. Particular attention should be paid to the flow behavior in these regions since they are correlated with substantial alterations in flow dynamics, sediment transport/deposition, and riverbed morphology. The primary purpose of this study is to investigate river confluences by a) using the 1D hydraulic model, HEC-RAS, to understand large-scale model behavior, and b) utilizing the 3D CFD model, Flow-3D, to understand the flow behavior comprehensively near the confluence area.

In the HEC-RAS model, a sensitivity analysis was carried out on the Ohio-Mississippi confluence, and hypothetical cases were generated and investigated by perturbing model parameters. A case study was also modeled at a nearly 9 km river reach with observed inputs to compare the model's capability in simulating a real case scenario with simple information.

In the 3D CFD analysis, Flow-3D was utilized. After performing a mesh independence analysis, the performance of *standard* $k - \varepsilon$, $k - \omega$, and *RNG* $k - \varepsilon$ turbulence models was investigated with a 90° confluence channel geometry. After defining the most appropriate turbulence model and mesh size, the flow behavior at different confluence angles, 90°, 60°, and 30°, was investigated in the equilibrium flow state. Then, the effects introduced by flow ratio on flow dynamics were examined when the confluence angle is 90°. Specific attention was given to variables including turbulent kinetic energy, surface velocity, depth-averaged velocity, and Froude Number in plan view flow analyses; velocity magnitude, secondary velocity, and turbulent kinetic energy in cross-sectional flow analyses; velocity, Froude Number, flow depth, total hydraulic head, and turbulent kinetic energy in point location analyses. The effects of four discharge ratios, 0.429, 0.500, 0.600, and 0.667, were investigated. Consistent with the literature, it was found that the length and width of the recirculation zone decrease as the confluence angle decreases. Additionally, the magnitudes of the secondary velocities become more significant as the confluence angle increases from 30° to 90°. Furthermore, it was perceived that the flow ratio affects the features of the recirculation zone and influences the flow dynamics downstream of the confluence.

Keywords: Numerical Modeling, HEC-RAS, CFD, River Confluence, Confluence angle, Sensitivity Analysis, Flow-3D, Discharge Ratio

ÖZ

NEHİR AKIMININ VE NEHİR BİRLEŞİMLERİNDEKİ AKIŞ DİNAMİKLERİNİN MODELLENMESİ

Yöner, Tutku Ezgi
Yüksek Lisans, İnşaat Mühendisliği
Tez Yöneticisi: Doç. Dr. Ali Ercan

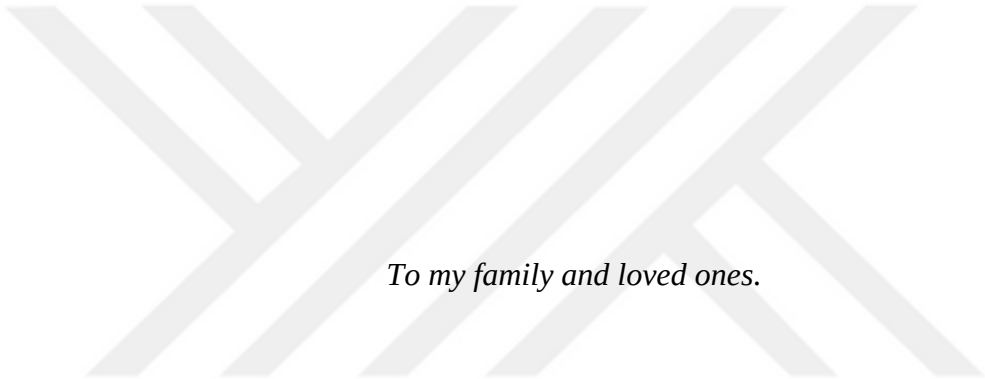
Nisan 2023, 132 sayfa

Akarsu ağlarındaki birleşme noktaları, hidrolojik, jeomorfolojik ve ekolojik özellikleriyle akışın mansap kısımları üzerinde önemli etkilere sahip kritik bileşenlerdir. Bu bölgelerdeki akış davranışı titiz bir dikkatle incelenmelidir, çünkü bu bileşenler, akış dinamiğinde, sedimentasyon taşınmasında/birikiminde ve yatak morfolojisinde önemli değişikliklerle ilişkilidir. Bu çalışmanın temel amacı, büyük ölçekli model davranışını anlamak için 1 boyutlu HEC-RAS hidrolik modelini kullanarak nehir birleşmelerini araştırmak ve daha sonra 3 boyutlu CFD modeli olan Flow-3D'yi kullanarak birleşme bölgesindeki akış dinamiklerini kapsamlı bir şekilde anlamaktır.

HEC-RAS modeli kullanılarak, Ohio-Mississippi birleşmesinde hassasiyet analizi yapılmış, model parametreleri pertürbe edilerek varsayımsal durumlar oluşturulmuş ve incelenmiştir. Ardından, gözlem verileri kullanılarak yaklaşık 9 km'lik bir nehir bölümünde bir durum çalışması modellenmiştir. Burada, HEC-RAS modelinin gerçek bir durum senaryosunu basit bilgilerle simüle etme yeteneği incelenmiştir.

3 boyutlu CFD analizinde Flow-3D kullanılmıştır. Ağ bağımsızlık analizi yapıldıktan sonra *standart* $k - \varepsilon$, $k - \omega$ ve *RNG* $k - \varepsilon$ türbülans modellerinin performansı, 90° birleşim açılı kanal geometrisi ile incelenmiştir. En uygun türbülans modeli ve ağ boyutu belirlendikten sonra, 90°, 60° ve 30° olmak üzere farklı birleşim açılarından kaynaklanan akış davranışı, kararlı akış durumunda incelenmiştir. Daha sonra, akış oranının akış dinamikleri üzerindeki etkileri, birleşme açısı 90° olan durumda araştırılmıştır. Plan görünümlü akış analizlerinde türbülanslı kinetik enerji, yüzey hızı, derinlik ortalamalı hız ve Froude Sayısı; kesit akış analizlerinde hız büyüklüğü, ikincil hız ve türbülanslı kinetik enerji; noktasal konum analizlerinde hız, Froude Sayısı, akış derinliği, toplam hidrolik yük, ve türbülans kinetik enerjisi gibi değişkenlere önem verilmiştir. Dört farklı akış oranının, 0.429, 0.500, 0.600, ve 0.667, birleşim bölgesie yakın akış dinamikleri üzerindeki etkisi incelenmiştir Literatürle tutarlı olarak, birleşme açısı azaldıkça devridaim bölgesinin genişliğinin ve uzunluğunun azaldığı saptanmıştır. Ayrıca, birleşme açısı 30°'den 90°'ye çıktığında ikincil hızların da büyüklükleri artmaktadır. Ayrıca, akış oranının sadece devridaim bölgesinin özelliklerini etkilemekle kalmayıp aynı zamanda birleşim noktasından sonraki akış dinamiklerini de etkilediği gözlemlenmiştir

Anahtar Kelimeler: Sayısal Modelleme, HEC-RAS, CFD, Nehir Birleşim Yeri, Nehir Birleşim Açısı, Duyarlılık Analizi, Flow-3D, Akış Oranı



To my family and loved ones.

ACKNOWLEDGMENTS

I would like to express my deepest gratitude and heartfelt appreciation to Assoc. Prof. Dr. Ali Ercan for his invaluable contributions and support throughout my journey. His guidance, mentorship, and expertise have been instrumental in shaping my academic and professional growth. His profound knowledge and innovative ideas have constantly pushed me to strive for higher. I am incredibly grateful for the countless hours Assoc. Prof. Dr. Ali Ercan has invested in mentoring me and providing constructive feedback. His patience, encouragement, and willingness to share his wisdom have profoundly impacted my development as a student and as an individual.

I would like to extend my most profound appreciation to the esteemed members of my thesis jury Prof. Dr. Zafer Bozkuş, Prof. Dr. A. Burcu Altan Sakarya, Assoc. Prof. Dr. Elif Oğuz, and Asst. Prof. Dr. Meriç Yılmaz for their invaluable contributions and guidance throughout my research journey. Their expertise, knowledge, and constructive feedback have been instrumental in shaping the outcome of my thesis. I want to express my utmost gratitude to Asst. Prof. Dr. Meriç Yılmaz, whom I have had the pleasure of knowing since my bachelor's degree days. Her continuous belief in my abilities and consistent support, encouragement, and mentorship have significantly impacted my academic and personal development.

I want to extend my heartfelt appreciation and gratitude to my assistant teachers, Gökalp Doğukan Demirbaş, Hasan Hüseyin Yıldırım, Ezgi Budak Çötel, Serkan Gökmener, Emre Haspolat, Erhan Tarık Karayama, Gülçin Dalkıç Melek, Kutay Yılmaz, and with special mention to Abiddin Berhan Melek. Their unwavering support, dedication, and guidance have been helpful in my academic journey. To Berhan Melek, I am immensely grateful for your patience, approachability, and willingness to assist me throughout my studies. Your clear explanations, discussions, and willingness to go the extra mile have significantly enhanced my understanding

of complex concepts. Your continuous support has been invaluable, and I am grateful for your impact on my academic and personal growth. Gökalp Doğukan Demirbaş, I express my most profound appreciation for your exceptional assistance and support. Your enthusiasm, extensive knowledge, and teaching dedication have profoundly impacted my learning experience. Your ability to simplify complex ideas and your willingness to provide additional guidance have been precious to my academic progress. Hasan Hüseyin Yıldırım, I am grateful for your guidance, encouragement, and dedication to helping me succeed. Your expertise and willingness to address my questions and concerns have substantially shaped my understanding of the subject matter.

I want to express my sincere appreciation to Doğa Derman Çiçek, Emre Kavak, Necla Lostar, and Hüseyin Gündoğdu for their contributions and support during my journey. To Doğa Derman Çiçek, I am grateful for your willingness to share your knowledge and provide valuable support. To Emre Kavak, Necla Lostar, and Hüseyin Gündoğdu, thank you for the countless laughs, shared moments of joy, and the bonds we have forged over delicious breakfasts and meals.

I want to express my heartfelt gratitude and appreciation to my beloved family members, Aysel, Duran, Ecem Yönter, and my grandparents, Suna and Mehmet Yönter. Their unwavering love, support, and encouragement have been the foundation of my journey and the source of my strength. To Aysel, my dear mother, I am profoundly grateful for your unconditional love, sacrifice, and constant belief in me. Your unwavering support, guidance, and nurturing presence have been precious in shaping the person I am today. Your boundless strength and resilience have inspired me to overcome challenges and pursue my dreams with determination. To Duran, my dear father, I greatly appreciate your endless support, wisdom, and guidance. Your unwavering belief in my abilities and constant encouragement has instilled a strong sense of self-confidence and determination. Your dedication to my well-being and success has been a pillar of strength throughout my journey. To Ecem, my dear sister, I am grateful for your love, friendship, and unwavering

support. Your presence has brought joy, laughter, and deep companionship. Your belief in my abilities and constant encouragement has motivated me to push boundaries and strive for excellence.

I want to dedicate a special acknowledgment to my closest friends, Berkay Barış Çalışkan, İ. Berna Güvenç Erdoğan, Gökçe A. Yahşi, S. Pınar Zafer, Ebru B. Tezcan Selvitopu, Çağatay Erbaş, Tuğçe Erdem Yıldırım, Pınar Akçay, Bengü Erkekli, Işıl Hande Süer, Büşra Yıldırım, Elçin Hayvalı Parlak, Osman Selvitopu, Okan Sezer, and Dilşad Nilsu Güzeler whose unwavering patience and steadfast support have been an incredible source of strength and comfort throughout my journey. Your friendship has been a beacon of light, illuminating even the darkest of times, and for that, I am eternally grateful. To Barış, thank you for always being there for me, no matter the circumstances. Your solid patience in listening to my worries, fears, and joys has been a blessing. Your empathetic ear and understanding of nature gave me solace and reassurance. To Berna, Gökçe, Ebru, Işıl, Tuğçe, Bengü, Pınar, and Büşra, your friendship has been a constant source of inspiration and motivation. Your patient guidance and belief in me have helped me overcome countless challenges. Your ability to lend an ear and offer gentle advice has been invaluable. To S. Pınar, your understanding has been a true testament to the depth of our friendship. Your ability to provide a safe space to express myself without judgment has been an incredible gift. Thank you for your enduring support. Your belief in me, even when I doubted myself, has strengthened me. Thank you for being by my side through thick and thin, lending a shoulder to lean on, and offering support when I needed it the most.

TABLE OF CONTENTS

ABSTRACT.....	v
ÖZ	vii
ACKNOWLEDGMENTS	x
TABLE OF CONTENTS.....	xiii
LIST OF TABLES	xv
LIST OF FIGURES	xvi
LIST OF ABBREVIATIONS	xix
CHAPTERS	
1 INTRODUCTION	1
1.1 General Overview	1
1.2 Scope and Objectives of Study	2
1.3 Dissertation Specifications.....	3
2 LITERATURE REVIEW	5
2.1 Literature Concerning a 1D Hydraulic Model, HEC-RAS	5
2.2 Literature Concerning the River Confluences	13
3 METHODOLOGY	21
3.1 Approaches in the Analysis of Fluid Flow Problems	21
3.2 Numerical Methods Used	22
3.2.1 One-Dimensional Unsteady Flow Computations.....	23
3.2.2 Computational Fluid Dynamics (CFD) Computations.....	27
4 ONE-DIMENSIONAL FLOW MODELING.....	33
4.1 Sensitivity Analysis	33

4.1.1	Test Case of the Beaver Creek.....	34
4.1.2	Test Case of Mississippi-Ohio Junction	36
4.2	Case Study of 9 km Cache Creek River Reach	53
4.2.1	Description of the Study Domain	53
4.2.2	Model Setup.....	54
4.2.3	HEC-RAS Model.....	57
4.3	Concluding Remarks	61
5	THREE-DIMENSIONAL CFD MODELING OF CONFLUENCES	63
5.1	Model Setup.....	64
5.2	Mesh Independence	68
5.3	Turbulence Models.....	70
5.4	Confluence Angles	78
5.4.1	Plan-View Analyses.....	78
5.4.2	Cross-Section-View Analyses	83
5.4.3	Point Location (Probe) Analyses	88
5.5	Discharge Ratios.....	104
5.6	Concluding Remarks	117
6	SUMMARY AND CONCLUSIONS.....	121
	REFERENCES	123

LIST OF TABLES

TABLES

Table 4.1. Available observation stations of the original test model	39
Table 4.2. Different correlation results at station #28.57.....	40
Table 4.3. Selected stations to analyze the flow behavior	42
Table 4.4. Comparison for all observation stations	47
Table 4.5. Flow hydrograph multiplying factors	49
Table 4.6. Manning's n values of channel bottom	58
Table 4.7. Manning's n values for overbanks	59
Table 4.8. R^2 and NSE values for flow conditions.	60
Table 4.9. Similarity results depending on the cross-section number.	60
Table 5.1. The number of meshes generated for each mesh size.....	69
Table 5.2. Velocities of the selected points.	90
Table 5.3. Froude Numbers of the selected points.....	94
Table 5.4 Flow depths of the selected points.....	97
Table 5.5. Total hydraulic heads of the selected points.....	99
Table 5.6. TKEs of the selected points	103
Table 5.7. Discharge ratios investigated in this study.....	105

LIST OF FIGURES

FIGURES

Figure 2.1. Basic schematic of channel confluence	13
Figure 3.1. Representative channel section with a control volume	23
Figure 4.1. The comparison of simulation results and the observed values	34
Figure 4.2. The correlation, R^2 , results	35
Figure 4.3. The geometry of the Mississippi-Ohio junction river reaches, ...	37
Figure 4.4. The change of water surface elevations for station #28.57	39
Figure 4.5. Representative schematic of the junction portion	41
Figure 4.6. Water surface elevation as a function of time	42
Figure 4.7. Velocity variations as a function of time	43
Figure 4.8. Froude number variations as a function of time	43
Figure 4.9. Water surface elevations as a function of distance	44
Figure 4.10. Velocities as a function of distance through the entire reach	45
Figure 4.11. Froude number changes as a function of distance	45
Figure 4.12. Water surface elevations	49
Figure 4.13. Velocity variations	50
Figure 4.14. Froude Number variations	50
Figure 4.15. Water surface elevations for junction stations	51
Figure 4.16. Velocities for junction stations	52
Figure 4.17. Froude numbers for junction stations.....	52
Figure 4.18. Case study area.....	53
Figure 4.19. Tiled files covering the entire region	55
Figure 4.20. Tiled DEM files covering the study region.....	56
Figure 4.21. DEMs merged together for case study.....	56
Figure 4.22. Clipped DEM file representing the entire study region.	57
Figure 4.23. Observed and simulated discharges	59
Figure 4.24. Separated hydrograph for high and low flows.	60
Figure 5.1. 90° confluence channel model.....	65

Figure 5.2. 60° confluence channel model	66
Figure 5.3. 60° confluence channel model	67
Figure 5.4. Generated 6,8, and 10 millimeters meshes	68
Figure 5.5. Mesh independence investigation for velocity	69
Figure 5.6. Generated 10 mm meshes focusing on the confluence portion	70
Figure 5.7. TKEs in plan view for 90° river confluence	72
Figure 5.8 Streamlines in plan view for 90° river confluence	73
Figure 5.9 Surface velocities for 90° river confluence	74
Figure 5.10. Depth averaged velocities for 90° river confluence	75
Figure 5.11. Froude Numbers in plan view near a 90° river confluence	76
Figure 5.12. Hydraulic energy in plan view for 90° river confluence	77
Figure 5.13. The turbulent kinetic energy in plan view <i>standard k – ε</i> model ...	79
Figure 5.14. Surface Velocity in plan view for <i>standard k – ε</i> model	80
Figure 5.15. Depth average velocity in plan view <i>standard k – ε</i> model	81
Figure 5.16. Froude Number in plan view for <i>standard k – ε</i> model	82
Figure 5.17. The turbulent kinetic energies in the cross-sectional view	85
Figure 5.18. Velocities in cross-sectional view	86
Figure 5.19. Secondary velocities in cross-sectional view	87
Figure 5.20. Representative confluent channel with probe locations	88
Figure 5.21. Velocities at points near the inner wall and in the middle	89
Figure 5.22. Froude Numbers near the inner wall and in the middle	93
Figure 5.23. Flow depths near the inner wall and in the middle	96
Figure 5.24. Total Hydraulic Heads near the inner wall and in the middle	98
Figure 5.25. Turbulent Kinetic Energies near the inner wall and in the middle ...	100
Figure 5.26. Velocity profiles for different discharge ratios.	106
Figure 5.27. Velocity values of the selected probe points	107
Figure 5.28. Froude Number profiles for different discharge ratios	109
Figure 5.29. Froude Number values of the selected probe points	110
Figure 5.30. Turbulent Kinetic Energy profiles for different discharge ratios	111

Figure 5.31. TKE values of the selected probe points.....	112
Figure 5.32. Flow depth profiles for different discharge ratios.....	113
Figure 5.33. Flow depth values of the selected probe points	114
Figure 5.34. Total hydraulic head profiles for different discharge ratios.....	115
Figure 5.35. Total hydraulic head values of the selected probe points	116



LIST OF ABBREVIATIONS

ABBREVIATIONS

1D	One-dimensional
2D	Two-dimensional
3D	Three-dimensional
3DEP	3D Elevation Program
BMA	Bayesian Model Averaging
CFD	Computational Fluid Dynamics
DEM	Digital Elevation Model
EBA4SUB	Event-Based Approach for Small and Ungauged Basins
GeoTIFF	Geographically referenced file in Tagged Image File Format
GIS	Geographic Information System
GLOF	Glacier Lake Outburst Flood
HEC-HMS	U.S. Army Corps of Engineers Hydrologic Modeling System
HEC-LifeSim	Hydrologic Engineering Center Life Simulation
HEC-RAS	U.S. Army Corps of Engineers River Analysis System
LEM	Limit Equilibrium Method
NSE	Nash-Sutcliffe model Efficiency
RNG	Renormalization Group Theory
USGS	United States Geological Survey
WASP	Water Quality Analysis System



CHAPTER 1

INTRODUCTION

1.1 General Overview

Observing extreme natural phenomena such as floods and droughts has become more common within the last few decades. Natural disasters continue to increase rapidly due to global warming and seasonal changes. This increase threatens living life to a great extent and causes significant loss of property. These significant losses, especially in heavy rain and storm events, result from irregular precipitation regimes, miscalculations in the hydraulic models, and a lack of knowledge of the system behaviors. Recently, understanding the natural systems' behavior is becoming more critical. Confluences, where two or more river channels meet, are fundamental to all river systems. They are crucial points where substantial amounts of water and sediment are combined and flow into the main channel (Best, 1987). Confluences are critical in open channel hydraulics since a confluence highly affects the downstream region in several aspects hydraulically and morphologically. The flow behavior changes dramatically near the confluence region where a lateral flow meets the leading channel/reach. In the long term, these changes may create a permanent morphological effect that results in economic consequences for an extended period. Near the confluence, the flow characteristics are affected by several factors, such as the angle of confluence where two incoming flows meet and the ratio of incoming discharges of these two channels/reaches. Efforts made in the last 80 years to explore and solve the challenge of flow dynamics at confluences have encompassed various studies, one of them being the research carried out by Taylor in 1944 (Best, 1987). Since it is not possible and practical to model and simulate all the potential scenarios concerning the confluence angle and the discharge ratio focusing on the confluence

location, understanding the flow dynamics in confluent channels/reaches is essential, especially near the confluence region. More specifically, the effect introduced by the confluence to the downstream part of the leading channel/reach should be investigated. A broad interpretation is necessary for the above reasons.

1.2 Scope and Objectives of Study

In the first part of this study, HEC-RAS is utilized to understand one-dimensional river flow with and without river junctions. In the second part, a Computational Fluid Dynamics (CFD) model, FLOW-3D, is utilized to investigate three-dimensional confluent channels where the focus is to examine the flow dynamics near the confluence region.

For investigating the effect of modeling parameters, solution methods, and algorithms used in HEC-RAS, sensitivity analyses are performed with test cases selected from the HEC-RAS documentation. Scenarios are considered with different combinations for one-dimensional (1D) flow modeling, presenting the results and conclusions. One of these test cases is performed to understand the parameters that affect the flow modeling, while the other one is to investigate the flow behavior near the confluence portion of two river reaches. After that, a hydraulic model is created for a specific region to examine the investigated parameters in a real case. This real-case model is performed for Cache Creek in California, USA. The reason for selecting this region is that the accessibility of the real-world data is easy, the monitoring intervals are small, and there is a relatively large catalog of measurement data from October 2015 to today. Another reason is the available bathymetric survey for that specified region since obtaining bathymetric data in river scale is not always easy with only satellite images (digital elevation models, DEMs). The model results are compared with the observed flow data obtained from existing monitoring stations to calibrate model parameters while the region's characteristics are investigated. This study focuses on investigating the effect of model parameters and determining the Cache Creek characteristics for the people who will study this region in the future.

After conducting a 1D analysis and investigating a large-scale system behavior, another study focuses only on the confluence portion of two open channels in a three-dimensional (3D) small-scale model created by CFD software, Flow-3D. This model helps to comprehend the flow behavior by focusing on the confluence portion of these channels. Flow characteristics such as Froude Number (Fr), velocity, water depth, and turbulent kinetic energy (TKE) just downstream and far away from this confluence are examined, and the findings are discussed. First, various grid sizes are tested so that the grid size does not affect the numerical solution of the flow behavior near the junction. Then, tests are performed using different turbulence models, selecting the most suitable one that reasonably presents the flow characteristics and uses minor computational power. After determining the appropriate turbulence model, for the subsequent analyses, the focus is on the effect of confluence angle by creating three different models, which are 90°, 60°, and 60° confluences. Flow behavior is examined, and the results are discussed. Then, the effect of the discharge ratios, which are 0.429, 0.500, 0.600, and 0.667, is investigated. This study improves the knowledge of flow behavior near the junction by conducting a small-scale and high-detailed analysis.

1.3 Dissertation Specifications

This thesis is organized into six chapters for a better understanding and more straightforward evaluation. The first chapter introduces the study while identifying its scope and objectives. A review of the previous studies related to surface water flow simulations and confluent channels is presented in the second chapter. The theoretical background and governing equations of 1D and 3D flow processes are explained in the third chapter. 1D hydraulic modeling of river flow is described in the fourth chapter, with sensitivity analyses and a scenario of a selected case study region, an actual river reach, with their results and concluding remarks in this chapter. Later, in the fifth chapter, a 3D CFD analysis that is performed with hypothetical scenarios for detailed behavior analysis near the open channel

confluence is described. The simulation results and concluding remarks presented in this chapter portray the flow dynamics of confluent channels with varying angles and discharge ratios. The final chapter, chapter six, presents the summarized conclusions of this study. The future works needed to improve this study are also described in this chapter.



CHAPTER 2

LITERATURE REVIEW

2.1 Literature Concerning a 1D Hydraulic Model, HEC-RAS

A growing body of literature recognizes the significance of the design of hydraulic modeling, whose history dates back to the 1950s. The dramatic increase in the frequency of flood events worldwide brings inevitable consequences, requiring precautions. With the help of useful hydraulic model tools such as HEC-RAS, the probable flood potential resulting in catastrophic scenarios can be forecasted (Yamani et al., 2016). In estimating flood extend areas, and affected structures, it is easier to utilize the stochastic hydraulic models in 1D and 2D with high precision due to the increased computational power (Zellou and Rahali, 2016). Kumar et al. (2017) suggest that for various aspects of hydrologic design, including investigations regarding the protection of the residents, the safety of hydraulic structures, changes in the stream bed, flash flood analyses, flood extend area calculations, and the loss due to floods, HEC-RAS, can be applied. To date, several studies have utilized HEC-RAS for flood exposure (Remo et al. (2011); Butt et al. (2012); Gilany and Iqbal (2020); Zhang et al. (2021); Yılmaz et al. (2022)); morphological changes due to sedimentation (Saleh et al. (2013); Mondal et al. (2016); Joshi et al. (2019); Shabani et al. (2021)); water quality management (Fan et al. (2009); Drake et al. (2010); Zhang and Johnson (2014); Shabani et al. (2021)); and coupling HEC-RAS with other tools for various purposes (Zhang and Johnson (2014); Vojtek et al. (2019); El Bilali et al. (2021)).

The probable flood effects on communities and structures can be modeled and analyzed with HEC-RAS because of its ability to simulate complex hydraulic

conditions. A great deal of previous research into flood risk analyses was focused on obtaining the expansion of the flood and the flood stage despite the uncertainties introduced by the hydraulic model, reliability of the input data of flow, and boundary conditions of the channel (Vojtek et al., 2019; Liu & Merwade, 2018). Rising temperatures and severe precipitations directly cause an increased risk of mud or debris flows (Yılmaz et al., 2022). Publications concentrating on floodplain mapping more frequently adopt an approach using HEC-RAS coupled with GIS tools because of its ability to simulate complex flow conditions accounting for the effects of hydraulic structures. In the study of Remo et al. (2011), various flood discharges were tested, and flood mitigation scenarios were made for different configurations of levee structures, establishing the balance between levees' potential to lower flood risk and their value in flood protection. The unsteady-flow engine of HEC-RAS was adopted to investigate the potential flood exposure locations through the Middle Mississippi River and determine the cost by the cause of this exposure.

In the study of Butt et al. (2012), the objective was to evaluate the environmental benefits and drawbacks resulting from the appearance of the artificial lake after the massive landslide occurred into the Hunza River since the landslide led to a significant risk for the downstream region. Their study used the HEC-RAS model and the GIS tool, Arc-GIS. Flood inundation areas were predicted using HEC-RAS under three different peak flow scenarios. The modeled cases were the most significant peak discharge resulting in a severe dam burst scenario, a moderate discharge where the gradual development of the breach was observed, and a controlled peak discharge, smaller in magnitude and weakly affecting the inundation area (Butt et al., 2012). The inundation areas simulated by HEC-RAS and the real-case values were compared, and it was seen that with the most significant peak discharge value, non-realistic results were obtained. An abrupt dam burst scenario results could be obtained with a moderate peak discharge value in the real world. More relevant results were achieved with a controlled peak discharge value that creates a minimum risk for the farmlands and urban areas through the 145 km downstream.

Gilany & Iqbal (2020) investigated the possible corresponding risk introduced by the glacier lakes clustered upstream of a large river basin that may threaten the downstream river valley areas, facilities, and farmlands in Shyok Basin, Pakistan. Two susceptible-to-danger glacial lake outbursting due to melting were studied with HEC-RAS using various flow discharge values to determine and compare the degree of glacier lake outburst flood (GLOF) scenarios. The findings may serve as a foundation for creating a management and early alerting mechanism to lessen the effect of future potential GLOFs (Gilany & Iqbal, 2020). Similarly, Zhang et al. (2021) investigated the potential impacts of future outburst floods introduced by the seven unstable glacier lakes in the Central Himalayas to Nepal and China. The HEC-RAS model was created for multiple drainage quantity scenarios to evaluate the danger and analyze the risk and extension for that region with the hazard maps produced by HEC-RAS. As a result, they found that these outbursts will affect the residential areas in Nepal and China, even if the lakes are drained partially or fully. Also, the evacuation time for the residents in China would be 30 minutes under the correct operation of the early warning system (Zhang et al., 2021).

Most river flood estimations are conducted initially with a hydrological model to obtain peak values from the flood routing analysis. Then, with these peak values, the water stage forecasts are done by a hydraulic model, HEC-RAS, in this two-stage procedure (Hicks & Peacock, 2005). In the recent study of Yılmaz et al. (2022), the aim was to assess the hazards of debris flows in a glacier area in Barsem, Tajikistan, by using the field observation values, topographical data, and a numerical model, which is HEC-RAS, to simulate the potential threats. They conducted a sensitivity analysis for open-source DEMs to see the effect of the resolution in the model. Their results showed that open-source, respectively, low-resolution DEMs can obtain the accurate prediction of an inundation area of a debris flow. Still, in the assessment of hazards, these DEMs fall short. Also, they concluded that protective implementations such as building check dams could reduce the hazard effect. In the generation of floodplain maps, Ghanbarpour et al. (2014) performed a study on the Neka River Basin, Iran, aiming to define the willingness of the residents to pay for

insurance to flood regarding their income and the distance their houses reside from the river. Later they conclude for sustainable flood risk management, various potential mitigating scenarios should be considered.

Concerns have arisen due to the increased frequency as well as the severity of floods, which can release and distribute sediment and contaminated soil in rivers and streams (Shabani et al., 2021). Rivers undergo significant erosion and deposition in the river banks and beds due to natural or human-caused dynamics (Joshi et al., 2019). In the long term, these erosions and depositions may dramatically change the river morphology, leading to problems related to siltation. Since all rivers carry eroded sediments, the analysis regarding this phenomenon has vital importance that many researchers investigate. HEC-RAS has a feature specialized for sediment transport, which is helpful. In hydraulic modeling, an unambiguous relationship exists between having compatible results and using high computational power. Keeping the modeling data low is challenging while expecting accurate results, especially for defining the morphology of the river bed, which exhibits differences in cross-sections and slopes as a response to environmental factors and human influences (Saleh et al., 2013). The environmental factors can be variations in the river's discharge values and the loads of sediments transported.

The sensitivity analysis was performed by Saleh et al. (2013) in HEC-RAS to better understand the effect of morphological changes in the river bed by using various scenarios in the Seine River, France. Their study used different bed slope combinations and high-resolution cross-section shapes to represent the channel geometry. They calibrated their model results with the observed gauge data so that, in the end, their model creates accurate results in discharge hydrographs while employing either reduced or simplified cross-sections. However, this model must be revised with simplified geometries in the water surface prediction. Saleh et al. (2013) concluded that through the whole river reach, there is a need for a correct representation of the channel bed slopes and geometries in predicting water surface elevations and depths. They saw that the water levels of the model are affected by

the longitudinal representation of river bed level profiles the most when compared to the shapes of cross-sections.

Mondal et al. (2016) used HEC-RAS 1D for the Ichamati River basin in their sedimentation and deposition analysis. If the quality of data from remote sensing is promising, they concluded that proxy approaches could also be used. They suggested that in-situ measurements related to flow can be an excellent source to validate the model performance. In the study of Joshi et al. (2019), the alterations in the morphology of the Maumee River, Ohio, due to the sediment flow in the river bed, were analyzed with a quasi-unsteady flow engine of HEC-RAS 1D. They used a 10-meter resolution DEM in the region's representation, modifying it in ArcGIS. In determining the river characteristics, they calibrated their model's Manning's n value with a previous study of Mueller (2008) by virtue of various statistical indices such as coefficient of correlation, R^2 , and Nash-Sutcliffe efficiency, NSE . They analyzed an 8534-meter reach with 108 cross-sections since the objective was to determine characteristics of transported sediment in the river bed morphology even if the reach has a smooth transition (Joshi et al., 2019). They indicated that the erosion and deposition in the river reach have maximums of 1.5 and 0.5 meters, respectively, having a non-uniform distribution.

Shabani et al. (2021) studied sediment and contamination transport for the Woodbridge watershed, USA, with the code they developed linking HEC-RAS 1D with the Water Quality Analysis Simulation Program (WASP). In the HEC-RAS model, they preferred 2D diffusion wave equations since it satisfies the model stability (Blanch, 2017) while approximating the solution in lesser duration. For the scour analysis and deposition investigation, they coupled HEC-RAS 1D and WASP due to the incapability of HEC-RAS to simulate 2D sediment or contaminant transport in v5.0.7 (Shabani et al., 2021). They also used HEC-HMS in the simulations regarding the flood discharge relationship. Since hydraulic models determine the accuracy of predictions concerning flood inundation and flood stage, using only one model may give rise to misgivings (Liu and Merwadei 2018). In the

study of Zellou and Rahali (2016), a real-case scenario was simulated with a 2D LEM, which can model sediment transport and flood since the study area has a complex topography. To benchmark their model performance, they used HEC-RAS 1D, where the results were compatible with the 2D model in flood modeling.

Gül et al. (2009) proposed a comprehensive approach for extreme case scenarios in the Bostanli River basin in their study. For that, they used a hydrological model, HEC-HMS, in generating awaited flow and a hydraulic model, HEC-RAS, in the progression of the flood and GIS for spatial integration. Mandal & Chakrabart (2016) studied a flash flood analysis for Teesta Watershed, India, where the occurrence of this devastating event is high, using HEC-RAS together with HEC-HMS. They validated the model with actual data from a previous flood event. Flood inundation analysis was done by Papaioannou et al. (2017) for the Xerias stream, Greece, by HEC-RAS 1D linked to Monte-Carlo simulations to ascertain the effect of the roughness parameter and extent of the flood. They used n values from the empirical equations and site studies to create probabilistic n values. These values were used to estimate potential inundation areas to mitigate the effects of flood risks (Papaioannou et al., 2017). Since hydraulic models determine the accuracy of predictions concerning flood inundation and flood stage, using only one model gives rise to misgivings (Liu & Merwade, 2018). In their study, Liu and Merwade (2018) used different models to improve the strength of the deterministic estimations in flood stage with Bayesian Model Averaging (BMA) for the Black River, Missouri. They validated their models with actual event data observed in the past. Vojtek et al. (2019) coupled HEC-RAS with a hydrologic model EBA4SUB to investigate their sensitivity in the rainfall distribution, properties related to the soil, land use, and roughness values with various combinations of these parameters in the flood area and volume calculations by concluding that the effect of the roughness parameter on the sensitivity was found to be the highest.

In the recent study conducted by El Bilali et al. (2021), HEC-RAS was coupled with the HEC-LifeSim model, a software used for the probable loss of lives for flood risk

evaluation and mitigation strategies. In the generation of a hydraulic model, the most challenging part can be acknowledged by the determination of the roughness of natural rivers since it shows an alteration along the river (Ardıçlıoğlu & Kuriqi, 2019; Ramesh et al., 2000; Szomorová & Halaj, 2015). In their study, Ardıçlıoğlu and Kuriqi (2019) used HEC-RAS as they were investigating the effect of the change of the roughness parameter for the Sarımsaklı River. As intermittent rivers, such as the Sarımsaklı River, have a significant unpredictability of drying up entirely during certain times of the year, there is a need for Manning's n values to be calibrated and validated with the HEC-RAS model by proposing a relationship, as stated in Ardıçlıoğlu & Kuriqi, 2019; Brahimi & Afzalimehr, 2018; Kiss et al., 2019; Leigh et al., 2015. They indicated that as n values get higher, the difference between the calculated values of Manning's n and the n values achieved from HEC-RAS increases. Fluid in motion is studied in hydraulic models where the cross-sectional data represent the channel geometry. Saleh et al. (2013) highlight that since the availability of high-resolution data on a regional scale is not always easy, using simplified cross-sections becomes a necessity for performing a hydraulic analysis.

Stream temperature plays a vital role in stream chemical and biological processes, making it a significant parameter for assessing habitat conditions, while groundwater discharge significantly influences the thermal regime of surface water systems, particularly in streams where it is present (Drake et al., 2010). The water quality assessment of a tidal river in Taiwan by combining the Qual2K and the HEC-RAS models was performed by Fan et al. (2009). Their study used HEC-RAS for two essential purposes: estimating the hydraulic constants required for calculating the re-aeration constant and determining the variations in the water level profile to comprise concentration changes caused by tidal influences (Fan et al., 2009). The findings implied that when considering the tidal effect, the HEC-RAS-assisted Qual2K simulations yield water quality indices that generally align with the monitoring river data. Fan et al. (2009) stated that this research presented the pioneering utilization of HEC-RAS and Qual2K models in the simulations of tidal river water quality; moreover, the combination of these models provided an alternative approach for

simulating the water quality of a tidal river when the dynamic monitoring data availability were inadequate to evaluate the river's tidal effect.

In the study of Drake et al. (2010), HEC-RAS 4.0 model was used to simulate a small creek in southern Ontario, focusing on the model's temperature modeling capabilities where it was shown that the current version of HEC-RAS 4.0 falls short in the simulation of shading or heat exchange processes based on watershed characteristics. Drake et al. (2010) indicated that alternative models such as HSPF or CE-QUAL-W2 should be considered instead of relying solely on HEC-RAS 4.0 for projects requiring a more comprehensive perception of complicated processes and exchanges. Drake et al. (2010) also noted that despite its limitations in dealing with 1D problems and conditions below 0°C, HEC-RAS 4.0 offers valuable visual tools for river and watershed managers, as they provide quickly available results. In Zhang and Johnson's study (2014), a detailed account of the application and evaluation was performed using a new nutrient water quality module called Nutrient Simulation Module I (HEC-RAS-NSM I) model for the river in Lower Minnesota. They stated that this module was appropriate for emerging studies/projects with inadequate available data since it requires fewer input data and computational effort, where HEC-RAS is widely used in hydraulic studies, many river/reach systems, including large and small rivers and streams in the US, have already been displayed with HEC-RAS (Zhang & Johnson, 2014).

In the study of Shabani et al. (2021), a Python-based External Coupler was developed to connect the HEC-RAS 2D hydrodynamic model with the Water Quality Analysis Simulation Program (WASP), enabling accurate sediment and contaminant transport simulation during flood events. By linking HEC-RAS v5.0.7 with WASP, Shabani et al. (2021) simulated the transport of sediment and associated toxicants successfully where they indicated that the integrated model provided valuable insights into sediment and contaminant transport during flood events, contributing to improved awareness of the risks related to polluted soil and sediment in water systems.

2.2 Literature Concerning the River Confluences

Confluences in open channels have considerable importance for the river systems as the mixing of two streams affects the downstream characteristics of the river (Best & Reid, 1984). The flow dynamics near the confluent part of two streams can be understood by recognizing generated regions: stagnation, deflection, maximum velocity, separation, flow recovery, and shear layer (Best, 1987).

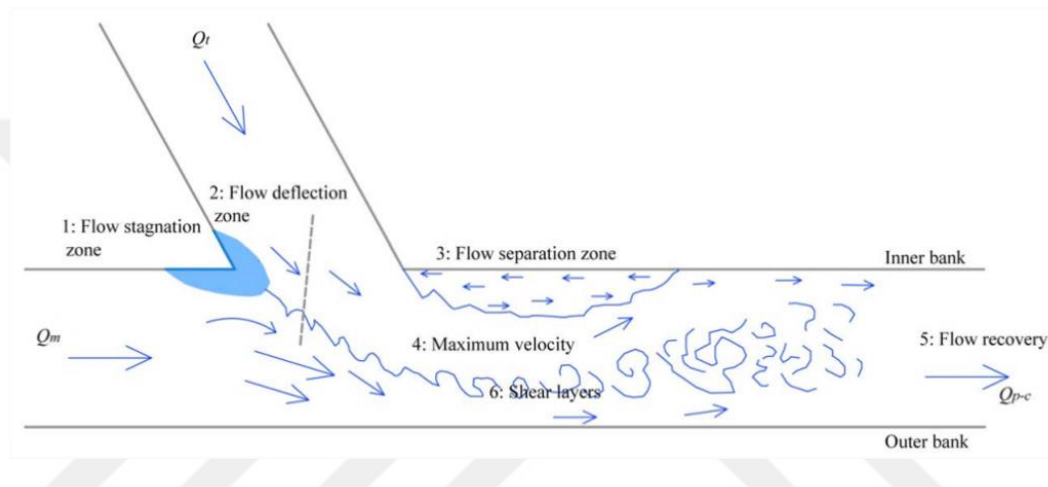


Figure 2.1. Basic schematic of channel confluence where all the zones introduced by the confluence are shown.
(Adapted from Leite Ribeiro et al., 2012)

Best (1987) indicated that these regions are controlled by the confluence angle and discharge ratios between these two streams. These six regions are shown in Figure 2.1—describing a basic schematic where confluence dynamics are demonstrated. In the figure Q_m , Q_t , and Q_{p-c} correspond to the main, tributary, and post-confluence channel discharges, respectively. As seen in the figure, incoming flow discharges of Q_m and Q_t intersect, and the first region is created due to this intersection, the flow stagnation zone. These flows deflect after the intersection, creating deflection zones both in the tributary and the main channels. This deflection triggers a flow separation area where the recirculation flow accumulates just downstream of the confluence, forming vortices. The most prominent region within the channel confluence is where separated flow forms beneath there. This abrupt change is due

to the flow separation near the inner wall at the downstream junction corner, which grounds the flow in a tributary to lose attachment to the channel wall when it enters the confluence. According to Best (1987), this lower-pressure region is dynamically active since all the turbulent characteristics can be observed due to the flow separation. The increased velocity and turbulence result in scouring, segregating the bedload from each channel. This segregation results from the mutual deflection of flows and the flow separation zone, which draws material from the tributary channel (Best, 1987).

As the flow moves downstream, it gradually recovers from the confluence dynamics, leading to a symmetrical bed morphology without cross-channel elevation asymmetry. Also, the intersection of these two flows generates the shear layer, where the bed shear stress is most dominant in this region. The two flows start to mix across the shear layer between the combined flows, which reduces turbulence levels. Maximum velocity region occurs between the shear and flows separation layers. After some distance from the confluence, the flow regains its characteristics, and a mostly more stable condition is achieved; this region is called the flow recovery region. The asymmetrical velocity distribution across the channel, which occurs due to flow separation and acceleration at the confluence, gradually disappears, and the flow pattern becomes more symmetrical, flowing parallel to the channel walls (Best, 1987). Many researchers have confirmed the existence of this distinctive separated flow region at the downstream junction corner and have studied its characteristics in detail. The area where flow separation occurs is crucial regarding sediment erosion, transport, and bed morphology within the junction. The flow dynamics in the separation zone can influence the patterns of sediment deposition and the formation of the riverbed, making it a significant factor to consider in the study of river morphology. As the flow moves downstream from the separation zone region, it gradually recovers from the effects of confluence dynamics.

According to Rice et al. (2008), published papers concerning the confluent channels in geomorphology and sedimentology, ecology, hydraulics, and hydrology

increased, especially after the 1990s. The initial endeavor to construct a comprehensive model of confluence hydraulics can be traced back to Mosley (1976), which was further refined by Best (1987). Best and Reid (1984) investigated the separation zone characteristics due to the lateral flow of the tributary channel. Four different angles, 15° , 45° , 70° , and 90° , were tested with rectangular identical cross-sectional straight channels where the width was 0.15 meters. They observed a recirculation zone forming as the flow separated from the sidewall, initiating a low-pressure region. Subcritical flows with Froude numbers between 0.1 and 0.3 were tested through the experiment with different discharge values, and a relation between the discharge ratio and separation zone was obtained. They indicated that as the angle between the primary and tributary increases from 15° to 90° , the velocity near the bottom of the channel and the separation zone increases under the same channel configurations. With an increasing discharge ratio, the separation zone length and width increase while keeping its shape for all cases. They concluded that the rise in tributary has a lesser effect as the angle becomes smaller, so the channel designs with lower angle confluences may reduce the damage introduced by the flooding (Best & Reid, 1984).

Best (1987) conducted another study providing an in-depth analysis of flow dynamics at confluent channels, considering its implications for sediment transport and bed morphology. It aims to shed light on the complex nature of flow at these junctions and its impact on sediment deposition and bed formation patterns. This analysis was performed in an experimental study for five angles, 15° , 45° , 79° , 90° , and 105° , and 0.15-meter-wide channels. The flow depths at the inlets and the outlet were constant, 0.20 meters, in the experimental setup. Three discharge ratios were studied that are 0.5, 1.0, and 1.5, for each junction angle configuration. The findings of Best (1987) can be listed as follows:

- 1- At the flow confluence, these primary and tributary channel flows deflect directly proportional to the junction angle and create a deflection zone that

increases with the increasing confluence angle reaching a point where a reverse flow is produced when the angle between them is 105° (Best, 1987).

- 2- The rise in the discharge ratio also creates a rise in the deflection magnitude and separation zone.
- 3- When the discharge ratio is larger, the stagnation zone in the lateral channel decreases. In the case of a 15° junction, however, the deflection of each stream is considerably low (Best, 1987). After this zone, the shear layer is generated between the confluent flows, where bed shear stress and velocity increase can be observed.
- 4- The area of separation, which could potentially occupy up to half of the post-confluence channel width, expands in length and width as the junction angle and channel discharge ratio increase.

In the study of Hager (1989), the approaching flow states in the main and tributary channels are subcritical, where the aim was to observe the limit for transitional flow to occur. If the discharge in the tributary channel occupies 15% or more of the total discharge, this aimed flow condition can be observed. In this study, some assumptions were made to use the momentum and energy equations. These assumptions are that the pressure should be hydrostatic, the velocity distribution should be uniform with a horizontal bed, approaching channels have identical flow depths, and their width should equal the downstream section. The frictions at the walls were neglected, and the streamlines were assumed to be parallel at the control volume boundaries. With these assumptions, the results attained from the related equations and the experimental model outcomes were compared. By using a one-dimensional approach, Hager's study (1989) aimed to predict the separation zone width. The experimental setup, 0.096-meter rectangular branches were used with 22.5° , 45° , and 90° angles. This study reached a reasonable agreement between the theoretical and experimental approaches, with no more than 8% deviations.

In their study, Gurram et al. (1997) specified that in the confluent channels, the flow dynamics are dominated by the contraction and curved regions in channels. The flow

states during the experiments were satisfied as subcritical, where the simple junction angles, 30°, 60°, and 90°, were tested with five discharge ratios (ratio between the lateral flow to the total flow), 0, 0.25, 0.50, 0.75, and 1. The lateral flow characteristics and flow patterns were examined throughout the study. The results were measured against a numerical model after that. Gurram et al. (1997) indicated that the challenges in this hydraulic phenomenon occur due to the mixing of flow followed by a secondary flow generation and the backwater formation. Since two flows intersect, due to their mixing, just downstream of the confluence, a flow separation region is created, as stated above. In this region, recirculating motion creates a secondary flow pattern. In the cases of high velocity coming from both upstream reaches, a backwater in both channels may occur after the intersection. Subcritical and transcritical states were examined to investigate the pattern of flows. Other aims were to assess a numerical approach and to propose a design approach (Gurram et al., 1997) for confluent channels. The findings can be listed as follows:

- 1- As the discharge ratio increases in 90° confluence, the effect of lateral flow is significant on the main channel's outer wall.
- 2- In the separation zone, the turbulence is high, and the surface waves occur downstream of this zone.
- 3- Lateral flow creates a sink in the primary channel after the confluence, which can be visible when the discharge ratio is 1.
- 4- In the design of confluent channels, considering discharge ratios and angles and the Froude number of the tailwater is essential. (Gurram et al., 1997).

A numerical study related to the confluent channels in a two-dimensional and three-dimensional form that was conducted by Wang and Cheng (2000) demonstrated the difference between two turbulence models, *RNG* (Re-Normalized Group) $k - \varepsilon$ and *standard* $k - \varepsilon$, while validating the results with a previous experimental study. In their study, Wang and Cheng (2000) aimed to investigate the ability of *RNG* $k - \varepsilon$ turbulence model in the representation of three-dimensional confluent channels where the focus was the length of recirculation region that occurs just downstream

of the confluence and examine the flow structure in that region. For the recirculation region, the results achieved from the *RNG k – ε* model agree well with the experimental results. The length of this region was underestimated by the *standard k – ε* model (Wang and Cheng, 2000). They also indicated that the size of the separation/recirculation region may vary vaguely in the vertical direction; as the roughness of the wall increases, the length of this region decreases. The length and width of the recirculation zone were observed to be directly proportional to the momentum flux ratio between the primary and lateral flows. Both the two-dimensional (2D) and three-dimensional (3D) *RNG k – ε* models show valuable estimations of the recirculation length than the *standard k – ε* model, and the 3D *RNG k – ε* models improve the results by around 60% (Wang and Cheng, 2000).

A numerical study regarding the confluent channels was performed by Goudarzizadeh and Musavi-Jahromi (2020), and the findings of the generated model were validated with the experimental model of Shumate (1998). They used a 90° confluence channel where the primary and lateral channels have lengths of 21.946 meters and 3.658 meters, respectively, with a horizontal bed. The boundary conditions were defined as average velocities at the main and lateral channel inlets, atmospheric pressure at the outlet, and the channel zones; wall boundary condition was used with the assumption of a smooth surface. They ignored the water surface variations, which led them to a ‘one-phase’ flow where the water surface has ‘symmetry’ boundary conditions. They satisfied the mesh independence with 252,450 nodes for cubic elements. The difference in bed shear stress introduced by the changing discharge ratio, q^* , was investigated. Three discharge ratios were investigated that are 0.25, 0.50, and 0.75.

$$q^* = \frac{\text{lateral channel discharge}}{\text{total discharge}} \quad 2.1$$

Similar results were achieved when comparing both cases, where the average prediction error ranges between 3.5 and 10%. It was seen that there is an inverse relation between q^* and the length and width of the separation, as well as the bed

shear stress' maximum value. Their model can be accepted as influential in the representation of flow patterns. In their study, Goudarzizadeh and Musavi-Jahromi (2020) used Reynolds stresses turbulent model (*RSM*) while calculating shear stress and obtaining flow patterns. High accuracy was achieved in flow field simulations. The slight difference was because of the constant flow depth assumption from the preprocessing part of the simulations. Goudarzizadeh and Musavi-Jahromi (2020) obtained fully developed velocity profiles (maximum at the center and zero at the wall). After the confluence, they investigated the shape index, the ratio related to the width and length of the separation zone, and they observed that as q^* increases, flow velocity and momentum flux in the main channel increase as well. For the lateral channel, its impact was the opposite, where the width and length of the separation zones were decreased.

CHAPTER 3

METHODOLOGY

3.1 Approaches in the Analysis of Fluid Flow Problems

Early literature on hydraulic modeling has taken its shape with mostly experimental studies since the computational power was not strong/decisive as it is now. However, from the 1960s onwards, the computational technique has become a vital component spanning a broad range of fields in design (Versteeg & Malalasekera, 2007). A large-scale analysis and discussion by Versteeg and Malalasekera (2007) demonstrate that the systems related to the flow of fluids and involving these phenomena can be analyzed with computer-based simulations with the help of computational fluid dynamics (CFD). The scope of CFD is so broad that the applications of its tools can be seen in many areas like hydraulic engineering when, especially, there is an open channel free surface flow. Ferziger et al. (2019) state that the accurate solution of a Newtonian fluid flow can be found using Navier-Stokes (N-S) Equations with computational effort. It is more likely to experience some limitations with the theoretical and experimental approaches. Analytical solutions by integrating a formula can only be possible if the flow characteristics are simple enough and the basic governing equations are used in their linear form. This limitation makes this approach not preferable for real case problems. In the case of the experimental method, a model should be constructed, which comes with scaling issues and expensive measurement devices.

On the contrary, in discretizing Navier-Stokes Equations in three-dimensional form, using the numerical approach can be more practical, as stated in Lopes (2013). The nonlinear nature and the complexity of the governing equations of fluid flow in differential forms are threatened by an iterative process in this approach. For the study of systems where the experimental method is tough to use, the numerical

approach, CFD, gives high-resolution results while simultaneously reducing the time and cost with the proper governing equations and boundary conditions correctly representing the flow field.

3.2 Numerical Methods Used

In this thesis, the numerical analysis will be performed for the hydraulic modeling of a specific region, Cache Creek. This process will use the software package HEC-RAS as the hydraulic modeling tool. HEC-RAS advanced at the Hydrologic Engineering Center (HEC) and first released in 1995, helps model water flow in open channels, such as river reaches and streams, calculating the flow velocities, water surface profiles, water depths, and other hydraulic parameters (Brunner, 1994). HEC-RAS allows analyzing steady flow in 1D, unsteady flow in 2D or combined 1D-2D, sediment transport, water quality, and temperature analyses (Brunner, 1994). In this study, with the flow hydrograph taken from the monitored data, the unsteady analysis will be performed with HEC-RAS. This section will describe the computation procedure with the governing equations in 1D flow hydrodynamics.

Flow hydrodynamics are expressed numerically in CFD with the help of the N-S Equations. It is the most commonly used transport equation since it can be applied to shallow water flows with some assumptions. As it is stated in the studies of Vreugdenhil (1994) and Liang and Borthwick (2009), in shallow flows, assuming a nearly horizontal flow can be more practical and economical since the acceleration of the particles in the horizontal direction is much higher than the flow depth (vertical direction). In the assumption of a horizontal flow, the acceleration in the vertical direction is neglected. The N-S Equation becomes depth-averaged, called the Shallow Water Equations (SWE), in one or two-dimensional forms (Toro & Garcia-Navarro, 2007). Dronkers (1964) states that these equations are quoted as Saint-Venant Equations (SVE) in the scientific society of France, even though Saint-Venant derived only the one-dimensional form of these equations. In the derivation

of SVE, some assumptions must be made, as stated by Chow et al. (2013). The assumptions necessary for the simplification are as follows:

- This equation is valid for an incompressible fluid whose density is constant with a hydrostatic pressure distribution.
- The flow is one-dimensional, where the flow depth and velocity variations are only in the longitudinal channel direction, while the acceleration in the vertical direction is neglected.
- The channel has a small slope with a fixed channel bed where the erosion and deposition in the channel bed are neglected.
- Manning's Equation can describe the flow resistance.

The complete derivation of SVE from the incompressible form of N-S Equations can be found in the hydraulic textbooks of Akan (2006), Chow (1959), and Vreugdenhil (1994), and also from the studies of Strelkoff (1970). For a detailed explanation of the complete derivations of this equation, one may want to review these textbooks.

3.2.1 One-Dimensional Unsteady Flow Computations

In a stream, water flow is governed by the conservation of mass and momentum equations (Brunner, 1994). In Figure 3.1, a representation of a channel section is shown where $Q(x,t)$ is the inflow discharge and $h(x,t)$ is the water depth. The control volume is defined as Δx where the inflow and outflow to that control volume are demonstrated.

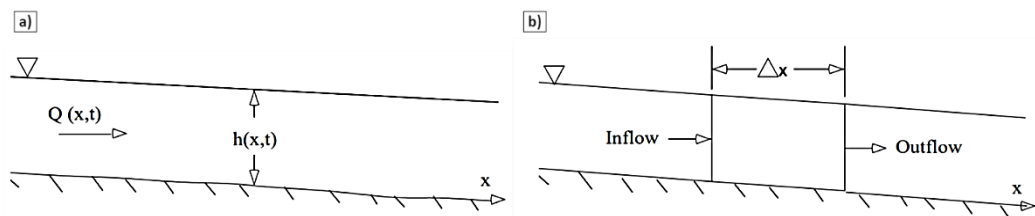


Figure 3.1. Representative channel section with a control volume where the channel bottom, water surface elevation, and flow depth are shown. (Adapted from Brunner, 1994).

Both the continuity and momentum equations are written for that representative cross-section. The net inflow rate to a control volume should be equal to the rate of change in the storage of that control volume in the continuity equation. In Equation 3.1, the fluid density is ρ , and the total area of the control volume is A_T , and the lateral inflow to that control volume is Q_l (Brunner, 1994).

$$\underbrace{\rho \frac{\partial A_T}{\partial t} \Delta x}_{\text{the net rate of change in storage}} = \rho \left[\underbrace{\left(Q - \frac{\partial Q}{\partial x} \frac{\Delta x}{2} \right)}_{\text{the rate of inflow}} - \underbrace{\left(Q + \frac{\partial Q}{\partial x} \frac{\Delta x}{2} \right)}_{\text{the rate of outflow}} \right] + \underbrace{Q_l}_{\text{lateral inflow}} \quad 3.1$$

After the simplifications, dividing both sides by $\rho \Delta x$. The continuity equation's final form for one-dimensional unsteady flow is obtained, as shown in Equation 3.2. where q_l represents the lateral inflow per unit length (Brunner, 1994).

$$\frac{\partial A_T}{\partial t} + \frac{\partial Q}{\partial x} - q_l = 0 \quad 3.2$$

The rate of change of momentum of a small finite volume of fluid within an extensive fluid continuum is expressed by Equation 3.3, which is similar to Newton's second law. The total derivative of the momentum change is equal to the summation of the forces acting on a control volume of fluid. This equation is valid for any volume in the continuum. A is the cross-sectional area, g is the gravitational acceleration, z_0 is the inverted elevation and S_f is the friction slope in the equation (Brunner, 1994).

$$\underbrace{\rho \Delta x \frac{\partial Q}{\partial t}}_{\text{momentum of fluid in c.v.}} = - \underbrace{\rho \frac{\partial QV}{\partial x} \Delta x}_{\text{net rate of momentum entering c.v.}} - \underbrace{\rho g A \frac{\partial h}{\partial x} \Delta x}_{\text{net pressure force}} - \underbrace{\rho g A \frac{\partial z_0}{\partial x} \Delta x}_{\text{gravitational force}} - \underbrace{\rho g A S_f \Delta x}_{\text{friction force}} \quad 3.3$$

Even if the flow rate does not change, the fluid parcel moves through a continuum; it may accelerate in time as well as in space. The equation demonstrates the change in the momentum in time and space for only one component, x , since only one-

dimensional unsteady flow is considered in this section. The water surface elevation z is equal to the sum of the water depth, h and invert elevation, z_0 , in Equation 3.4. With this, Equation 3.3 can be rewritten in a more compact form, dividing both sides with $\rho\Delta x$, and moving all the terms to the left side of the equation (Brunner, 1994).

$$\underbrace{\frac{\partial z}{\partial x}}_{\text{water surface slope}} = \underbrace{\frac{\partial h}{\partial x}}_{\text{change in depth in } x \text{ dir.}} + \underbrace{\frac{\partial z_0}{\partial x}}_{\text{change in invert elevation in } x \text{ dir.}} \quad 3.4$$

The final form of the momentum equation (Brunner, 1994) for 1D unsteady flow in a stream can be obtained by these adjustments, demonstrated in Equation 3.5 (Brunner, 1994).

$$\frac{\partial Q}{\partial t} + \frac{\partial QV}{\partial x} + gA \left(\frac{\partial z}{\partial x} + S_f \right) = 0 \quad 3.5$$

Burden et al. (2015) state that, in approximating the time-dependent partial differential equations, explicit or implicit approaches can be adapted to numerical solution procedures, depending on the solving problem, the desired accuracy, and the computational efficiency. An explicit solution is a numerical approach where each new value is determined solely based on a single equation at a given time step without considering the values at other points. This approach requires smaller time steps to prevent the solution from divergence. Distinctively, an implicit solution involves computing the solution by solving a set of equations simultaneously for the entire region at each time step. This method is more intensive computationally but tolerates larger time steps and provides excellent stability in the solution. In HEC-RAS, an implicit approach is used with the Newton-Raphson iteration technique addressed in Brunner (1994). This technique can be considered self-correcting since the iteration continues until convergence. For further information, one may check the textbook of Gerald and Wheatley (1984). In solving one-dimensional unsteady flow equations, the most effective and applicable scheme is accepted as a four-point

implicit scheme, also called the box scheme, denoted in the report of Barkau (1982). For further information and the numerical progression, the readers should review the report of Barkau and the Hydraulic Reference Manual of HEC-RAS. In the implicit finite difference scheme of HEC-RAS, the solution is simultaneously done for the whole reach using comparatively larger time steps than the explicit solution schemes (Brunner, 1994).

In this study, the stability of the solution will be investigated with a sensitivity analysis by various combinations of intervals regarding the time steps and distances. The factors that cause non-stability in this solution scheme will be examined for all cross-sections and model parameters to have a stable solution and reliable results. These factors can be listed as follows:

- In channel slope, if there is an abrupt change, it affects the stability of the solution.
- If the cross-section of a channel dramatically changes, it brings instability to the solution.
- For the solution stability, the complexity introduced by the hydraulic structures should be handled in the river reach, such as bridges, culverts, and weirs.
- The flood wave can also bring instability if the model is not established according to that flood wave.

For every reach in the river system, boundary conditions should be provided. They should be applied to each reach satisfying the equilibrium between the number of equations and the number of unknowns. Flow hydrograph can be used as a boundary condition if there is measured data for that specific reach as a time series both upstream and downstream. For the river reach connections, interior boundary conditions should be applied for the model to run as a continuous whole. The downstream boundary condition can be specified as normal depth, whereas Manning's equation defines the flow downstream if the condition is uniform. These

boundary conditions are applied to fulfill the flow and stage continuity. One may check Brunner (1994) for further information on boundary conditions.

3.2.2 Computational Fluid Dynamics (CFD) Computations

In CFD applications, there are two methods used to describe the fluid motion that is the Lagrangian and the Eulerian approaches, where the physical quantities of the fluid, such as velocity, density, and pressure, are described as a function of time and space (Zhang & Chen, 2007). In the Lagrangian approach, each particle of fluid flow is tracked individually, and the variations in the pathway of these particles are described along their trajectory, whereas, in the Eulerian approach, the particle phase is treated as a continuum, and the conservation equations are developed for the whole control volume in this approach (Dronkers, (1964); Mei (2001); Saidi et al. (2014); Zhang & Chen, (2007)). Flow-3D is a CFD solver where non-hydrostatic Navier Stokes Equations are solved with the Lagrangian approach. Navier-Stokes equations are CFD's most commonly used transportation equation. When the forces acting on a solid body are thought of, since all the equations learned so far examine the motion of a solid body before moving on to a fluid, starting with the basics can be a good approach. Newton's second law in differential form is given by Equation 3.6 (Pope, 2000).

$$F = ma \rightarrow F = m \frac{dv}{dt} \quad 3.6$$

The sum of the forces acting on a solid body is represented with F , the force vector, equal to the inertia of the body m times the acceleration vector a . It is more convenient to write the rate of change in velocity in differential form since, as a fluid property, velocity is a function of space and time. If the mass of the body changes with time, then Equation 3.7. can be more practical (Pope, 2000).

$$F = \frac{d(mv)}{dt} \quad 3.7$$

The sum of the forces acting on a solid body equals the rate of change of its momentum. Since this is a vector equation, it has three components in the cartesian coordinate system. These components for the x, y, and z axes are given by Equation 3.8 (Pope, 2000).

$$F_x = \frac{d(mv_x)}{dt} \quad F_y = \frac{d(mv_y)}{dt} \quad F_z = \frac{d(mv_z)}{dt} \quad 3.8$$

The N-S equation is used if the body is not solid but fluid (liquid or gas). The rate of change of momentum of a small finite volume of fluid within an extensive fluid continuum is expressed by Equation 3.9, which is similar to Newton's second law.

$$\frac{D(m\mathbf{U})}{Dt} = F \quad 3.9$$

This time, F is the sum of the external forces acting on a fluid parcel, and $(m\mathbf{U})$ is the mass of this parcel multiplied by its velocity, $\mathbf{U}$. If this equation is divided by the fluid volume, Equation 3.9 is simplified to Equation 3.10 (Pope, 2000).

$$\frac{D(\rho\mathbf{U})}{Dt} = f \quad 3.10$$

The momentum of the fluid ($\rho\mathbf{U}$) is the multiplication of ρ , the density of the fluid, with $\mathbf{U}$, the velocity. The total derivative of the momentum change is equal to the sum of the forces acting on a unit volume of fluid, f in Equation 3.10. This equation is valid for any volume in the continuum. The forces acting on a surface can be found with a numerical method where the velocity of the liquid as it moves around a body is calculated. The Navier-Stokes equation, represented by a total derivative of the momentum change by time, is given in Equation 3.11 (Pope, 2000).

$$\frac{D(\rho\mathbf{U})}{Dt} \quad 3.11$$

Even if the flow rate does not change, the fluid parcel moves through a continuum; it may accelerate in time as well as in space. The total derivative can be expanded

into a time derivative of the change in the momentum and the change in space for all three components, x , y , and z , of the cartesian coordinate system as Equation 3.12. The spatial derivatives are multiplied by the velocity in three directions (Pope, 2000).

$$\frac{D}{Dt} = \underbrace{\frac{\partial}{\partial t}}_{\text{Time}} + \underbrace{U_x \frac{\partial}{\partial x} + U_y \frac{\partial}{\partial y} + U_z \frac{\partial}{\partial z}}_{\text{Space}} \quad 3.12$$

Writing Equation 3.12 in a more compact form, Equation 3.13 is obtained where the Laplacian term replaces all partial derivatives in space for all three coordinates ∇ .

$$\frac{D}{Dt} = \frac{\partial}{\partial t} + \mathbf{U} \cdot \nabla \quad 3.13$$

The Navier-Stokes equation is written with the expanded form of Equation 3.10 in Equation 3.14 (Pope, 2000).

$$\frac{\partial(\rho \mathbf{U})}{\partial t} + \mathbf{U} \cdot \nabla (\rho \mathbf{U}) = \mathbf{f} \quad 3.14$$

After the conservation of mass and product rule used to simplify Equation 3.14, finally, Equation 3.15 can be obtained. In this equation, the sum of the external forces acting on a unit volume of fluid equals the change in the fluid volume's momentum in time and space (Pope, 2000).

$$\frac{\partial(\rho \mathbf{U})}{\partial t} + \nabla \cdot (\rho \mathbf{U} \mathbf{U}) = \mathbf{f} \quad 3.15$$

Due to pressure, viscosity, and gravity, many forces act on a small fluid volume. In Equation 3.16, the pressure gradient term, the gradient of the shear stress, and acceleration due to gravity are written on the right-hand side of the equation. The forces per unit volume are static pressure, p , shear stress, τ , and acceleration due to gravity, g . Under the effect of these forces, the acceleration in the fluid body is represented by the terms on the left-hand side (Pope, 2000).

$$\frac{\partial(\rho \mathbf{U})}{\partial t} + \nabla \cdot (\rho \mathbf{U} \mathbf{U}) = \underbrace{-\nabla p}_{\text{Pressure}} + \underbrace{\nabla \cdot \tau}_{\text{Shear stress}} + \underbrace{\rho g}_{\text{Gravity}} \quad 3.16$$

The pressure gradient has a minus sign in front of it. The reason is that the fluid will accelerate in the direction of decreasing pressure. As is seen from Equation 3.16, there is a $(\rho \mathbf{U} \mathbf{U})$ term where $\mathbf{U}^2$ term appears in the equation. This makes the equation non-linear and difficult to solve as a differential equation. Complicated geometries require numerical methods to be used. Because of that, the Navier-Stokes equation is used to obtain the velocity of the fluid body.

In Flow-3D, the numerical calculations are based on the continuity of mass and momentum for a specified control volume. The mass continuity equation utilized in the solution procedure is shown in Equation 3.17, where V_F represents the fractional volume open to the fluid flow, ρ being the fluid density, R representing a turbulent diffusion term (Flow Science, Inc., 2019). The velocity components in 3D form are (u, v, w) , representing the coordinates of (x, y, z) , respectively. A is the fractional area open to flow in three dimensions. R_{DIF} and R_{SOR} are the turbulent diffusion and density source terms, respectively (Flow Science, Inc., 2019).

$$\begin{aligned} V_F \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho u A_x) + R \frac{\partial}{\partial y}(\rho v A_y) + \frac{\partial}{\partial z}(\rho w A_z) + \xi \frac{\rho u A_x}{x} \\ = R_{DIF} + R_{SOR} \end{aligned} \quad 3.17$$

Turbulent diffusion term, R_{DIF} , is shown in Equation 3.18.

$$R_{DIF} = \frac{\partial}{\partial x} \left(v_\rho A_x \frac{\partial \rho}{\partial x} \right) + R \frac{\partial}{\partial y} \left(v_\rho A_y R \frac{\partial \rho}{\partial y} \right) + \frac{\partial}{\partial z} \left(v_\rho A_z \frac{\partial \rho}{\partial z} \right) + \xi \frac{\rho v_\rho A_x}{x} \quad 3.18$$

Compressible problems use the complete form of Equation 3.17 since they require the solution of complete density transport equations (Flow Science, Inc., 2019). However, with a constant density in incompressible cases, Equation 3.17 reduces to Equation 3.19 with incompressibility condition (Flow Science, Inc., 2019).

$$\frac{\partial}{\partial x}(uA_x) + R \frac{\partial}{\partial y}(vA_y) + \frac{\partial}{\partial z}(wA_z) + \xi \frac{uA_x}{x} = \frac{R_{SOR}}{\rho} \quad 3.19$$

The modified continuity equation is in the form of Equation 3.20.

$$\frac{V_F}{\rho c^2} \frac{\partial p}{\partial t} + \frac{\partial uA_x}{\partial x} + R \frac{\partial vA_y}{\partial y} + \frac{\partial wA_z}{\partial z} + \xi \frac{uA_x}{x} = \frac{R_{SOR}}{\rho} \quad 3.20$$

The Navier-Stokes equations, which govern the fluid velocity components in three dimensions (x, y, z) , are augmented with additional terms to account for additional factors. These factors include external forces, turbulence, heat transfer, and other physical or chemical processes within the fluid. These modified equations provide a more comprehensive representation of fluid motion, considering the intricate interactions and influences that impact the fluid's behavior. Equations 3.21, 3.22, and 3.23 present the momentum equations in three dimensions (x, y, z) , respectively. Again, (u, v, w) terms are the velocities in (x, y, z) directions.

$$\begin{aligned} \frac{\partial u}{\partial t} + \frac{1}{V_F} \left\{ uA_x \frac{\partial u}{\partial x} + vA_y R \frac{\partial u}{\partial y} + wA_z \frac{\partial u}{\partial z} \right\} - \xi \frac{A_y v^2}{xV_F} \\ = -\frac{1}{\rho} \frac{\partial p}{\partial x} + G_x + f_x - b_x - \frac{R_{SOR}}{\rho V_F} (u - u_w - \delta u_s) \end{aligned} \quad 3.21$$

$$\begin{aligned} \frac{\partial v}{\partial t} + \frac{1}{V_F} \left\{ uA_x \frac{\partial v}{\partial x} + vA_y R \frac{\partial v}{\partial y} + wA_z \frac{\partial v}{\partial z} \right\} - \xi \frac{A_y uv}{xV_F} \\ = -\frac{1}{\rho} \left(R \frac{\partial p}{\partial y} \right) + G_y + f_y - b_y - \frac{R_{SOR}}{\rho V_F} (v - v_w - \delta v_s) \end{aligned} \quad 3.22$$

$$\begin{aligned} \frac{\partial w}{\partial t} + \frac{1}{V_F} \left\{ uA_x \frac{\partial w}{\partial x} + vA_y R \frac{\partial w}{\partial y} + wA_z \frac{\partial w}{\partial z} \right\} \\ = -\frac{1}{\rho} \frac{\partial p}{\partial z} + G_z + f_z - b_z - \frac{R_{SOR}}{\rho V_F} (w - w_w - \delta w_s) \end{aligned} \quad 3.23$$

Where (G_x, G_y, G_z) are body accelerations, (f_x, f_y, f_z) are vicious accelerations, (b_x, b_y, b_z) are flow losses in porous media (Flow Science, Inc., 2019). $\mathbf{U}_w = (u_w, v_w, w_w)$ is the velocity of the source component, generally will be non-zero for a mass source (Flow Science, Inc., 2019).

The term $\mathbf{U}_s = (u_s, v_s, w_s)$ is the velocity of the fluid at the surface of the source relative to the source itself (Flow Science, Inc., 2019). dQ is mass flow rate, ρ_{QSR} is fluid source density, dA is the area of the source surface, $\mathbf{n}$ is the outward normal to the surface (Flow Science, Inc., 2019). The fluid velocity at the source surface is given in Equation 3.24.

$$\mathbf{U}_s = dQ/(\rho_{QSR} \cdot dA) \cdot \mathbf{n} \quad 3.24$$

Turbulent flows are expected in various situations, with varying geometry, driving forces, and physicochemical processes involved (Yakhot & Orszag, 1986). *Standard $k - \varepsilon$* model is the most widely used model for simulating various engineering and practical problems; however, it is commonly known that the *standard $k - \varepsilon$* model is only suitable for attached boundary layer flows (Wang & Cheng, 2000). For detailed information, one may check Pope (2000) and Wilcox (2010) for the derivations and applications of additional turbulence terms to the Navier-Stokes Equations.

The *$k - \omega$* model, which has been developed by Wilcox (2010) for over two decades, is the second most commonly utilized two-equation model. It shares similarities with the *standard $k - \varepsilon$* model described comprehensively by Wilcox (2010). One may check Wilcox (2010) and Pope (2000) again for a more detailed explanation.

The *RNG* (Re-Normalization Group) *$k - \varepsilon$* turbulence model is another widely used model in CFD simulations to predict turbulent characteristics (Yakhot & Orszag, 1986). It is based on the Reynolds-averaged Navier-Stokes (RANS) Equations. For more information regarding the *RNG $k - \varepsilon$* turbulence model, one may check Yakhot and Orszag (1986) and Shen et al. (2022).

CHAPTER 4

ONE-DIMENSIONAL FLOW MODELING

In hydraulic modeling, in the context of river simulations, the existing river system is aimed to be represented correctly since the accuracy of the results is mainly linked to the model's performance which is affected by various factors. In the delineation of the exact geometry of a river, hydraulic models do not always give high accuracy due to the simplified nature of the digital elevation data, uncertainty in Manning's coefficient, or the solution algorithms and their related parameters used in the model (Saleh et al., 2013, Warner et al., 2021). These factors should be investigated to succeed in an accurate model. One needs to undergo a calibration process for an effective and reliable hydraulic model. Zimoch and Bartkiewicz (2018) state that this involves adjusting the model parameters to reduce the discrepancy between the model's output and real-world observations. If reliable monitored data is available, the hydraulic model can be calibrated with the previous measurements, such as water surface elevations, discharge values, and water depths. The calibration procedure is mostly trial-error tests until high accuracy is achieved, where the model's sensitivity is investigated for various coefficients and algorithms.

4.1 Sensitivity Analysis

Here, before moving to the analysis of the case study, Cache Creek, sensitivity analysis will be conducted to observe the effect of different model parameters on the output values. This sensitivity analysis will be conducted for two of the HEC-RAS Applications Guide test cases. This study will be performed to understand the effect of the methods, algorithms, and river-related parameters provided in the software package of HEC-RAS and to determine the most appropriate parameters representing the river reach correctly in the case study.

4.1.1 Test Case of the Beaver Creek

There are available methods to represent real-world scenarios in modeling hydraulic structures inside river systems. This test case investigated the effect of chosen methods in representing real-world analysis. In this test case, two different methods, pressure/weir flow and energy methods were used to compare the reliability of the results according to the observed values (Warner et al., 2021). As stated by Warner et al. (2021), in creating geometric data, the cross-sections were interpolated to gain more accuracy where the energy loss due to the bridge and the abrupt change in the velocity can be more dominant in the flow. The test case was run according to the study region's water surface elevations observed in the 1974 flood. The first method used in the modeling is the Energy Method, where the method's accuracy was compared with another simulation with the Pressure/Weir Method. The model outputs can be seen in Figure 4.1—representing the water surface elevation data obtained from the model run and the observed data obtained from the field measurements for both cases through the main channel distance starting from the most upstream location (0 m) using the Energy Method and the Pressure/Weir Method.

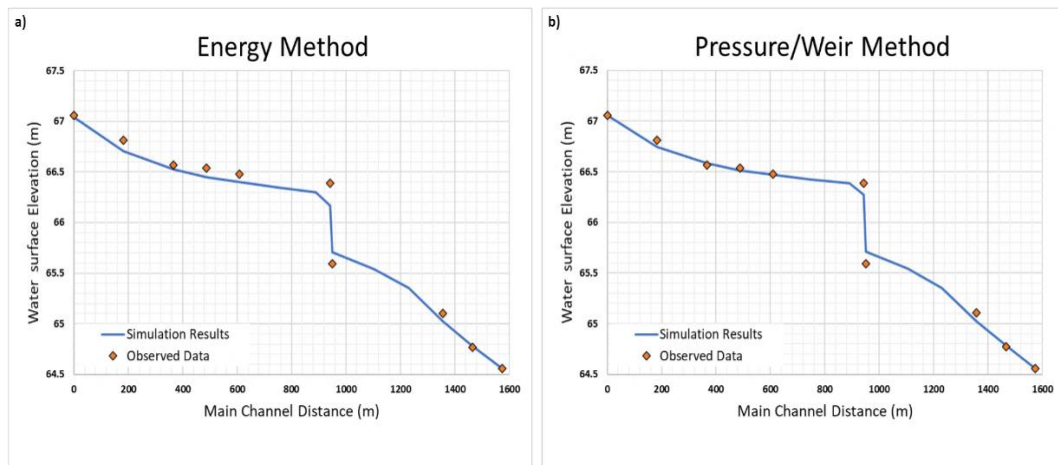


Figure 4.1. The comparison of simulation results and the observed values of water surface elevations using a) the Energy Method and b) the Pressure/Weir Method.

The simulation results are represented with continuous lines throughout the river reach, where the dots represent the measured observations for specific locations in the reach. The correlation between the simulated results and observed values was investigated to compare the performance of both methods. Figure 4.2 represents the correlation results of these methods. The correlation, R^2 , is examined between the observed values and simulation outputs. In the determination of the relationship between them, Equation 4.1, which is adapted from Lin et al. (2017), was used.

$$R^2 = \left(\frac{\sum_{i=1}^m (O_t - \bar{O})(P_t - \bar{P})}{\sqrt{\sum_{t=1}^T (O_t - \bar{O})^2} \sqrt{\sum_{t=1}^T (P_t - \bar{P})^2}} \right)^2 \quad 4.1$$

Where O and P being the observed and simulated values, respectively. The values range between $t = 1$ to m , where T accounts for the time series values. $\bar{O}$ and $\bar{P}$ are the mean observed and simulated values, respectively. As seen in the figure, the R^2 value for the Energy Method is 0.9916, while this value is 0.9952 in the Pressure/Weir Method. Pressure/Weir provides more consistent and better results when compared to the Energy Method, which makes it preferable.

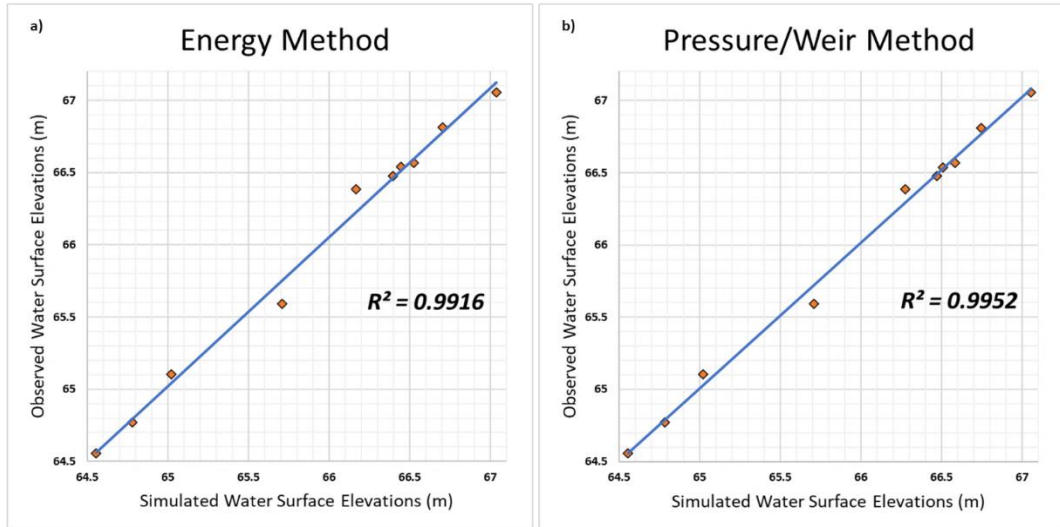


Figure 4.2. The correlation, R^2 , results for water surface elevations using a) the Energy Method and b) the Pressure/Weir Method.

4.1.2 Test Case of Mississippi-Ohio Junction

The second part of the sensitivity analysis was done with another test case in the HEC-RAS documentation, Manning's n (bottom roughness coefficient) calibration. This test case was selected for this thesis to investigate the modeling parameters and behavior of the junction portion of a natural river system. HEC-RAS provides large-scale outputs for the flow paths and has a reasonable estimation of the affected areas by the flow with a correct model. Investigating the junction portion in detail requires narrowing the scale and focusing on a relatively minor part. The behavior change at the junction can be examined by conducting a 3D CFD analysis on a much smaller scale with comparatively high resolution and precision. Later in Chapter 5, a more detailed CFD analysis will be performed for river confluences. Getting back to this topic, the researchers conducted this case to entirely represent a natural river system with the model created at the confluence portion of the Ohio and Mississippi Rivers (Warner et al., 2021). They used reliable historic gauge flow values to calibrate their model for this case. The reason for them to conduct this study was to assess Manning's n value, which has the most variability compared to the other input parameters. They used a feature of HEC-RAS that calibrates n values automatically. The confluence portion of the Mississippi and Ohio Rivers is shown in Figure 4.3, where the Upper Mississippi, Ohio, and Lower Mississippi reach the storage areas and observation stations seen. The green lines represent the cross sections, the blue-filled regions represent the storage areas, and the navy blue line represents the river reaches. All observation stations are shown in detail. Upper Mississippi Reach and Ohio Reach conjugate at the Ohio_Miss point in Figure 4.3, and they flow together as Lower Mississippi Reach.

This sensitivity analysis was conducted with a focus on the junction location of three reaches. In the first part, different n values were tested, and their effect on the model outputs was investigated. A representative n value was obtained for both reaches without the automatic calibration feature of HEC-RAS. In the second part, the original flow values of both upstream reaches (Upper Mississippi and Ohio) were

disturbed by a multiplier factor to observe the change in the water surface elevations, velocity, and Froude number resulting from these multiplications. Also, the energy grade line (EGL) of the most downstream station of Lower Mississippi Reach was graphed as a function of time through the simulation with the thought of taking it as a baseline in the later CFD simulations where a relatively small channel geometry will be created, and the turbulence effects will be observed.

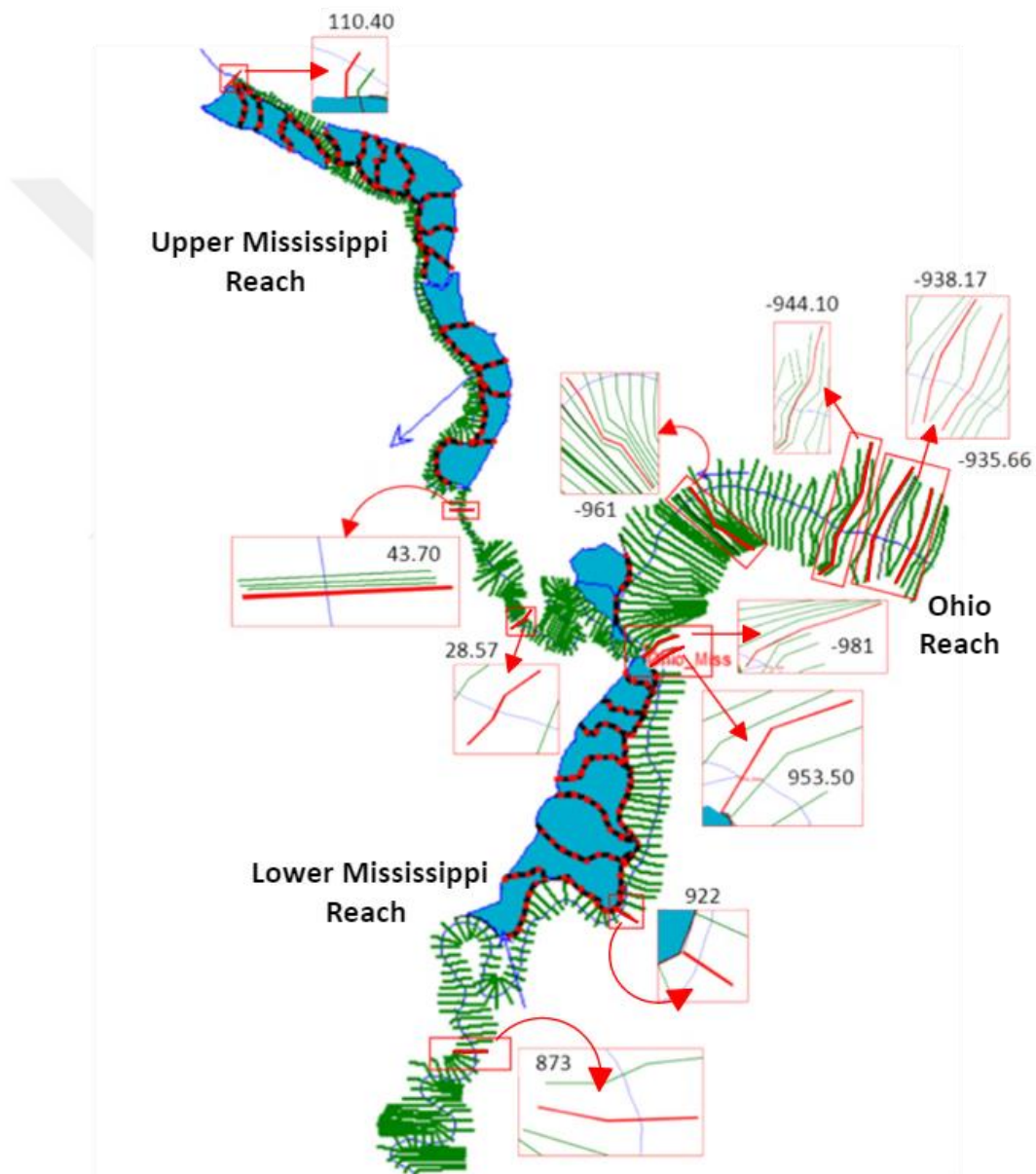


Figure 4.3. The geometry of the Mississippi-Ohio junction river reaches, Adapted from Brunner (1994).

4.1.2.1 Manning's Roughness n

In this sensitivity study, various n values were tested, and the effect of change of n values on multiple parameters, such as Froude number, water surface elevations, and velocity, was investigated. This study has three reaches where the Upper Mississippi and Ohio Reaches are the upstream, and Lower Mississippi is the downstream reaches. Observation stations were selected and listed in Table 4.1 for both the upstream and downstream reach. They were categorized with their reach name and the distance from this river system's most upstream location, the upstream station of the Upper Mississippi Reach (0th km). The upstream reaches collide in the 177th km (junction location) and flow as the Lower Mississippi Reach afterward, accepting the whole distance of both reaches as a single line where the most upstream location of Upper Mississippi Reach is the 0th km. Ohio Reach starts from the 100th km on this single-line distance. HEC-RAS has a feature that automatically calibrates Manning's n value during the iteration process in the solution procedure. Tests regarding the n values were performed for every observation station using the auto-calibration feature and various n values to obtain an n value that gives accurate model results compared to the observed data.

The water surface elevation change for all observation stations was investigated with specified n values. Figure 4.4 shows specified n results of a specific station, station #28.57, on the Upper Mississippi Reach by virtue of the distance of this station to the most upstream station. The auto-calibrated simulation was represented with the blue line; a black triangle represents the observed data. The orange, red, and green lines represent where n is equal to 0.05, 0.03, and 0.02, respectively.

The first analysis was performed with the auto-calibration feature. It was realized that n values change during the simulation process for every time step with the auto-calibration feature of HEC-RAS. This feature uses the field measurement data and forces HEC-RAS to intersect the simulation results with the observed field values.

This is why the auto-calibrated and observed values match, and the correlation between them is higher.

Table 4.1. Available observation stations of the original test model for both reaches.

<i>Reach</i>	<i>Observation Station</i>	<i>Distance from the most upstream point (km)</i>
Upper Mississippi	110.40	0
	43.70	106.7
	28.57	131.1
Ohio	-935.66	103.8
	-938.17	107.9
	-944.10	117.3
	-961.00	145
	-981.00	176
Lower Mississippi	953.50	177.3
	922.00	227.3
	873.00	306.7

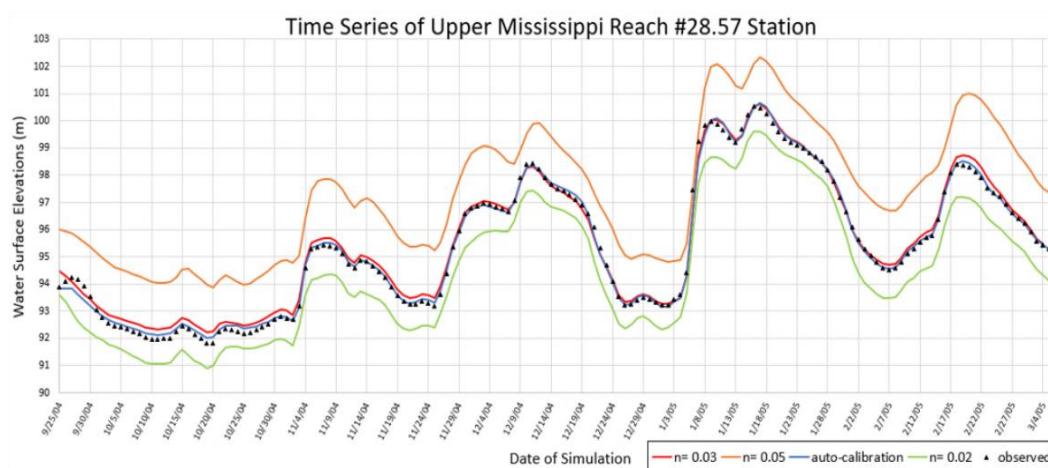


Figure 4.4. The change of water surface elevations for station #28.57 throughout the simulation period.

The simulations were repeated for different n values to determine the most appropriate n for that region without the auto-calibration feature. This analysis was performed and repeated for all the observation stations where the parameters such as the correlation coefficient, R^2 , and Nash-Sutcliffe model efficiency coefficient, NSE , was calculated for specified n cases. NSE is a widely utilized statistical metric for evaluating the estimation performance of models in hydrology and other domains that the success of hydrological models. It is recognized as an appropriate metric to monitor the quality of fit between observed and simulated data. This fit was investigated using Equation 4.2, with Q_{sim} , Q_{obs} and $\overline{Q_{obs}}$ being the simulated, observed, and mean observed discharge values, respectively. t is the simulation time starting from 1 to the end of the simulation, T .

$$NSE = 1 - \frac{\sum_{t=1}^T (Q_{sim}^t - Q_{obs}^t)^2}{\sum_{t=1}^T (Q_{obs}^t - \overline{Q_{obs}})^2} \quad 4.2$$

The computed coefficient results are displayed in Table 4.2—representing the values regarding station #28.57. However, this study aims to determine the model parameters where station distance is essential. Despite the distance from the most upstream station, station #28.57 has high correlation results.

Table 4.2. Different correlation results at station #28.57
between observed values and simulation results with specified n values.

n value	R^2	NSE
auto-calibrated	0.9961	0.9960
0.02	0.9921	0.8414
0.03	0.9947	0.9911
0.05	0.9715	0.3455

Since the correlation was accepted as sufficient for n equal to 0.03, tests were not repeated for different n values around 0.03. At this point, the objective is to obtain the dates where the minimum and maximum water surface elevations are observed. For that, two analyses were performed. Initially, the neighboring stations of the

junction point were tested to see the junction effect. These stations will be named ‘junction stations’ from hereupon. After that, the closest stations to the junction stations were tested. In Figure 4.5, at the left-hand side, the whole river reaches are shown objecting the junction portion with a red rectangle where, at the right-hand side, a closer look at the junction location is displayed with the corresponding reach names and station numbers on the right-hand side to identify the near-junction behavior by using the closest stations to the junction point and the nearby stations. The junction stations are 1.39, -981, and 953.5 on Upper Mississippi, Ohio, and Lower Mississippi Reaches. Stations 2, -980, and 953 are the closest stations to the junction stations, where the second part of the analysis will be performed for these stations. These stations are shown in Table 4.3—representing their reaches and distances from the junction point (Ohio_Miss).

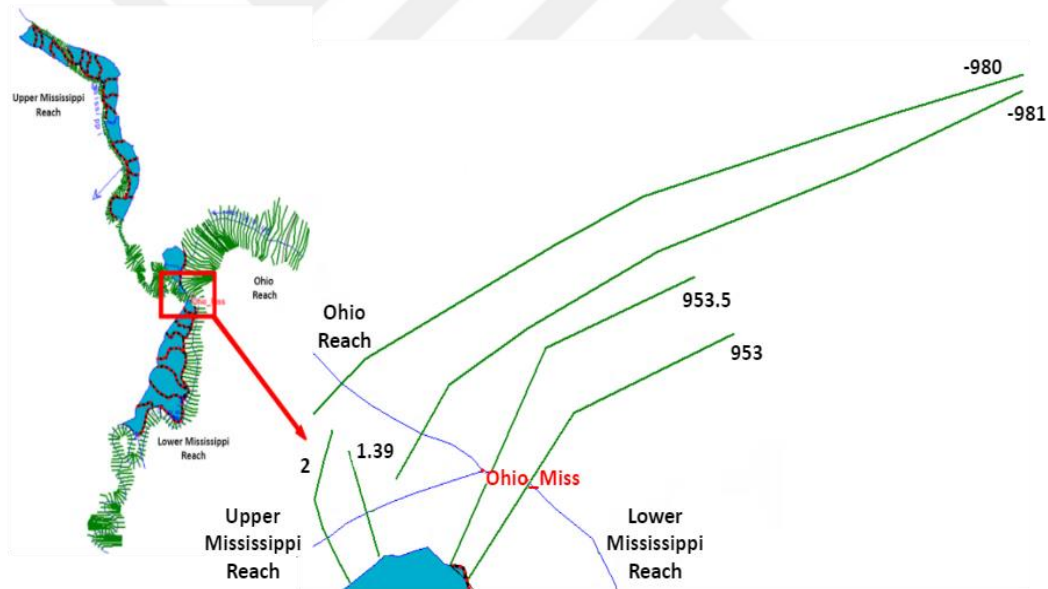


Figure 4.5. Representative schematic of the junction portion focusing on the junction location (Reprinted from Brunner, 1994).

The simulations considering the junction effect were processed for these stations, grouping them as the junction stations (1.39, -981, and 953.5) and the adjacent stations to junction stations (2, -980, and 953). The impact of distance as well as the junction was investigated.

Table 4.3. Selected stations to analyze the flow behavior near the junction area and the distances from the junction point.

<i>Reaches</i>	<i>Stations</i>	<i>Distance from Junction Point (km)</i>
Upper Mississippi	1.39	2.18
	2	3.13
Ohio	-981	1.42
	-980	3.05
Lower Mississippi	953.5	0.7
	953	1.39

The water surface elevation, velocity, and Froude number change were examined. A study was performed initially to explore the correlation related to the water surface elevations for junction stations throughout the simulation period. It was noticed that the results were identical since only one line is shown in Figure 4.6-a. The correlation, R^2 , between these values was calculated as 0.999987. This study was repeated for the other selected stations, and the correlation, R^2 , was calculated as 0.999993—shown in Figure 4.6, b. The margins were kept constant while creating the charts for the two cases. The correlation between Figure 4.6-a and -b was also calculated, and the result was 0.999999, nearly 1.

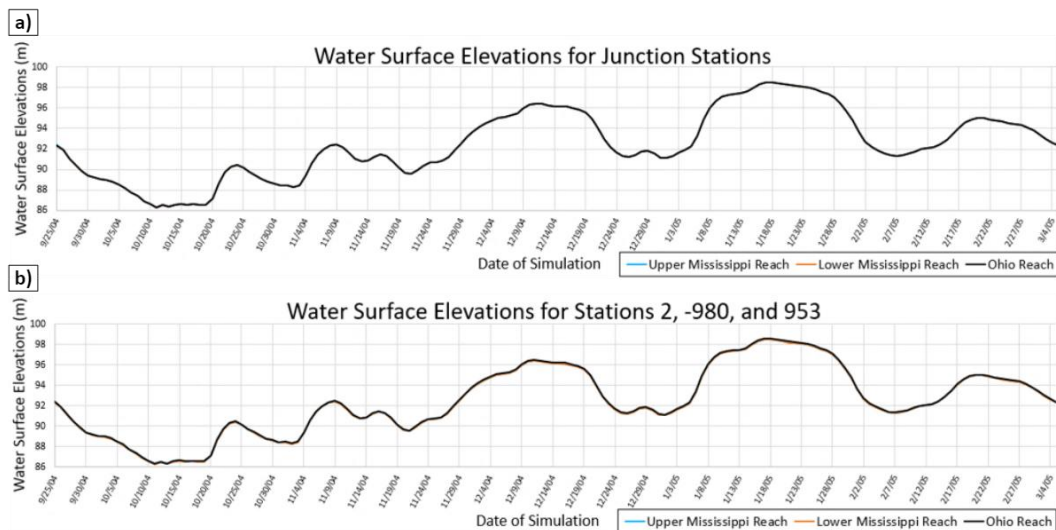


Figure 4.6. Water surface elevation as a function of time focusing a) junction stations and b) adjacent stations to the junction stations.

In these studies, minimum and maximum water surface elevations were obtained on the 11th of October, 2004, and the 18th of January, 2005, respectively. The simulation results for the velocity change of the selected stations were compared and demonstrated in Figure 4.7. Figure 4.7-a shows the velocity change during the entire simulation period for junction stations, where Figure 4.7-b represents the adjacent stations' results. Another comparison was made for Froude number values, similarly illustrated by two figures.

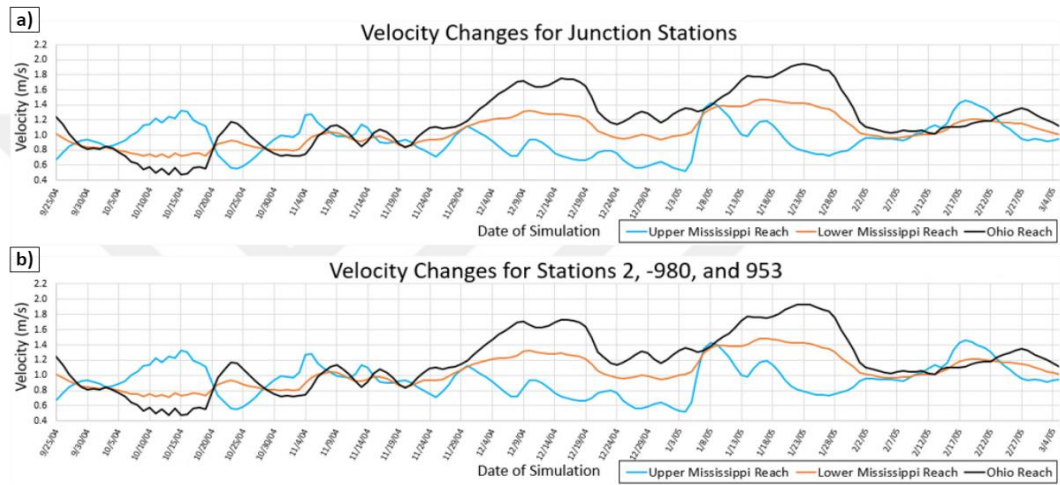


Figure 4.7. Velocity variations as a function of time focusing a) junction stations and b) adjacent stations to the junction stations.

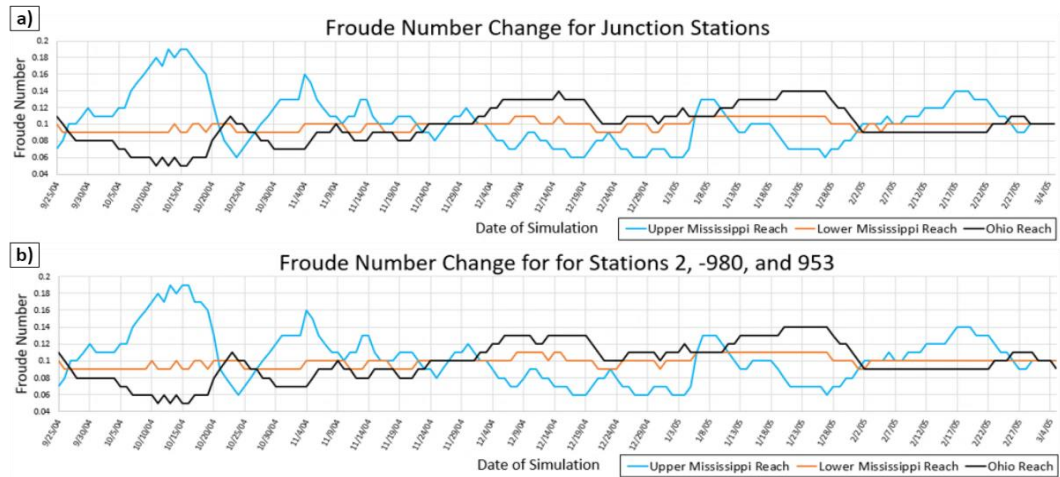


Figure 4.8. Froude number variations as a function of time focusing a) junction stations and b) adjacent stations to the junction stations.

Figure 4.8-a illustrates the Froude number change for junction stations. Clearly, this change for Lower Mississippi Reach primarily depends on the value of Ohio Reach. This test was repeated for adjacent stations, and similar results were obtained — Figure 4.8-b. When the findings were compared, it was seen that the correlation results were identical.

After defining the representative n value and the dates where the minimum and maximum water surface elevations were observed, the extreme points, the tests were performed for these dates and represented as a distance function. Figure 4.9 demonstrates the change in the water surface elevations on the selected dates for the maximum and minimum values through the entire reach. In the following figures, obtained by the simulations as a function of distance, 0th km is the upstream location of the Upper Mississippi Reach, and 350th km is the downstream location of the Lower Mississippi Reach. The junction is at 176th km, and the upstream location of Ohio Reach has 76 km distance from the junction location. The effect of water surface elevation on the reach velocities and Froude numbers was also investigated for the selected dates.

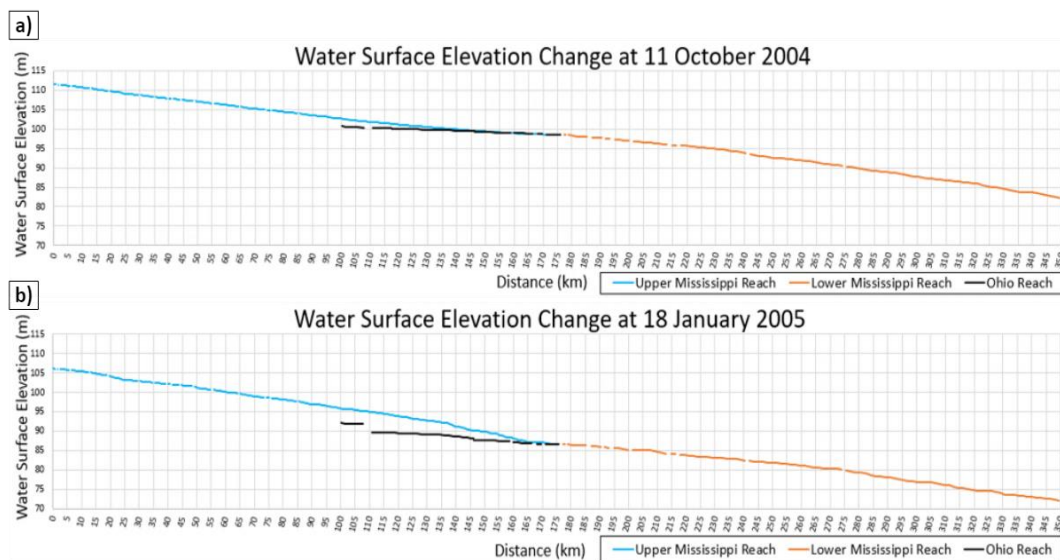


Figure 4.9. Water surface elevations as a function of distance for a) 11th October 2004 (maximum) and b) 18th January 2005 (minimum).

Velocity changes for the dates when the maximum and minimum water surface elevations occurred were obtained as a function of distance and shown in Figure 4.10. Similarly, the graphs were generated for Froude Numbers. All three reaches are represented with their corresponding Froude Number values through the river distance in Figure 4.11.

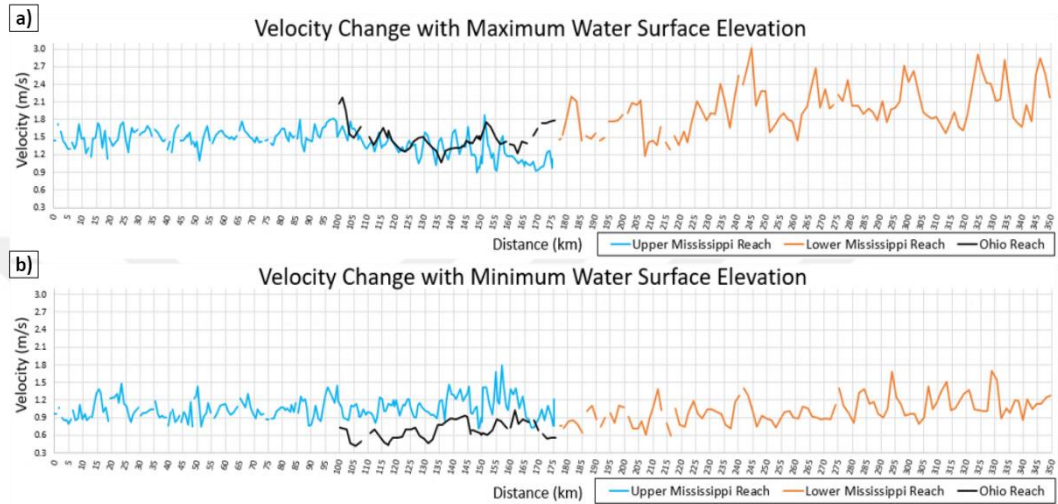


Figure 4.10. Velocities as a function of distance through the entire reach for a) 11th October 2004 (maximum), and b) 18th January 2005 (minimum).

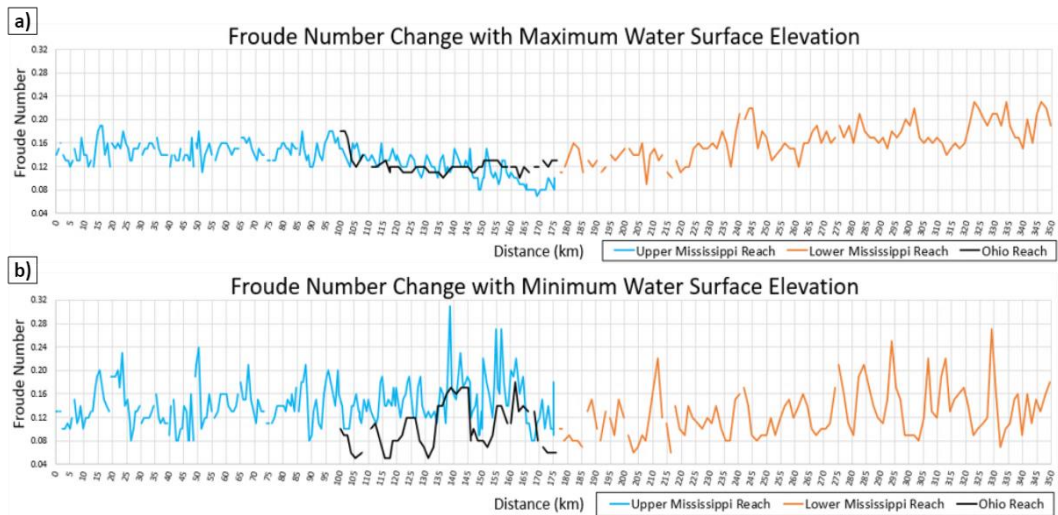


Figure 4.11. Froude number changes as a function of distance for a) 11th October 2004 (maximum), and b) 18th January 2005 (minimum).

This test investigated the junction effect on the downstream reach for water surface elevations, velocities, and Froude numbers. It was seen that although the Upper Mississippi Reach is longer in size compared to the Ohio Reach, the effect of Ohio on the downstream was significant when the analyzed parameters were taken into account. Also, the correlation results regarding the water surface elevations, velocities, and Froude numbers show similarities for the selected stations. The test results presented above were for the simulations using Manning's n value as 0.03. All observation stations were examined afterward for the study to be more precise. In the evaluation process, since only water surface elevation values are available as observed data, the comparison was made between the observed and the simulated values. The obtained results of both the automatic-calibration feature and Manning's n equal to 0.03 are listed in Table 4.4, where correlation, R^2 , and Nash-Sutcliffe model efficiency coefficient, NSE values are shown. It was noticed that, without the calibration feature of HEC-RAS, relevant results could be obtained using an appropriate n value if there is no available observed data. HEC-RAS gives related results without a vast amount of information.

Table 4.4. Comparison for all observation stations
 n value is 0.03, and using the automatic calibration feature, with R^2 , and NSE values.

<i>River Station</i>	<i>n value</i>	R^2	NSE
110.40	auto-calibrated 0.03	0.9973 0.9959	0.9973 0.9899
43.70	auto-calibrated 0.03	0.9971 0.9962	0.9969 0.9848
28.57	auto-calibrated 0.03	0.9961 0.9947	0.996 0.9911
953.50	auto-calibrated 0.03	0.9982 0.9887	0.9878 0.9723
922.00	auto-calibrated 0.03	0.9883 0.9882	0.988 0.9862
873.00	auto-calibrated 0.03	0.9877 0.9836	0.9875 0.9768
-935.66	auto-calibrated 0.03	0.9772 0.9734	0.9763 0.9292
-938.17	auto-calibrated 0.03	0.9952 0.9896	0.9942 0.9563
-944.10	auto-calibrated 0.03	0.9824 0.9800	0.9817 0.9538
-961.00	auto-calibrated 0.03	0.9971 0.9927	0.9968 0.9844
-981.00	auto-calibrated 0.03	0.9882 0.9887	0.9878 0.9724

4.1.2.2 Effects of Flow Conditions at Mississippi-Ohio Junction

Previous tests were to investigate the change in Manning's n values and the applicability of different n values on both reaches. This section aims to examine the flow change effects on the water surface elevations, velocities, and Froude numbers. The behavior change after the junction point is sought to be investigated under various flow conditions where the nearer portion of the junction location of the river system is focused. The upstream boundary conditions for Upper Mississippi and Ohio reaches were defined as 'Flow Hydrograph' type. The structures inside both reaches mostly have the 'Lateral Inflow Hydrograph' type boundary conditions where the DSS values were used in the original case. For this study, hypothetical cases were created without the DSS values of flows. Some multipliers were used to manipulate the upstream discharges to investigate the impact for both reaches.

This section aimed to examine the effect of the change in the flow values on the water surface elevations, velocities, and Froude numbers through the river system. The impact of a slight disturbance in the flow values was investigated in this section. The initial conditions for both upstream reaches were kept constant, but the flow hydrographs were modified by multiplying them with some values. The modified flow values were the flow hydrographs of the upstream reaches, Upper Mississippi and Ohio. The coefficients, C_m and C_o represent the multiplying factors of the flow hydrographs of Upper Mississippi and Ohio Reaches, respectively. The analyses were performed while keeping one of these coefficients constant and changing the other one to observe the impact one by one. Table 4.5 describes the change in these coefficients. The values of '1.00' represent the actual values for both C_m and C_o coefficients. Other multiplication factors were defined as 1.25 and 0.75 to manipulate the flow hydrographs of upstream reaches slightly since the test case was to examine a real-world system.

Table 4.5. Flow hydrograph multiplying factors
 C_m and C_o .

Reach	Coefficients	Multiplying Factors
Upper Mississippi	C_m	1.25
		1.00
		0.75
Ohio	C_o	1.25
		1.00
		0.75

The following figures were generated by focusing on the junction area. The distance values start from 112 km downstream from the most upstream location showing the junction portion in detail, and end at 212th km. The margin change was due to focus only on the junction portion for better representation. The difference in the water surface elevations when the flow hydrographs were multiplied by the multiplication factors is shown in Figure 4.12. The change in the C_m and C_o values separately was investigated — Figure 4.12.-a and Figure 4.12-b, respectively.

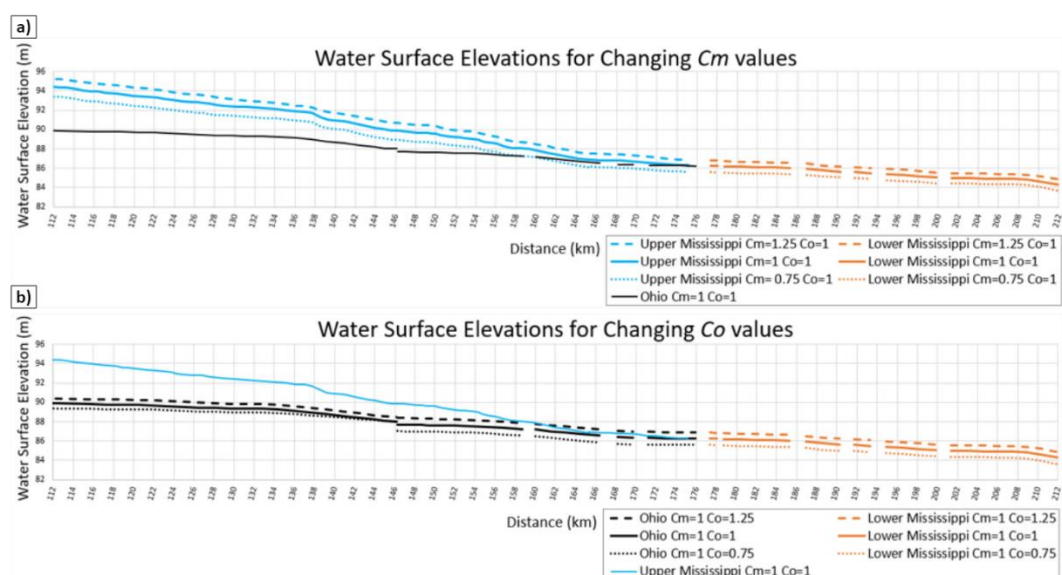


Figure 4.12. Water surface elevations
of a) Upper Mississippi Reach, and b) Ohio Reach with multiplying factors.

This test was repeated to observe the velocity change as a function of distance starting from the same margins as in the water surface elevations above. Figure 4.13 illustrates the velocity behavior after the multiplication factors C_m and C_o were applied to the flow hydrographs separately.

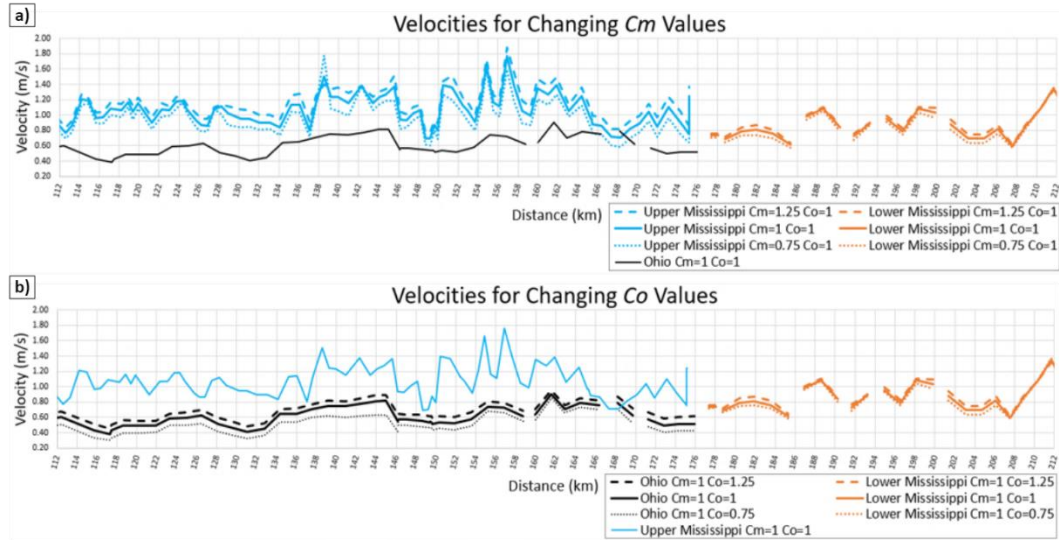


Figure 4.13. Velocity variations of a) Upper Mississippi Reach, and b) Ohio Reach with multiplying factors.

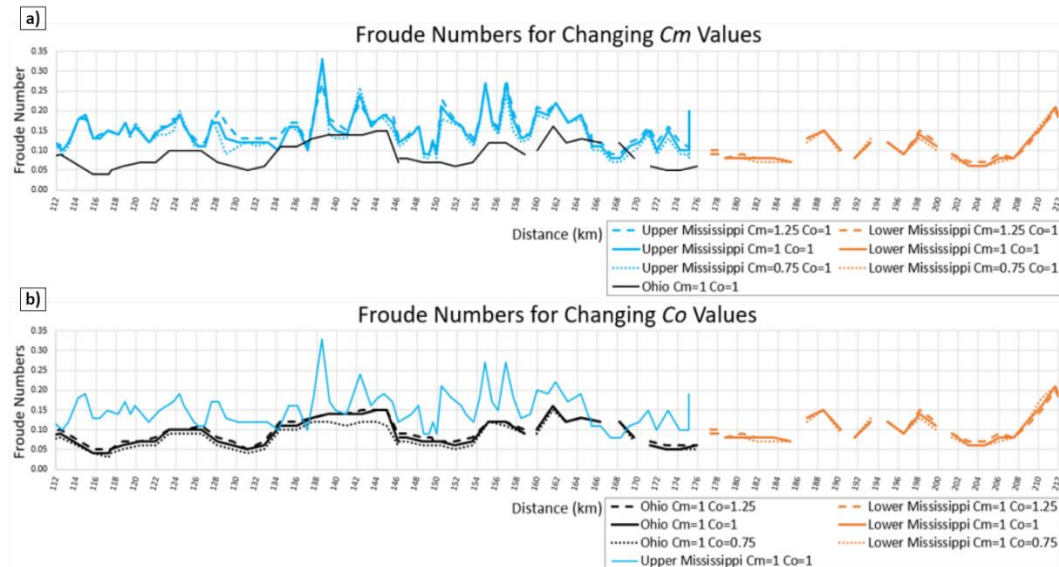


Figure 4.14. Froude Number variations of a) Upper Mississippi Reach, and b) Ohio Reach with multiplying factors.

A similar schematic was generated for the change in Froude numbers for the junction region. The margins were kept the same, and the effect of multiplication factors of C_m and C_o were observed separately and displayed in Figure 4.14. The plots were again generated for the junction stations as a time function throughout the simulation. In Figure 4.15, the water surface elevation changes under the effect of multiplication factors C_m and C_o is shown. Since the water surface elevation values for both junction stations were identical, only three lines are shown for both different C_m and C_o values.

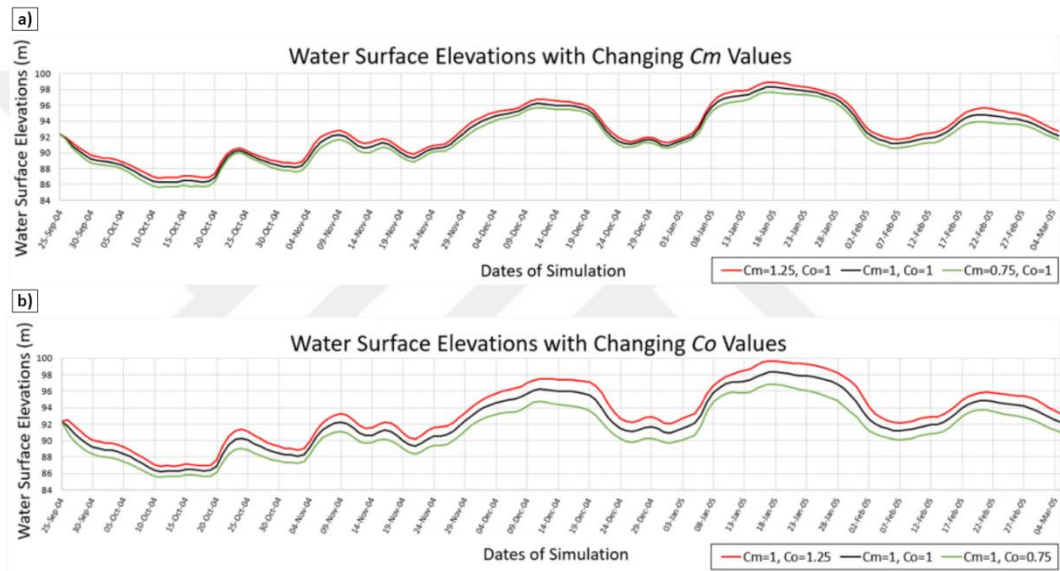


Figure 4.15. Water surface elevations for junction stations with different multiplication factors of a) Upper Mississippi Reach and b) Ohio Reach.

Changes in velocities and Froude numbers were also investigated through the simulation period, and graphs were generated as a function of time for the junction stations. The effect of the change in multiplication factors C_m and C_o in velocities is shown in Figure 4.16 separately. Figure 4.17 illustrates the Froude number change in the junction stations when the same multiplication was applied to the flow hydrographs. The sensitivity analysis is complete with these tests, and the results will be discussed in the Results and Discussions section later in the thesis.

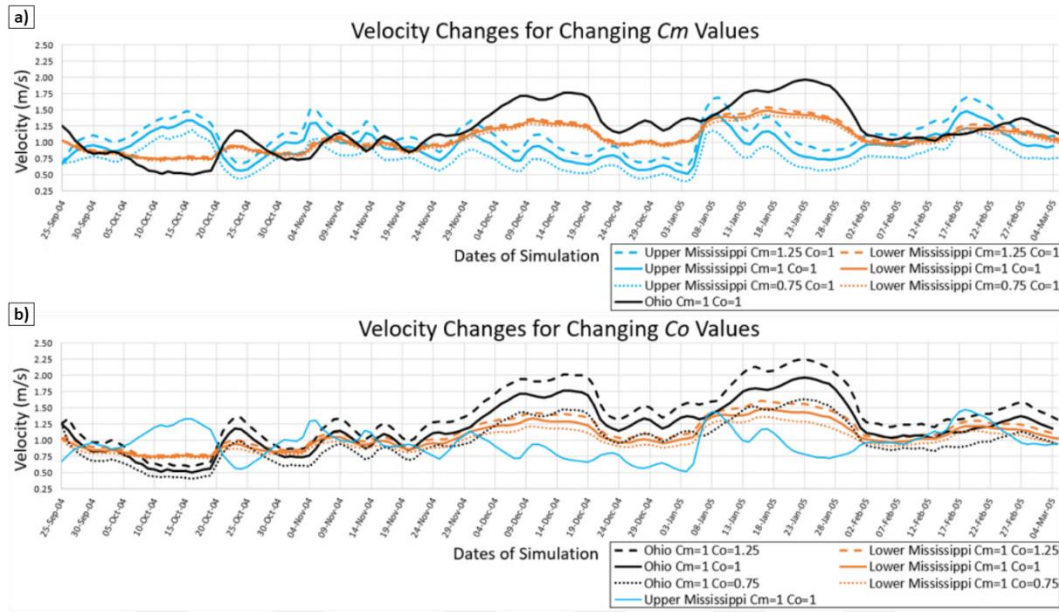


Figure 4.16. Velocities for junction stations with different multiplication factors of a) Upper Mississippi Reach and b) Ohio Reach.

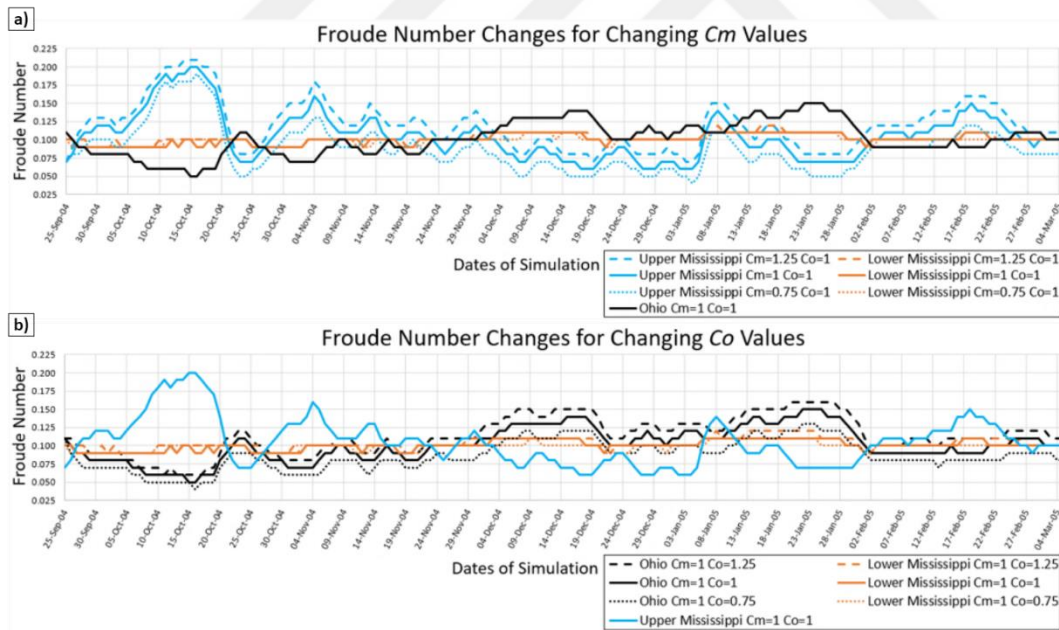


Figure 4.17. Froude numbers for junction stations with different multiplication factors of a) Upper Mississippi Reach and b) Ohio Reach.

4.2 Case Study of 9 km Cache Creek River Reach

The main reason for the sensitivity analysis was to examine the effect of different parameters on the model. The parameters that give high correlation results were selected from this analysis. In addition, the parameters that make a difference in the results were discovered. In the case study, the plan is to focus on these parameters. The model for the test case is specified as Cache Creek in the United States. This creek is located near Yolo County, Northern California. After defining the study region, the portion that will be studied is also determined. Figure 4.18 shows the whole region where orange arrows label the stations of Yolo on the left and Road 102 on the right part of the domain, where an 8780 m reach will be modeled. Yolo station is located near Yolo County, where most of the population resides, and Road 102 station is a rural area covered by farmlands. The model will be built for this region using the monitored hydraulic time series data, and the study will be performed with information about this region's characteristics and channel cross-sections. Since the bathymetry of the region is partially available, this study will be performed with actual values. The region's characteristics will be examined in this study with these real-world data.



Figure 4.18. Case study area
stations Yolo the upstream, and Road 102, at the downstream.

4.2.1 Description of the Study Domain

The region of interest for this part of the study is Cache Creek, located near Yolo County, Northern California. While Cache Creek's south fork originates from the

oldest and largest lake in California, Clear Lake (Goldman & Wetzel, 1963), the north fork of it flows out of the Indian Valley Reservoir together with Bear Creek (Fischer, 2014). Domagalski et al. (2004) state that Cache Creek is located in the lower part of the predominantly rural area, Cache Creek basin, which occupies nearly 3000 km². Historically, three flood events occurred in the basin, 1939, 1940, and 1983, with an intense storm in 2006. A broader perspective adopted by Higgins and Kamman (2010) notes that in the basin, the winters are cool and moist, the summers are dry and warm, while most of the precipitation falls as rainfall, majorly from November through March. In the watershed, hydraulic regulation is controlled by four main structures: Cache Creek Dam, Indian Valley Reservoir Dam, Davis Dam, and Capay Dam; however, only the Capay Dam intersects with the study of interest for this thesis. The study of interest stays in the basin's middle part, while most of the runoff in the watershed is fed from the upper part of it (Fischer, 2014). The region between the Yolo and Road 102 stations, nearly 9 km, will be studied since there is available flow data for Yolo and Road 102 stations for a specific period. This region is selected since the discharge values at the upstream station, Yolo can be obtained from the monitoring analysis, the flow hydrograph can be generated according to the monitoring values, and the model results can be compared with the monitored values at the Road 102 station. Observed data supports this study in determining the related parameters, such as Manning's *n* values of Cache and the model's reliability. The sensitivity analysis will use these algorithms to determine the most appropriate ones representing the real-world scenario. As stated in the large number of studies related to the effect of climate change in California, the variations in the hydrologic regime are caused by the discrepancy in the atmospheric conditions (Cayan et al., 2008; Villarini et al., 2018).

4.2.2 Model Setup

The digital elevation model (DEM) data for the case study was obtained from The National Map of the United States Geological Survey (USGS). Firstly, two different

1/3 arc-second format geographic coordinated DEMs were used. Since the region of interest occupies a small portion of the map and is located in the intersection of two DEMs, these files were merged and clipped to produce one single. This process was done using ArcGIS, where GIS stands for Geographical Information System, a comprehensive mechanism that captures, stores, displays, and analyzes geographical data. The river model was generated with this single file. It was seen that the elevations in Cache Creek were not represented correctly due to the resolution of this file and the lack of bathymetric data. It was decided that the DEMs could be used as small, tiled files with a resolution of 1 meter from the 3D Elevation Program (3DEP) of USGS. Two tiled files containing the whole study region were used in the case study. In Figure 4.19, these tiled files are shown with orange squares.

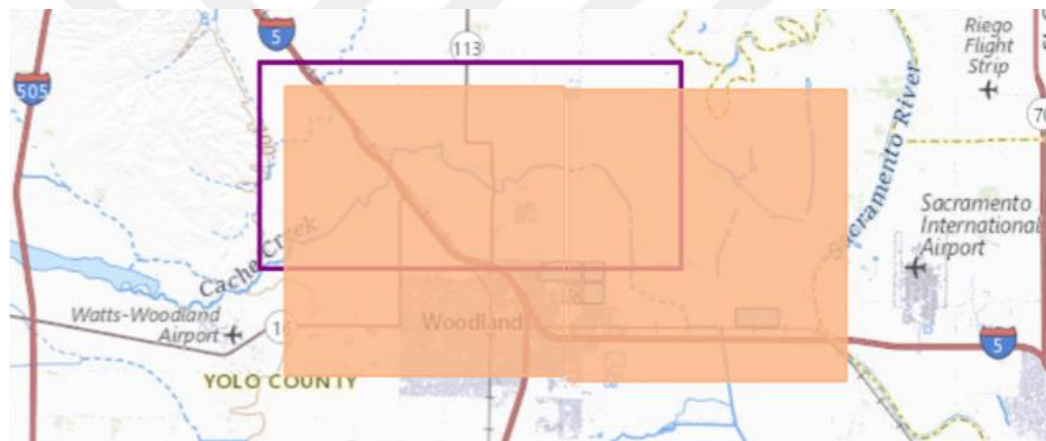


Figure 4.19. Tiled files covering the entire region
(Retrieved from the National Map of USGS).

All tiled files were downloaded in GeoTIFF format, and the spatially referenced information was contained in these files. TIFF stands for Tagged Image File Format, where Geo is the shortened form representing the geographically referenced file. The files downloaded had a 1-meter x 1-meter resolution in the x and y axes covering 10000-meter x 10000-meter regions separately. ArcGIS has a tool named ArcMap that has a toolbox that performs various tasks. The tiled files have added the catalog in ArcMap and opened together. These files were merged into a single Raster file, a spatial data format that divides the study of interest into rectangle sections, also

called cells, where these cells can hold multiple values (Hijmans, 2023). Figure 4.20 shows DEMs –signifying the region between Yolo and Road 102. These separate DEMs were merged together by paying attention to the intersected sides of them could have the correct elevation values.

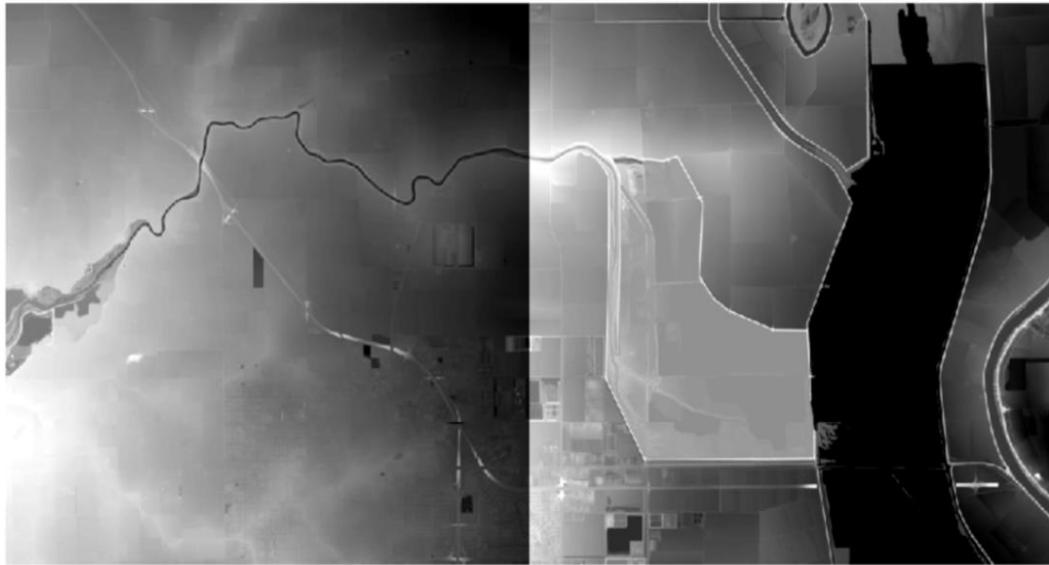


Figure 4.20. Tiled DEM files covering the study region with the stations of Yolo and Road 102 in 1-meter x 1-meter resolution.



Figure 4.21. DEMs merged together for case study.

During the simulation, the resolution of DEMs affects the simulation duration. Because of that, these files should be clipped containing only the necessary parts. The clipping procedure was performed by creating a shapefile, firstly drawing a polygon in Google Earth Pro and saving it in '.kml' format containing the information about the coordinates of this specified region where KML stands for Keyhole Markup Language (Werneck, 2008). This '.kml' file was converted into a '.shp' file using an ArcMap conversion tool. The merged DEM file was clipped, taking the boundaries of the created shapefile – Figure 4.22. The file size was reduced by clipping, and the possible simulation duration was shortened. In the modeling part of this study, in HEC-RAS, this elevation model will be used as the primary terrain file in .tif format. The region covering the selected river stations, Yolo upstream and Road 102 downstream, are presented in this DEM file. The simulation will be done between these two stations in this study.

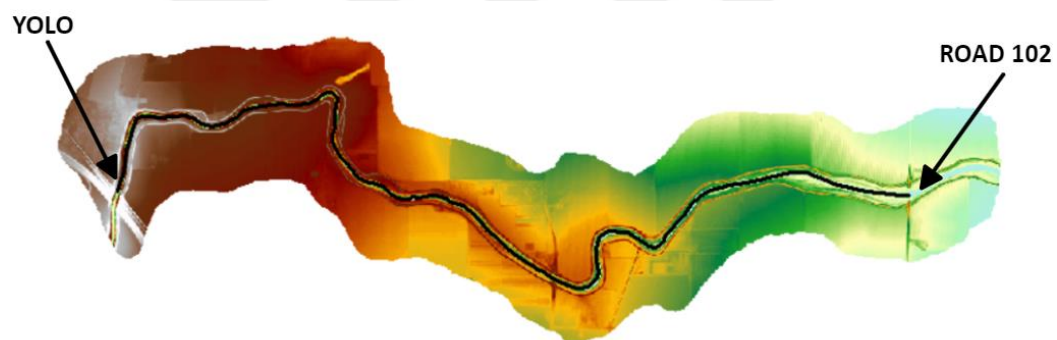


Figure 4.22. Clipped DEM file representing the entire study region.

4.2.3 HEC-RAS Model

RAS Mapper was used while creating the river geometry with its bank lines and cross sections considering the elevation profile of the region. Possible flow lines were added to the geometry. The checks were needed in every cross-section, including the bank line point locations. In the first part, only upstream and downstream cross-sections were used to represent the entire reach when the interpolation between these cross-sections was made by HEC-RAS automatically. An unsteady flow analysis

was performed with a flow hydrograph between 11th December 2014 and 21th January 2015. The first analysis was to define the representative n value of the reach between the specified stations. Various n values were tested, and the correlation, R^2 , and the Nash-Sutcliffe model efficiency coefficient, NSE , between the simulated results and observed values was calculated. Table 4.6 shows R^2 and NSE results obtained by testing different n values. It was seen that using n as 0.033 gives higher results compared to other values when two cross-sections were used. Since these studies were performed for varying n values for the left-right over-banks and main channel, 0.033 was the main channel's n value. The left and right over-banks were taken as 0.07 since the study aimed to keep the flow in the channel.

Table 4.6. Manning's n values of channel bottom with corresponding R^2 and NSE parameters.

n	R^2	NSE
0.025	0.9953	0.9950
0.030	0.9960	0.9956
0.031	0.9961	0.9957
0.032	0.9961	0.9957
0.033	0.9962	0.9958
0.034	0.9960	0.9955
0.035	0.9959	0.9954
0.040	0.9957	0.9951

The tests were repeated to investigate whether a difference was formed by changing the left and right over-bank n values. The change in R^2 and NSE results due to various overbank roughness values are shown in Table 4.7.

Table 4.7. Manning's n values for overbanks
With corresponding R^2 and NSE parameters.

Overbank n values	R^2	NSE
0.060	0.9956	0.9951
0.065	0.9958	0.9953
0.069	0.9958	0.9951
0.070	0.9962	0.9958
0.071	0.9957	0.9951
0.075	0.9956	0.9950
0.080	0.9955	0.9948

By using the Manning's values of 0.033 and 0.07 for channel and overbanks, respectively, a graph was generated for the simulation period comparing the observed and simulated discharge values for Road 102 station – Figure 4.23.

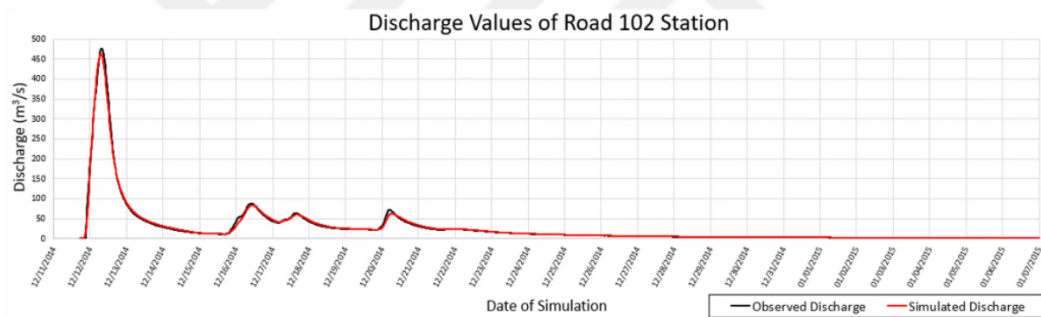


Figure 4.23. Observed and simulated discharges
for n is 0.033 for the most downstream station, Road 102.

When the high and low flow events separate by using the Manning's values of 0.033 and 0.07 for channel and overbanks, respectively, as in Figure 4.24, their corresponding R^2 and NSE values were obtained. The value of R^2 for high flow is 0.9956, and NSE is 0.9962. For the low flow condition, the corresponding R^2 is 0.9905, and NSE is 0.9923 (see Table 4.6). In high-flow conditions, R^2 and NSE parameters give better results when compared to those of the low flow conditions.

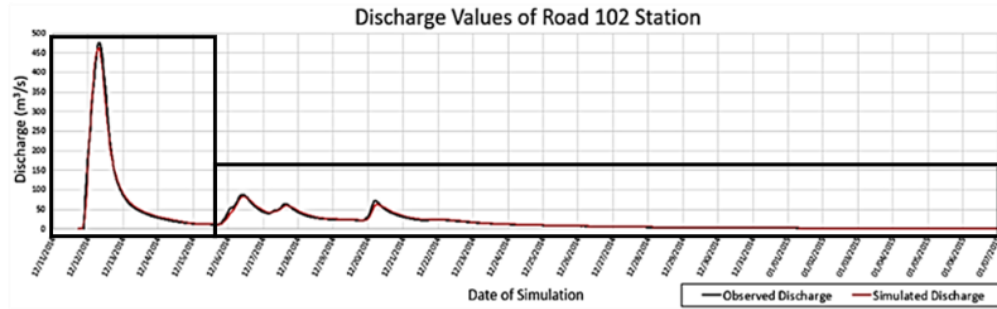


Figure 4.24. Separated hydrograph for high and low flows.

Table 4.8. R^2 and NSE values for flow conditions.

<i>Flow condition</i>	R^2	NSE
High Flow	0.9956	0.9962
Total Flow	0.9962	0.9958
Low Flow	0.9905	0.9923

Another analysis was performed by changing the cross-section number while running the model with the selected Manning's n value. The impact of the cross-section number used in the model domain on the model was investigated through the same simulation period. Table 4.8 shows the similarity results obtained by using different numbers of cross-sections. It was seen that, as the cross-section number increases, the reliability of the simulation results increases as well.

Table 4.9. Similarity results depending on the cross-section number.

<i>Cross Section Number</i>	R^2	NSE
2	0.6666	0.6504
3	0.8069	0.7963
4	0.8000	0.7886
5	0.9754	0.9729
6	0.9764	0.9739
7	0.9992	0.9988

4.3 Concluding Remarks

The junction portion of the Ohio and Mississippi Rivers was studied in Section 4.1, sensitivity analysis. The results in that section indicate that the junction portion's flow behavior cannot be fully understood since the model is 1D. A 3D model can be constructed for more detailed behavior investigation, and simulations can be performed afterward with the help of CFD. With this aim, Flow-3D will be used as a CFD tool, and the simulations will be committed to monitoring the effect of different junction angles. 1D or 2D depth-averaged codes fall short when modeling rapidly varying flow conditions.

An unsteady flow analysis was performed with a flow hydrograph between 11th December 2014 and 21st January 2015 for a 9 km reach of Cache Creek. As shown in Table 4.9, even with low 5 cross-sections, relatively good results ($R^2 = 0.9754$, and $NSE = 0.9729$) are obtained for routed flow magnitude and timing of the maximum flow in the downstream boundary at Road 102. It was also seen that, as the cross-section number increased, the reliability of the simulation results increased as well. Note that these simulations are performed when the flow is constricted within channel cross-sections and the focus of the study is not flow elevation. If the inundated area is the target, then the number of cross-sections and reliability of the river bathymetry becomes very important.



CHAPTER 5

THREE-DIMENSIONAL CFD MODELING OF CONFLUENCES

Confluences, where two or more river channels meet, are fundamental to all river systems. They are crucial points where substantial amounts of water and sediment are combined and flow into the main channel (Best, 1987). Efforts made in the last forty years to explore and solve the challenge of flow dynamics at junctions have encompassed various studies, one of which is the research carried out by Taylor in 1944. Although several studies have investigated the flow behavior at channel confluences, a widely accepted model that describes the dominant flow features at these junctions is needed (Best, 1987). Such a model could prove helpful in comprehending sediment transportation, deposition patterns, bed morphology, and the design considerations required at channel junctions. The sensitivity analysis was performed for a real-case scenario, including a junction portion of Ohio and Mississippi Reaches with HEC-RAS. The results showed that a one-dimensional model such as HEC-RAS gives reliable results for the general picture when two reach confluence; however, the flow behavior cannot be observed precisely due to the simplified nature of one-dimensional models. That is why conducting a three-dimensional small-scale analysis can help to understand the behavior near the river confluences. With this object in mind, a three-dimensional CFD analysis will be performed in this section.

Flow-3D will be used as a CFD tool to represent an open-channel free-surface flow. Flow-3D is a CFD solver where non-hydrostatic Navier Stokes Equations are solved, offering a robust solution that satisfies computational speed, accuracy, and simplicity (Kim et al., 2022). In the solution procedure, no assumptions are made regarding the accelerations of vertical fluids. It uses structured rectangular grids to generate meshes where individual control volumes can be defined as separate parts. Since the

open-channel flow acts as a multiphase region, the free surface is handled by the volume of fluid (VOF) and FAVOR methods in Flow-3D. One may check Flow Science, Inc. (2019) for detailed information regarding these methods. Tracking the free surface is dealt with in three steps, where the surface at any given cell is located and followed as a sharp interface within the mesh. This procedure is done by defining the volume of fraction, meaning each cell in the domain has a value of fraction where empty cells have the value of 0 and cells full of fluid take a value of 1. After that, the boundary condition is applied to the free surface by one fluid void approximation. The software-defined free-surface interface allows the user to specify only one fluid phase. After the simulations, the post-processing part is handled by FlowSight, a post-processor designed specifically for Flow-3D data.

In this chapter, the CFD models created will be presented with their corresponding results. First, the mesh independency will be investigated for a 90° confluence of two channels, and the results will be presented. The most appropriate model mesh will be selected depending on its capabilities of solving the multiphase fluid system with a lower computational cost and acceptable results. Then, the difference introduced by the selected turbulence models will be discussed. For that purpose, $k - \varepsilon$, $k - \omega$, and *RNS* models will be used since these are the accepted turbulence models used in many CFD tools. The turbulence model effects on the solution will be analyzed with a 90° confluence of two channels. The later tests will be performed with the selected turbulence model. Next, using the chosen turbulence model and mesh size, the analyses will also be performed for 60° and 30° channel confluences in order to investigate the confluence angle on flow dynamics. Specifically, parameters such as depth-averaged velocity, turbulent kinetic energy, surface velocity, total hydraulic head, and Froude Number will be investigated near the confluences.

5.1 Model Setup

In this study, three different model geometry were created by Autodesk Inventor, where 90°, 60°, and 30° channel confluences were represented. The following figures

show these geometries with the corresponding channel dimensions in millimeters. The three-channel geometries created for this study have zero slopes since a small portion of a river confluence is represented, and the slope is ignored near the confluence. There is a main and tributary channel in the following geometries, where lengths of them are 5.82 meters and 1.60 meters, respectively. The main channel inlets are 1.60 meters apart from the junction location of the tributary channel.

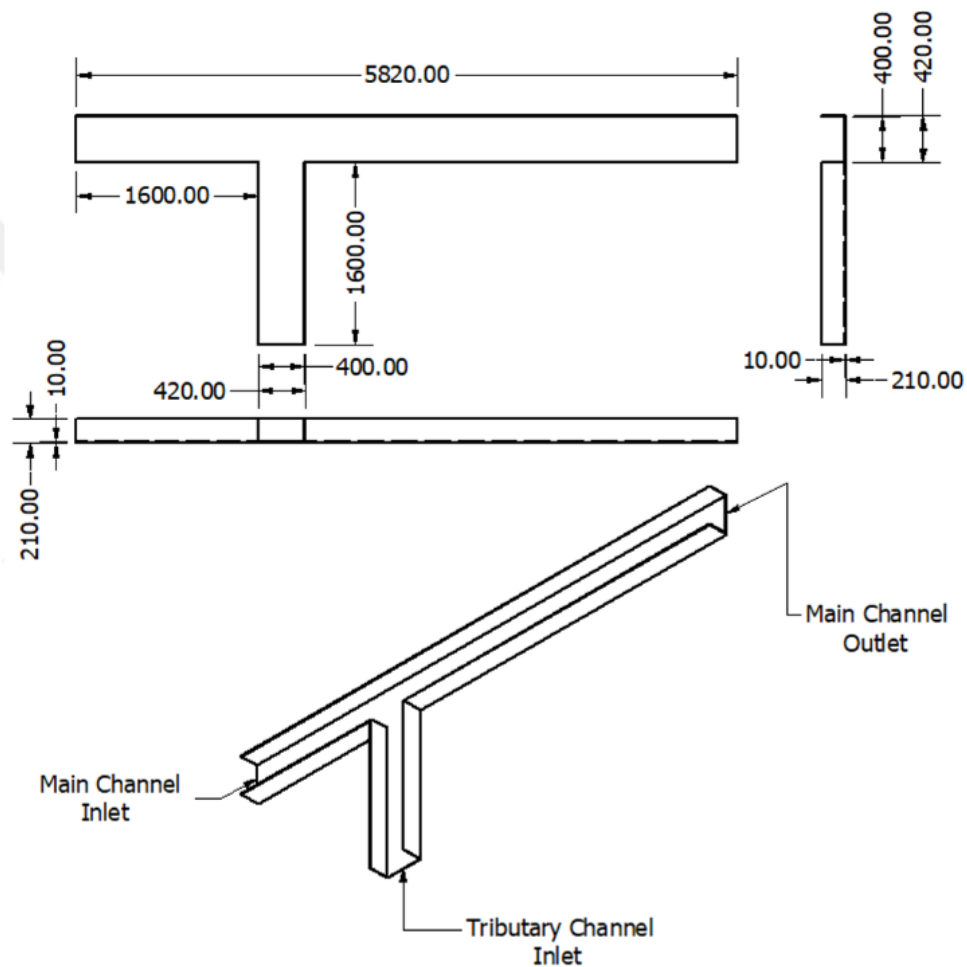


Figure 5.1. 90° confluence channel model generated in Autodesk Inventor where the dimensions are in millimeters. The plan and side views are presented with representative measurements.

The width and depth were defined as 0.40 meters and 0.20 meters for the main and tributary channels, respectively. The channels have walls of 0.01-meter thickness throughout the channel. In Figure 5.1, the 90° confluence geometry used in most of

the cases is presented. Figure 5.2 shows the model geometry of a 60° confluence channel where the angle between the main and tributary channels is 60°. The channel dimensions were the same as in the 90° confluence geometry.

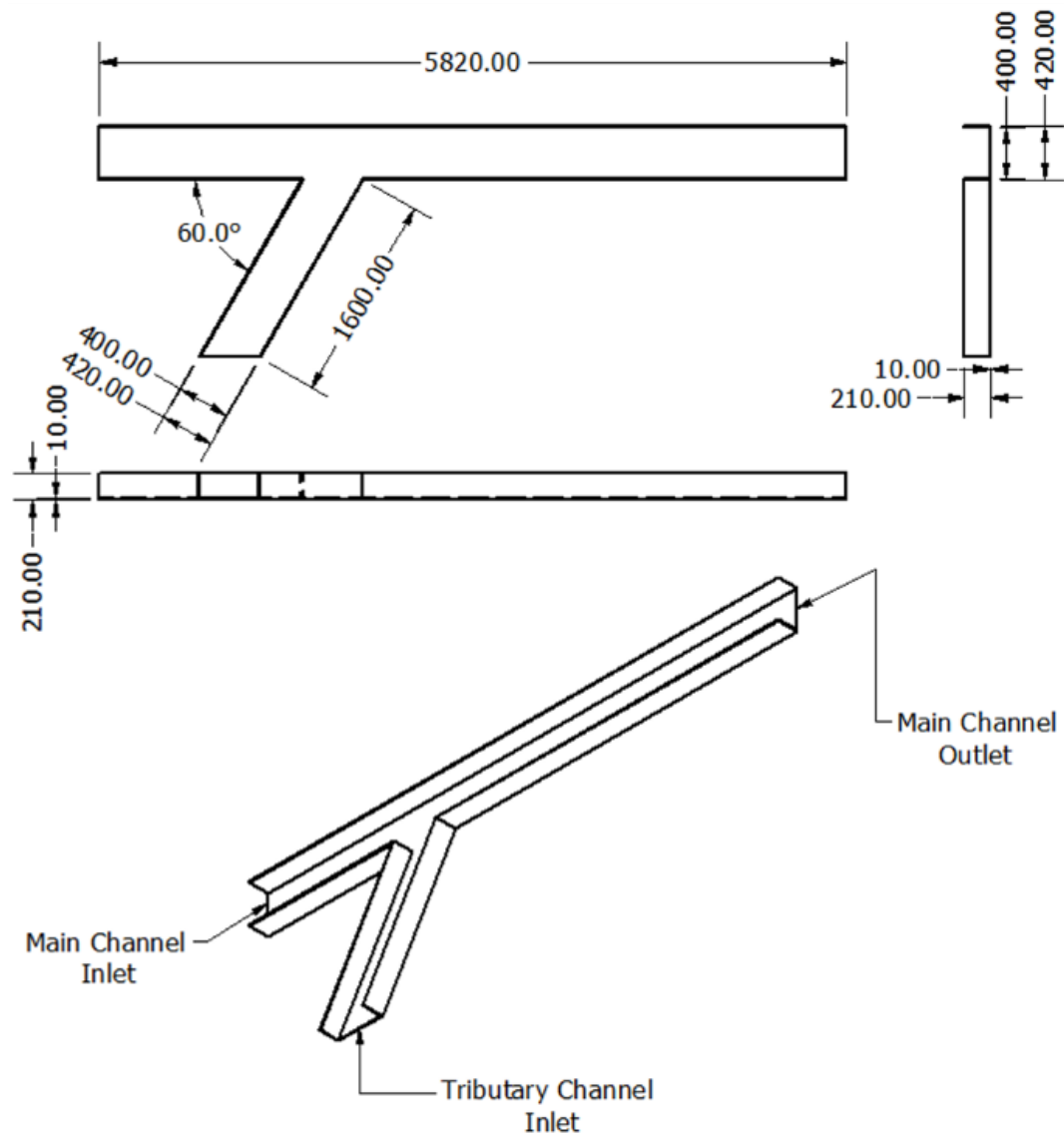


Figure 5.2. 60° confluence channel model generated in Autodesk Inventor where the dimensions are in millimeters. The plan and side views are presented with representative measurements.

Figure 5.3 shows the model geometry of 30° confluence channels. The measurements in both three figures were in millimeters. Figure 5.3 shows the geometry of a 30° channel confluence. The main and tributary channel lengths were

kept the same in 90° and 60° confluence geometries. It may be practical to remark that as the angle between two channels decreases, the length of the tributary channel stays the same.

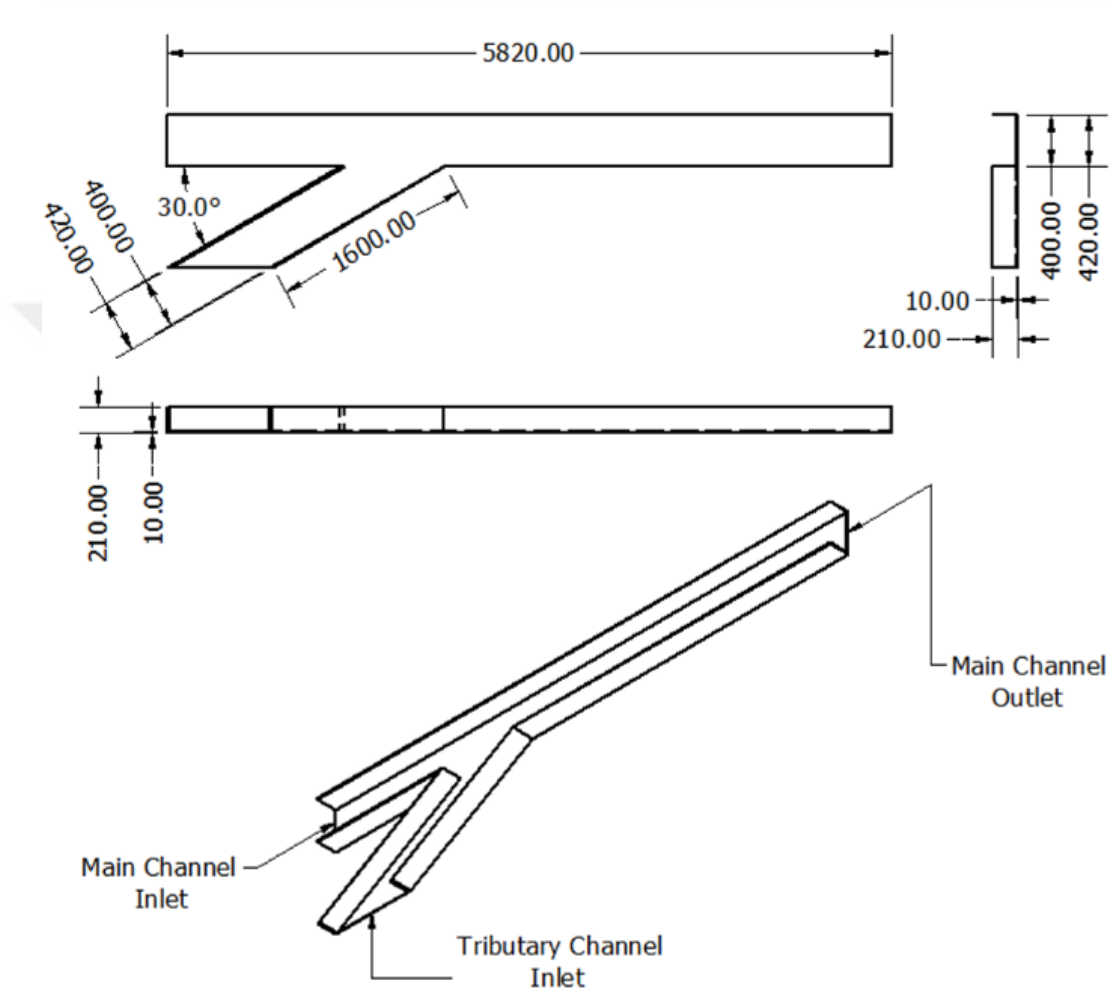


Figure 5.3. 60° confluence channel model generated in Autodesk Inventor where the dimensions are in millimeters. The plan and side views are presented with representative measurements

The reason for choosing a rectangular cross-section for both channels was to keep a uniform Froude number (Hager, 1989) in these channels. After creating all the model geometries, they will be moved to the Flow-3D geometry editor, where all the essential model inputs will be decided, and the meshes will be generated.

5.2 Mesh Independence

According to Almohammadi et al. (2013), many researchers suggest that enhancing the mesh can serve as a solution to mitigate errors caused by various factors, including rounding, iterative, and discretization errors. As stated in the study of I. Chirica et al. (2019), a high-quality mesh is an initial stage that guarantees the solver in generating an accurate solution while requiring less computing power. It may be practical to say that a low-quality mesh falls short while generating outputs and providing convergence. When the mesh size decreases, it is expected to observe peak values in the dynamic pressure since the solution converges to a more accurate result due to the increase in the number of mesh cells (I. Chirica et al., 2019). Bates et al. (2005) indicate that mesh independence focuses on using increased mesh resolution to analyze truncation errors systematically and can also be used to verify solution accuracy. Vast numbers of grid cells may be required through the flow domain to ensure mesh independence in the lower sections of the flow. This results in inefficient computation. In this study, mesh independence analysis was performed for the 90° confluence channel model using $k - \varepsilon$ turbulence model by mesh sizes of 6 (finer), 8, and 10 (coarser) millimeters in the horizontal plane. Figure 5.4 shows each mesh size in the horizontal plane, focusing on the confluence portion in 90° channel confluence geometry.

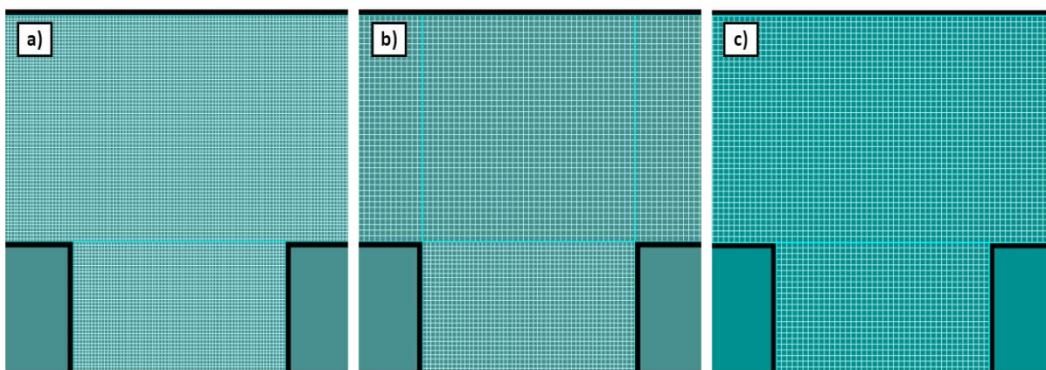


Figure 5.4. Generated 6,8, and 10 millimeters meshes focusing on the confluence portion in 90° confluence channel geometry.

It was seen that Flow-3D renders a warning message for the user as ‘the solution is nearly steady-state’ when the tolerance is in the range of acceptable convergence. This analysis aimed to reach the equilibrium solution in a smaller simulation duration, so each simulation was stopped when the simulation reached nearly equilibrium conditions. The corresponding number of nodes for these sizes and the simulation durations are presented in Table 5.1.

Table 5.1. The number of meshes generated for each mesh size.

<i>Mesh size</i>	<i>Number of Meshes</i>
6 mm	2,735,007
8 mm	715,600
10 mm	593,600

Flow-3D allows users to select a point probe to obtain information for different hydraulic properties. In this mesh independence test, the probe location was chosen at the downstream end of the channel, where the streamlines are parallel. For velocity, corresponding results until the equilibrium condition are shown as a function of time in Figure 5.5.

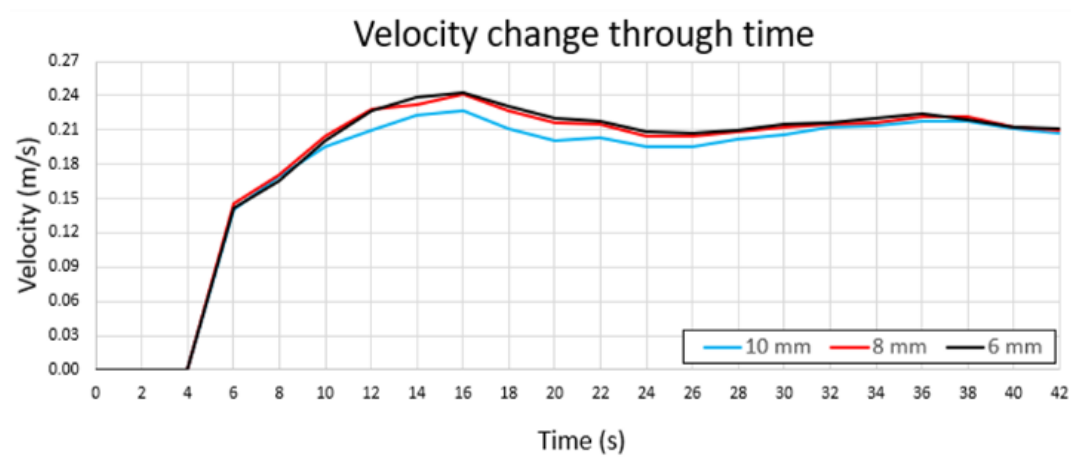


Figure 5.5. Mesh independence investigation for velocity for mesh sizes of 6 mm, 8 mm, and 10 mm.

The difference in velocities when mesh sizes are six- and eight-millimeters and eight- and ten-millimeters were calculated as 1.15% and 1.01%, respectively. These values indicate that a mesh-independent solution is achieved with a 10-millimeter mesh size. Simulation time for a 6-millimeter mesh is 6 hours and 46 minutes, and that for a 10-millimeter mesh is 55 minutes. As such, simulations in the following sections will use 10-mm mesh. Figure 5.6 shows the generated meshes for each confluence angle with 10-millimeter mesh sizes for 90°, 60°, and 30° confluences.

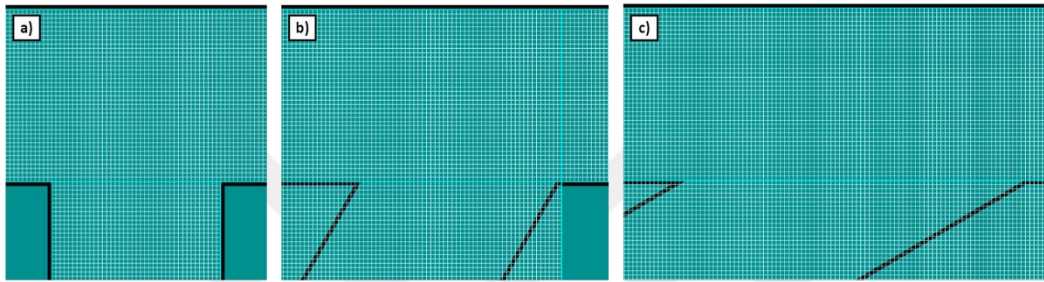


Figure 5.6. Generated 10 mm meshes focusing on the confluence portion for a) 90° confluence, b) 60° confluence, and c) 30° confluence.

The total number of mesh cells for 10 millimeters mesh sizes is 593,600 for 90°, 1,236,600 for 60°, and 1,250,400 for 30° confluences. After the mesh generation, an initial water depth was defined for each model where the depth was 0.10 meters. The simulation duration was 120 seconds, long enough for the system to reach equilibrium. The boundary conditions for the inlets of main and tributary channels were selected as ‘velocity,’ and the initial velocities were determined as 0.10 m/s. For the channel walls, the ‘wall’ boundary condition was used. The outlet of the tributary channel, which coincides with the main one, was defined as ‘continuative.’ Finally, the free surface was described as a ‘symmetry’ boundary condition.

5.3 Turbulence Models

Here, the capabilities of three turbulence models, namely *standard* $k - \varepsilon$, $k - \omega$, and *RNG* $k - \varepsilon$ models, will be investigated for only 90° river confluence in modeling flow dynamics near the confluences. As noted in the previous section, the

time for the models to reach an equilibrium-state solution differs. The following figures will present the results for the equilibrium-state solutions of each case. In this section, different flow parameters will be discussed, and according to the effects of these models on the solution results, the most appropriate turbulence model will be decided and used in later studies. According to Wilcox (2010), each turbulence model discussed in this study has its strengths and weaknesses, and the choice of model depends on the specific flow configuration and desired level of accuracy.

The *standard $k - \varepsilon$* turbulence model is a computationally efficient two-equation model that solves equations for turbulent kinetic energy and dissipation rate (Menter, 1994). The *$k - \omega$* turbulence is known for its ability to predict complex flows such as separation and reattachment. Still, it may be less accurate in predicting transitional flows and is more computationally expensive than the *standard $k - \varepsilon$* model (Pope, 2000). According to Speziale and Thangam (1992), the *RNG $k - \varepsilon$* turbulence model is an enhanced version of the *standard $k - \varepsilon$* model. This model is more accurate than the *standard $k - \varepsilon$* model for flows with high turbulence levels, but it is more computationally expensive and may require additional calibration for specific flow configurations (Akram Abbaspour & Kia, 2014).

With the capabilities of these models, different parameters were compared later in this section for equilibrium conditions. Equilibrium states are reported at 67 seconds in *standard $k - \varepsilon$* , 120 seconds in *$k - \omega$* , and 74 seconds in the *RNG $k - \varepsilon$* models. The following figures are the plan view of 90° confluence channels using 10 millimeters mesh sizes. Figure 5.7 compares these models' turbulent kinetic energies (TKE) with the same legend. The TKE is more visible in *standard $k - \varepsilon$* and *RNG $k - \varepsilon$* models compared to *$k - \omega$* model. It can be seen that the *$k - \omega$* model fails to capture the TKE values for this analysis with this geometry and flow conditions and underestimates this variable on the surface when compared with other turbulence models.

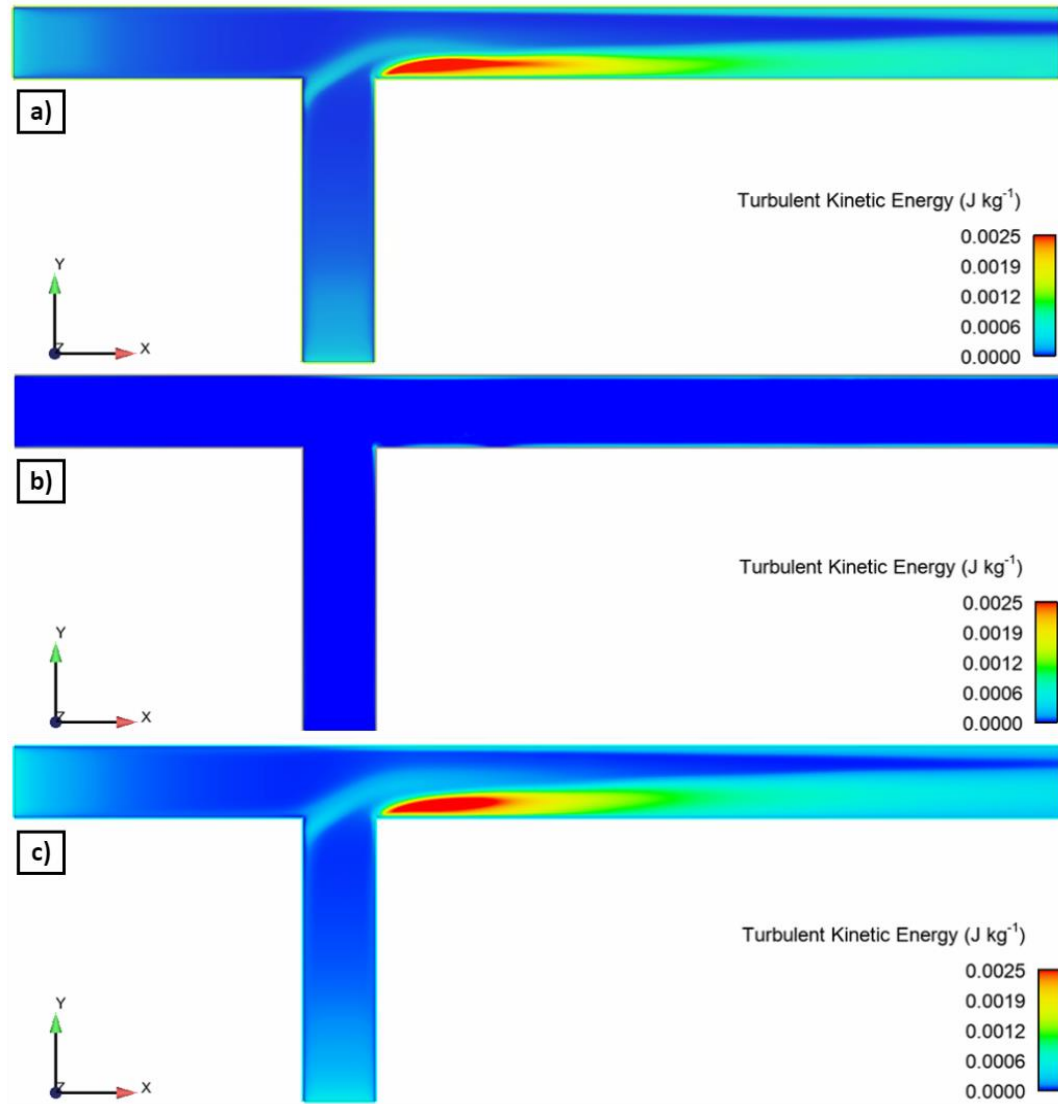


Figure 5.7. TKEs in plan view for 90° river confluence by three turbulence models: a) *standard $k - \varepsilon$* ; b) *$k - \omega$* ; and c) *RNG $k - \varepsilon$* .

Streamlines generated from the velocity profiles by these three turbulence models are shown in Figure 5.8. The most dynamic streamlined profile was achieved with *$k - \omega$* turbulence model. The vortexes generated near the flow separation region moved forward and backward during the simulation. The shape of it has a changing profile considering its width and length. *standard $k - \varepsilon$* model gives a partially dynamic profile, whereas *RNG $k - \varepsilon$* model has a stable profile during the simulation. With *RNG* model, it was as if the simulation reached an equilibrium state comparatively early than the other turbulence model simulations.

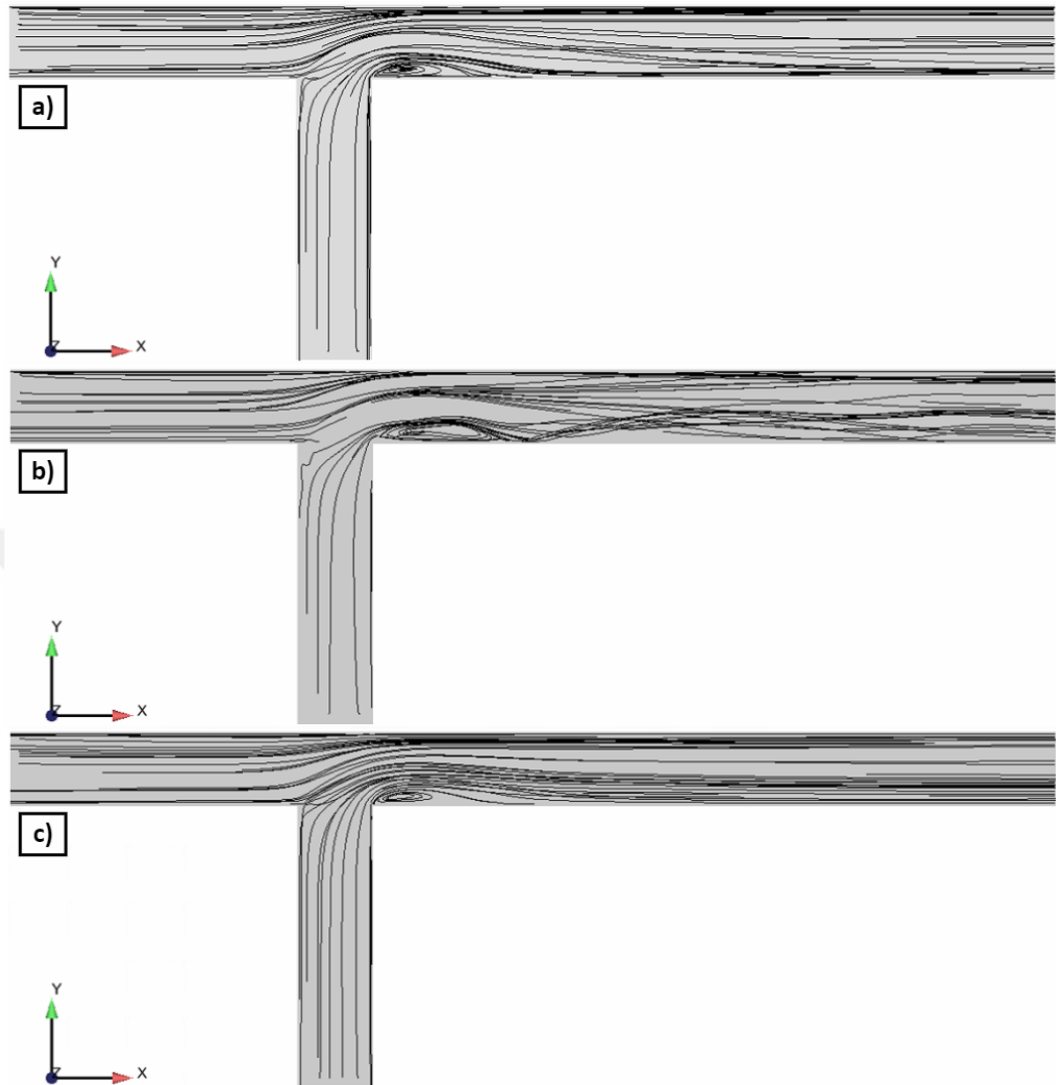


Figure 5.8 Streamlines in plan view for 90° river confluence
by three turbulence models: a) *standard $k - \varepsilon$* ; b) *$k - \omega$* ; and c) *RNG $k - \varepsilon$* .

In Figure 5.9, surface velocities for the three turbulence models are shown. One may notice that after the 90° confluence, the separation just downstream of the junction is evident since this region has formed a vortex. The results of *standard $k - \varepsilon$* and *RNG $k - \varepsilon$* models are approximately similar, where the reattachment lengths were generated almost on the same plane. The reattachment length refers to the distance downstream from a point where a fluid flow separates from a surface due to encountering an obstacle (Graziani et al., 2017).

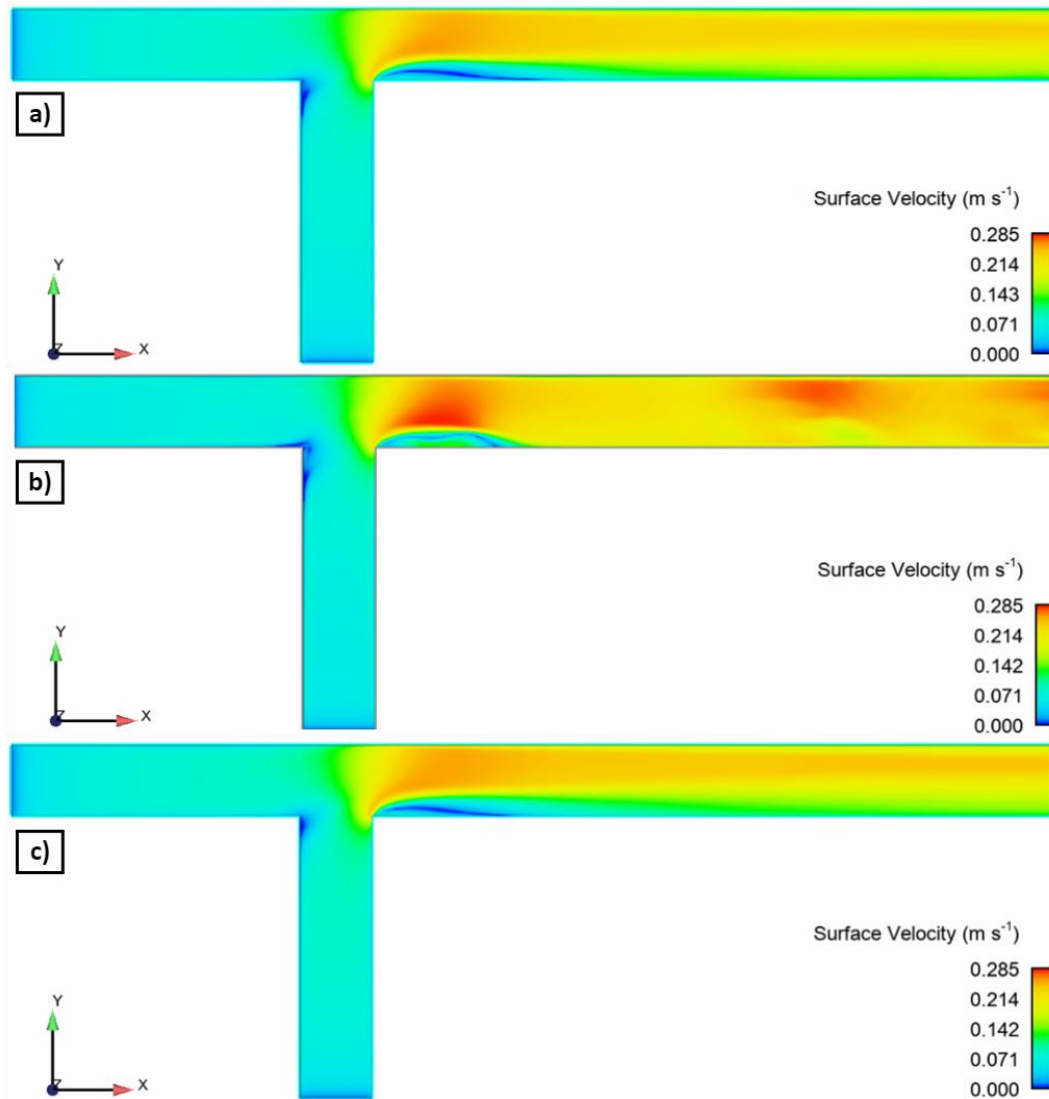


Figure 5.9 Surface velocities for 90° river confluence in plan view by three turbulence models: a) *standard $k - \varepsilon$* ; b) *$k - \omega$* ; c) *RNG $k - \varepsilon$* .

When the fluid flow encounters the obstacle, it loses velocity and cannot overcome the obstacle's pressure gradient, resulting in a region of recirculating flow known as a separation bubble (Iehisa Nezu & Nakagawa, 1989). In Figure 5.10, the depth-averaged velocities of these models are shown. The velocity distribution in $k - \omega$ model is oscillatory. It is due to the dynamic nature of $k - \omega$ model that the solution is always dynamic with moving vortex cores, even though the simulation duration was higher.

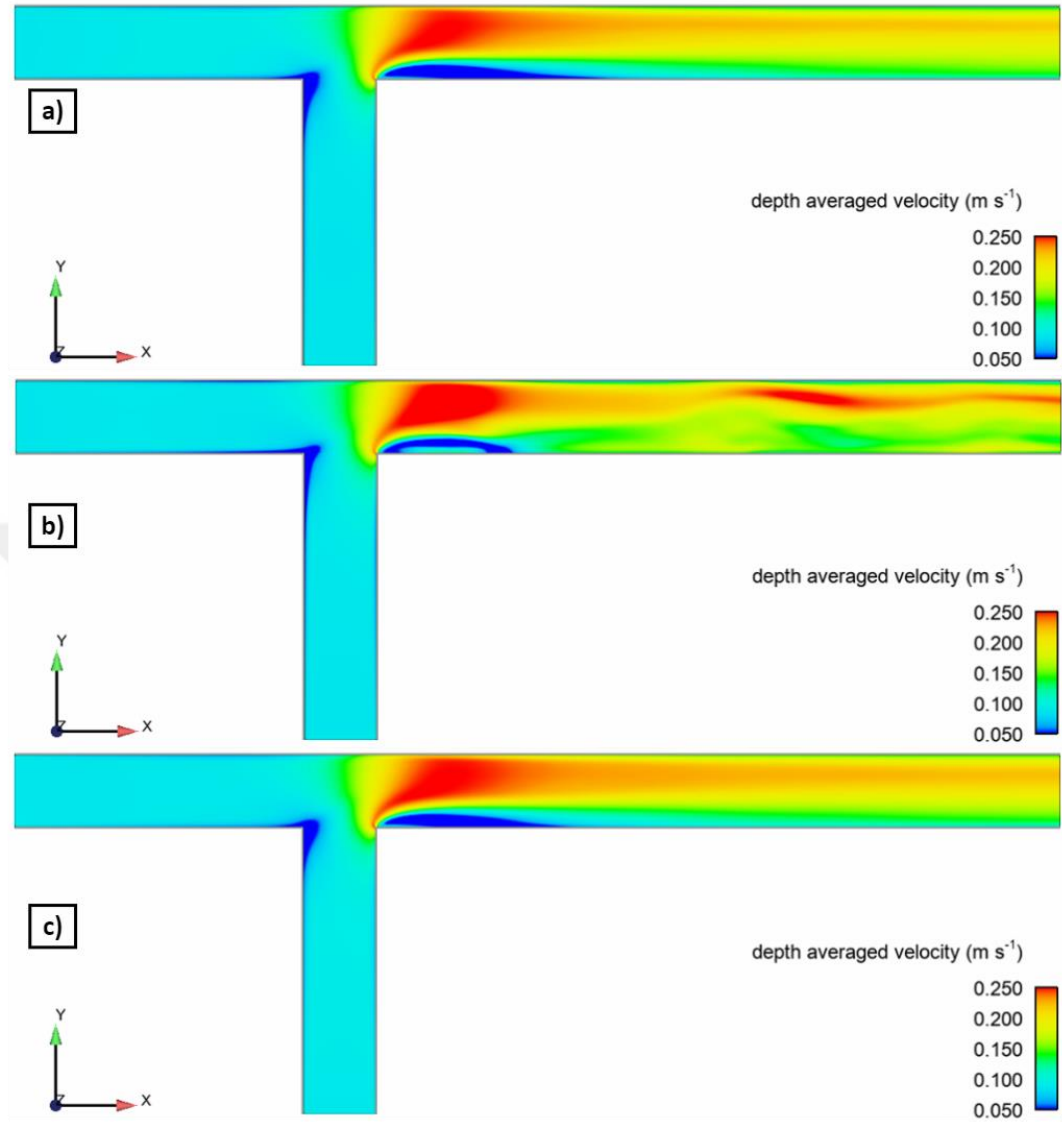


Figure 5.10. Depth averaged velocities for 90° river confluence in plan view by three turbulence models: a) *standard $k - \varepsilon$* ; b) *$k - \omega$* ; c) *RNG $k - \varepsilon$* .

The simulations were performed with subcritical flow states for the main and lateral channels. The Froude Numbers of confluence flows are shown in Figure 5.11, where *standard $k - \varepsilon$* and *RNG $k - \varepsilon$* models give similar results compared with the *$k - \omega$* model. It can be seen that *standard $k - \varepsilon$* and *RNG $k - \varepsilon$* behave the same way for these variables for surface velocities, depth-averaged velocities, and Froude numbers. Another comparison was made by contouring the total hydraulic heads of

these model results. In Figure 5.12, the total hydraulic head results of these models are shown.

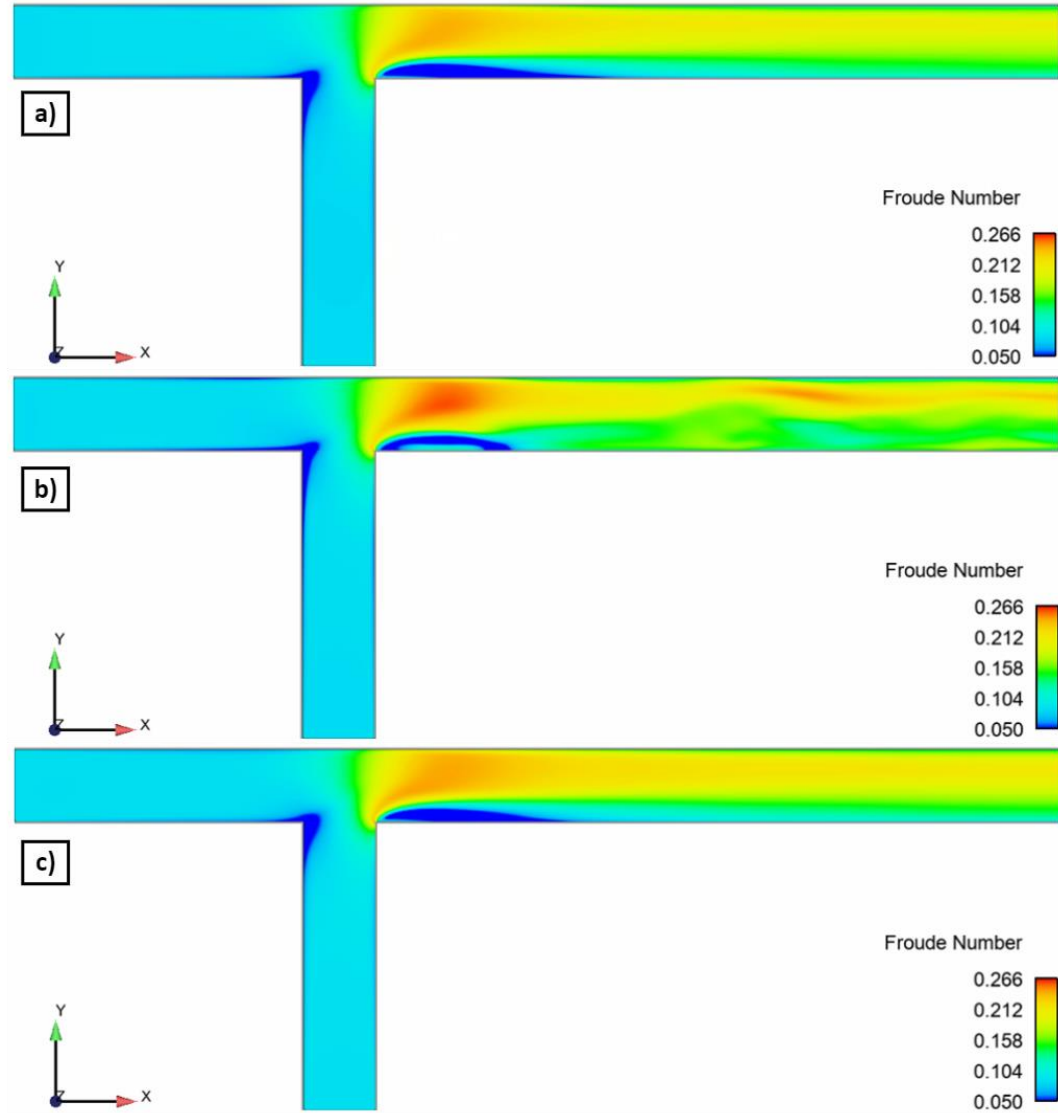


Figure 5.11. Froude Numbers in plan view near a 90° river confluence by three turbulence models: a) *standard $k - \varepsilon$* ; b) *$k - w$* ; c) *RNG $k - \varepsilon$* .

The legend scale in Figure 5.12 was created for a small range. The reason for this was the total hydraulic head was sensitive; the changes in the entire head were nearly negligible since the channels were created with zero slope and the inlet velocities were identical.

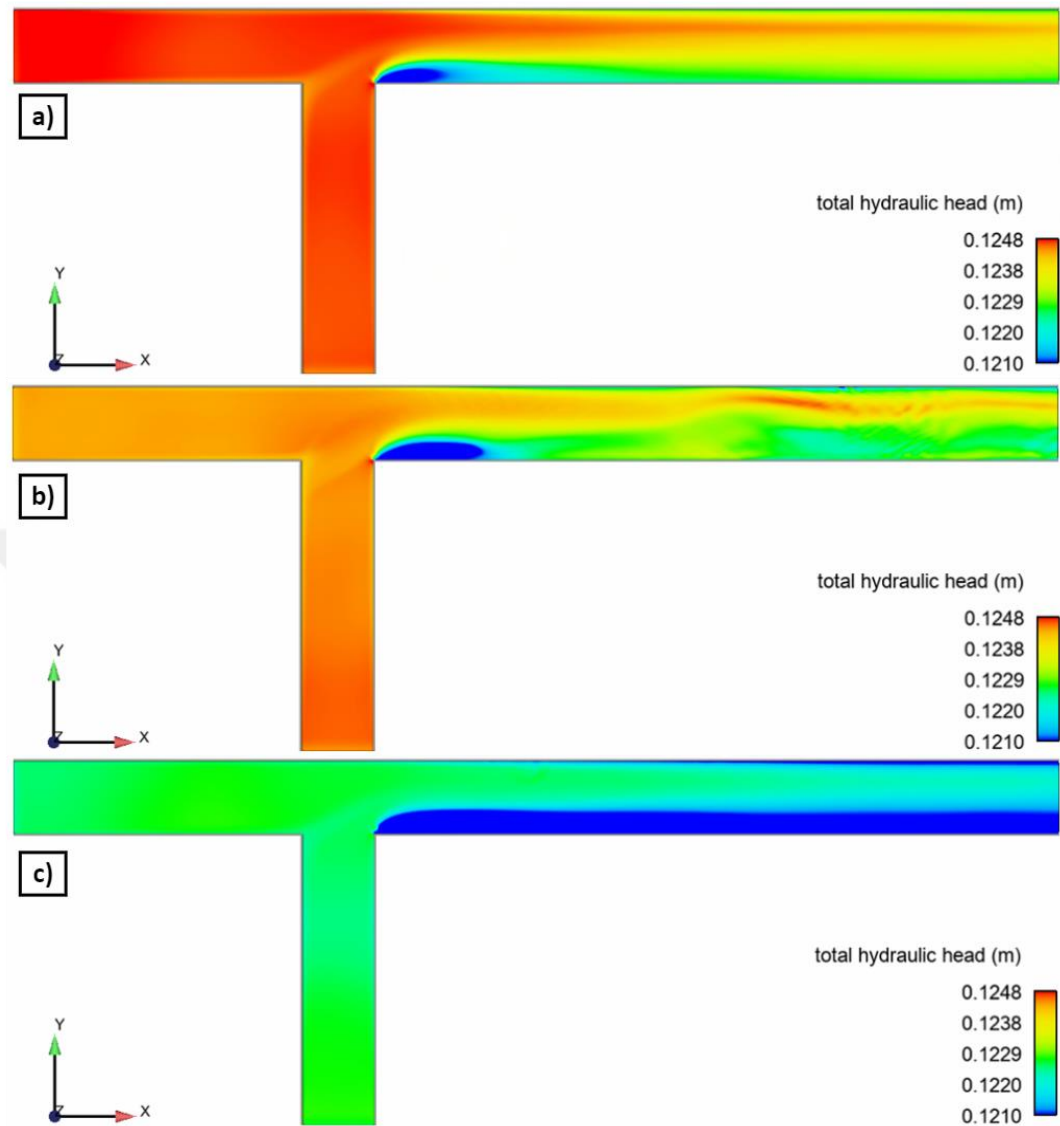


Figure 5.12. Hydraulic energy in plan view for 90° river confluence by three turbulence models: a) *standard $k - \varepsilon$* ; b) *$k - w$* ; c) *RNG $k - \varepsilon$* .

Three different turbulence model was investigated and discussed in this section. *standard $k - \varepsilon$* model was chosen considering the computational time required to achieve an equilibrium-state solution and the capability to capture the dynamic nature of the flow in confluent channels. Next, the analyses regarding the confluence angle will be performed, and their results will be presented with the *standard $k - \varepsilon$* turbulence model and 10 mm mesh sizes since this model uses less computational time and represents the flow characteristics compatible with the literature.

5.4 Confluence Angles

Rice et al. (2008) state that research has been limited to natural confluences, especially when considering angled junctions. The confluence angle and discharge ratio were the primary factors influencing the zones created near the confluence region (Rice et al., 2008). Best and Reid (1984) demonstrated the impact of these variables on the extent of the separation zone, and they concluded that as the confluence angle or discharge ratio increases, the separation zone expands and progressively becomes more influential in shaping the confluence dynamics.

In this section, the effect of confluence angle on flow dynamics will be investigated numerically, using the selected turbulence model, *standard $k - \varepsilon$* , and mesh size, 10 millimeters, since it is not possible and practical to explore all the possible confluence angles, three selected angles will be examined, which are 90°, 60°, and 30°. The dimensions of the models were introduced in Section 5.1, Model Setup, where these three angled junctions can be seen. This section will present the difference in flow dynamics introduced by confluence angles by analyzing different model parameters. The zones generated due to the confluent flows will be discussed and compared to the findings of the publications in the literature. Considering the incoming velocities from the inlet boundaries, the velocity components were selected as 0.1 m/s with an initial water depth of 0.10 meters for the main and lateral channels. The flows in the channels had subcritical flow states. The effect of confluence angle on flow dynamics will be investigated by plan-view, cross-section-view, and point location (probe) analyses.

5.4.1 Plan-View Analyses

Figure 5.13 illustrates the turbulent kinetic energy in plan view for each 90°, 60°, and 30° channel confluence. One may see that as the confluence angle decreases, the turbulent kinetic energy decreases. In Figure 5.14-a, the incoming flows coincide with a 90° angle, resulting in a large separation zone. In Figure 5.13-b In Figure

5.13-b, this separation was lower for a 60° confluence angle, and the turbulent kinetic energy through the downstream part is distributed in the channel width. In Figure 5.13-c, for a 30° confluence angle, the turbulent kinetic energy can only be seen in the downstream reaches near-wall regions.

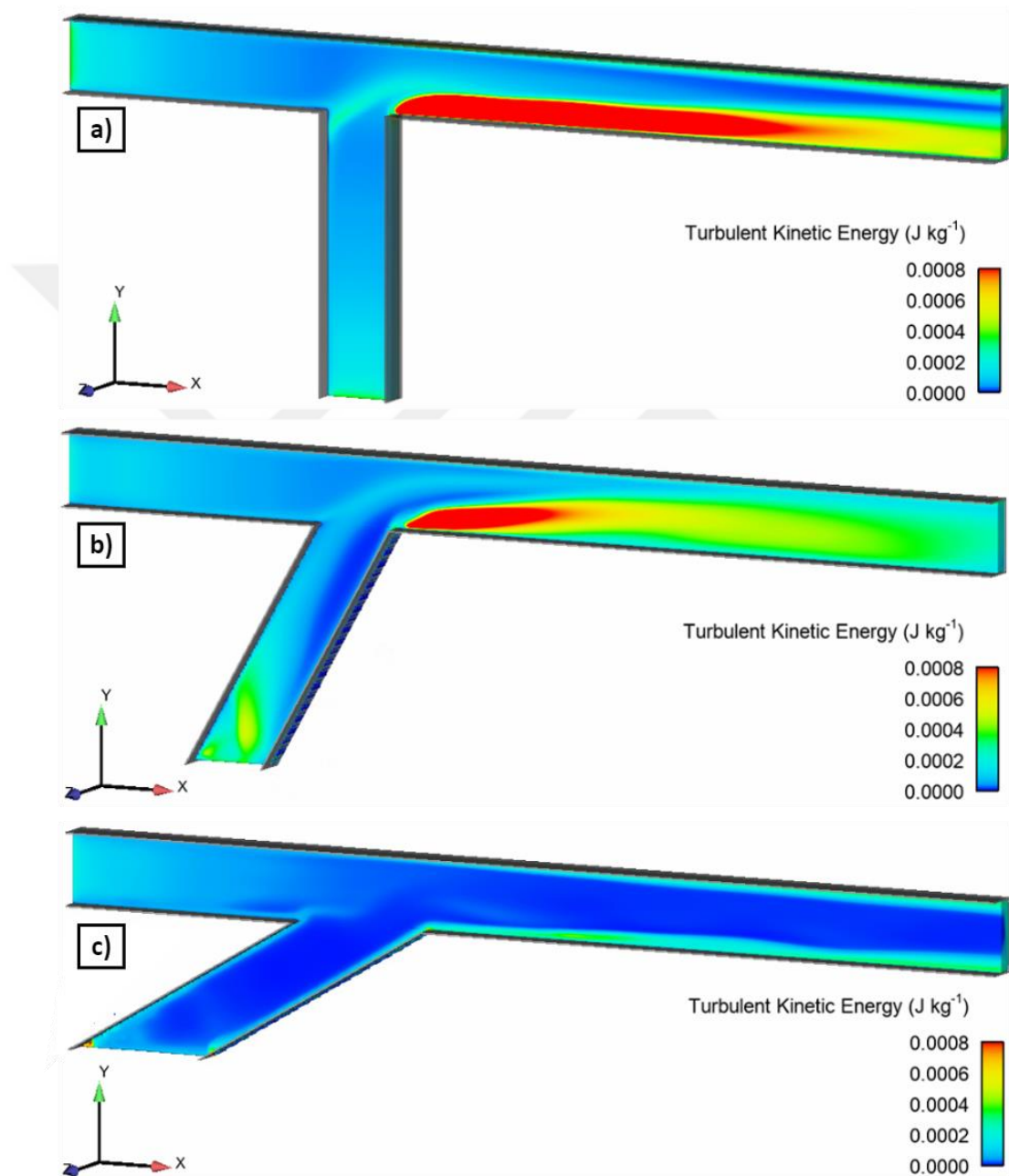


Figure 5.13. The turbulent kinetic energy in plan view *standard* $k - \varepsilon$ model for a) 90°, b) 60°, and c) 30° confluences.

The TKE has its maximum value distant from the confluence area. The thickness, as well as the width of the vortex, differ in three cases.

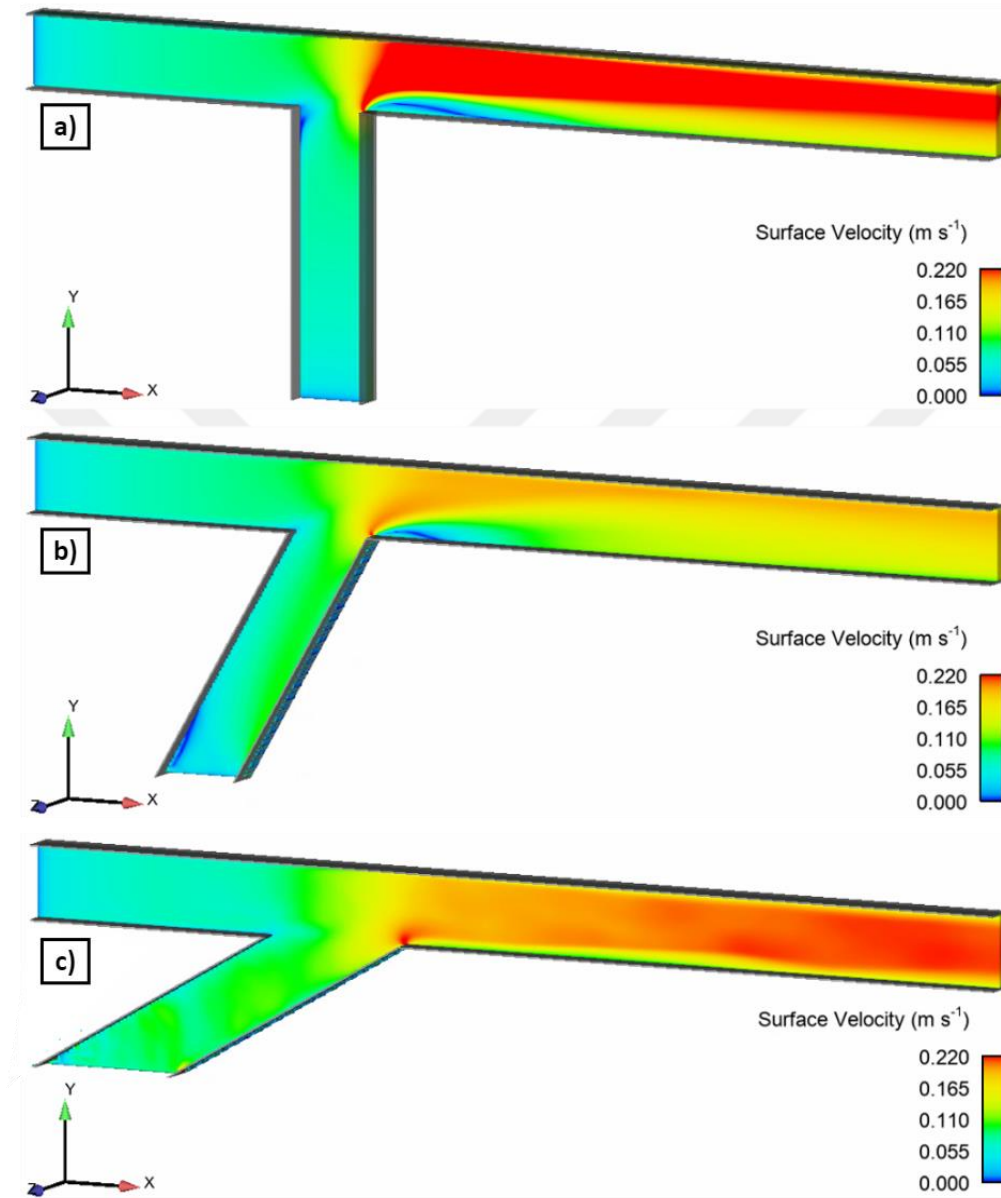


Figure 5.14. Surface Velocity in plan view for *standard* $k - \varepsilon$ model for: a) 90°, b) 60°, and c) 30° confluences.

In Figure 5.14, the surface velocities for the three cases are presented. When the legend is kept the same, the higher surface velocity is observed in a 90° channel confluence across the wall and progresses in that region. Due to the vortex formation

near the junction, the velocity magnitude falls close to 0 m/s as a thin layer in that region. One may observe the velocity behavior in the 30° channel confluence in that the velocity magnitude spreads through the channel width showing an organized motion.

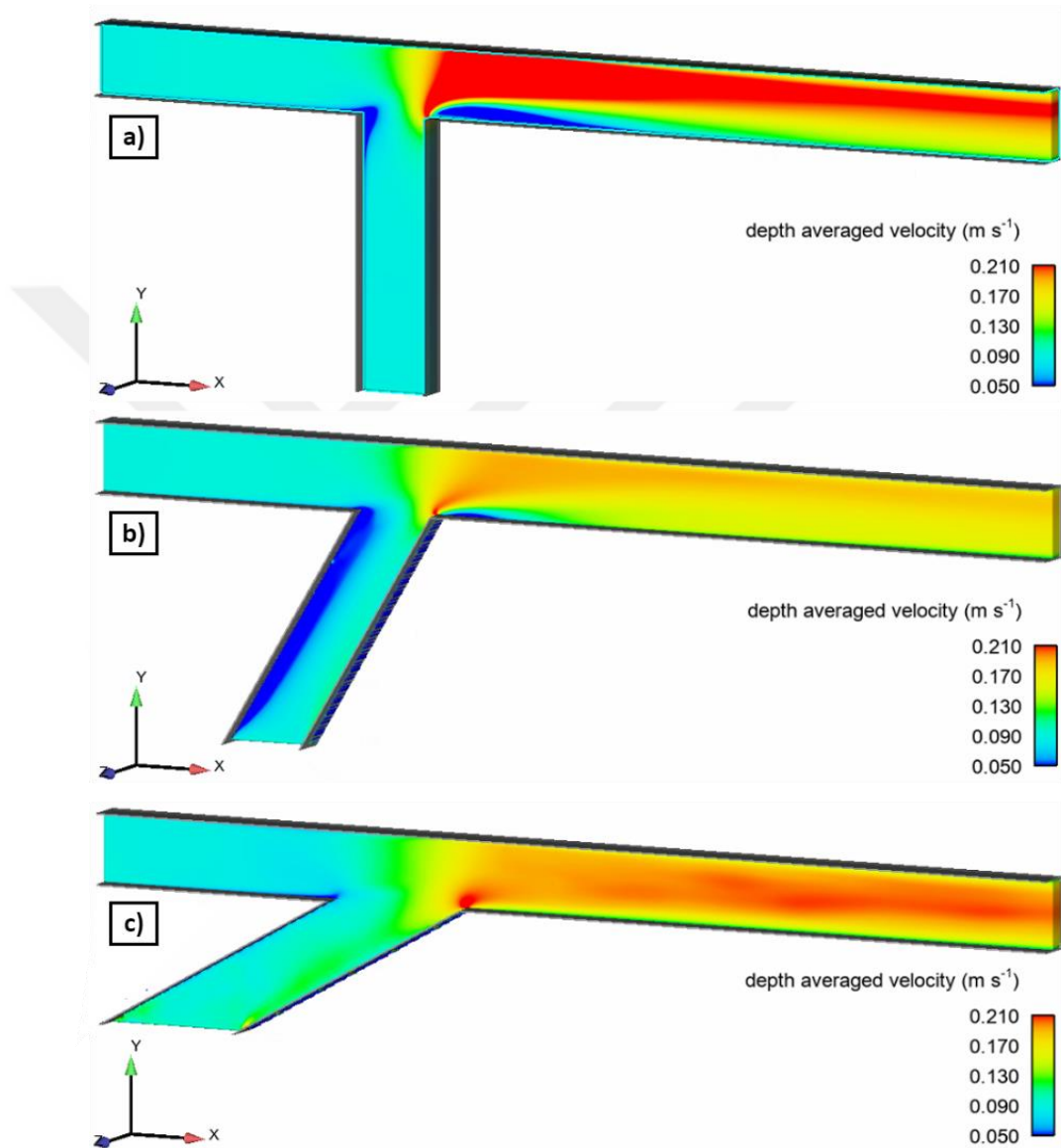


Figure 5.15. Depth average velocity in plan view *standard k – ε* model for: a) 90°; b) 60°; c) 30°confluences.

Depth-averaged velocities are presented in Figure 5.15, where the difference can be observed. The shear layer for the 90° confluence angle, as shown in Figure 5.15-a, can

be seen clearly where the depth-averaged velocities differ near the outer and inner wall layers near the junction. The zones mentioned in the literature part are visible in 90° confluence. One may see that these results are consistent with the literature. For 60°; and 30°confluences, in Figure 5.15-b-c, the shear layer visible in the 90° junction faded as the angle decreased.

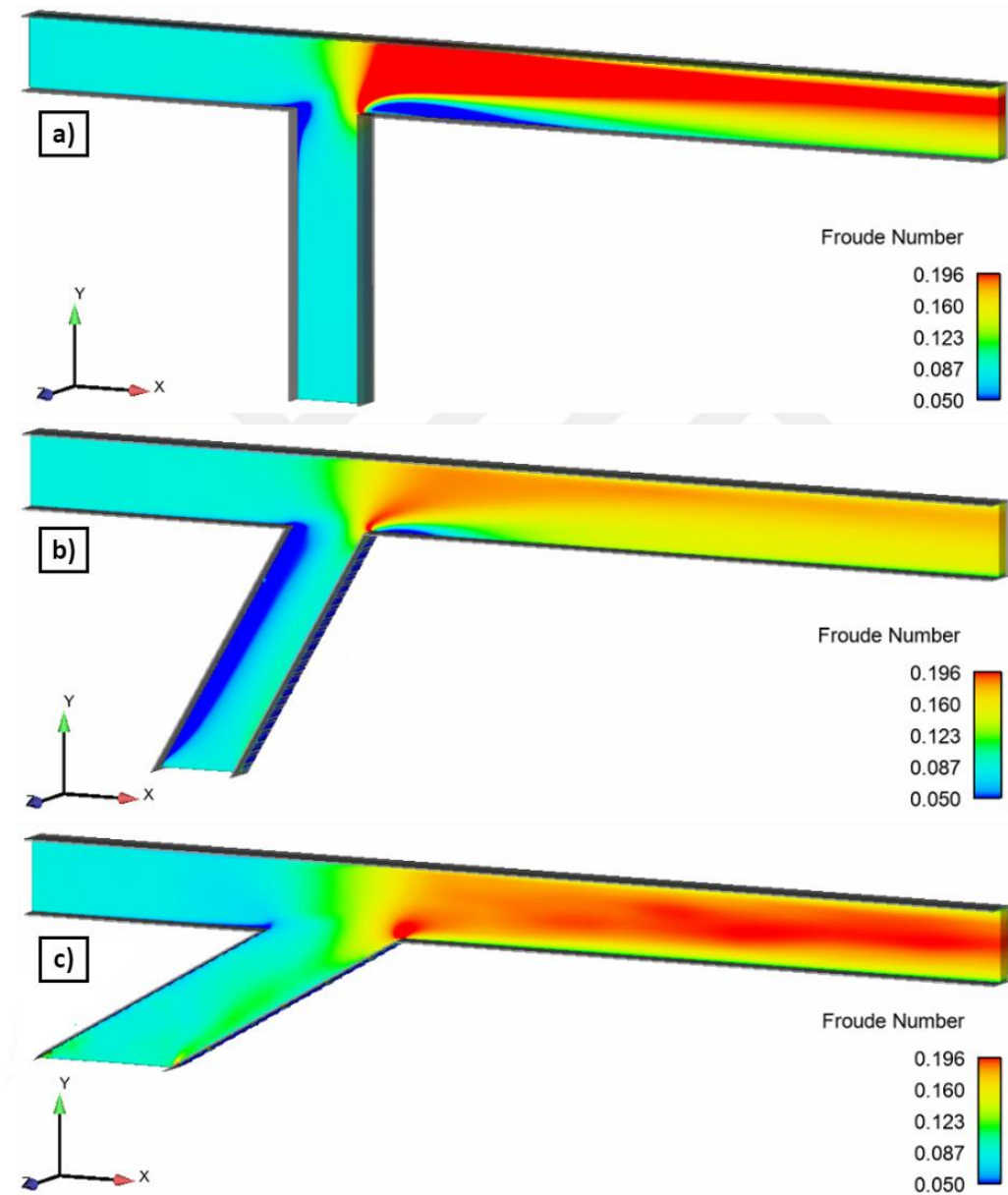


Figure 5.16. Froude Number in plan view for *standard* $k - \varepsilon$ model for: a) 90°, b) 60°, and c) 30° confluences.

Due to this decrease, the flow behavior after the confluence tends to be uniform through the downstream of the channels. The separation and stagnation zones become smaller with the decreasing angle. The recirculation zone generated just downstream of the confluence differs for each case. As Wang and Cheng (2000) stated, neither the recirculation length nor width is solely dependent on the momentum flux ratio, and correlation formulas for recirculation length and width with velocity and width ratios were proposed based on the numerical results. With this in mind, the simulations were performed with a specific inlet velocity, and the only changing parameter was the confluence angle with this model and section.

The plan view figures were generated for Froude Numbers for each angle where the results were consistent with the other hydraulic properties. In Figure 5.16, the Froude Number changes due to the confluence can be seen. These results may be considered nearly identical in depth average velocity cases. The flow behavior after the confluence for different confluence angles was investigated. Until here, the presented figures were generated for various parameters representing the flow behavior from the inlets through the outlet in plan views. One may see the general behavior from these figures. Another representation will be discussed for the cross sections placed in the main channel, majorly through the downstream part of the confluence.

5.4.2 Cross-Section-View Analyses

Another comparison was made with generated cross sections near the confluence portion of the main channel for 90°, 60°, and 30° confluent channels. They were placed 0.20 meters (half of the channel width) apart on the main channel before and after the confluences. Since only the plan views of different hydraulic parameters were presented until now, cross-sectional views may serve to better understand the flow behavior before and after the confluence for the main channel. This aim generated cross-sectional views of turbulent kinetic energies and secondary velocities. The difference introduced by the confluence is more precise with channel cross-sections. For the generated figures, the flow directions were indicated. The

cross-section view for the turbulence kinetic energy is presented in Figure 5.17, where the effect is higher in 90° confluence. Also, the location of turbulence (or recirculation zone) moves in the flow direction as the angle decreases. Figure 5.18 shows velocity magnitude in a cross-sectional view for the three cases with the corresponding flow direction. One may notice that the shear layer is visible in a 90° confluence.

Figure 5.19 shows the secondary (y-direction, which is perpendicular to the main flow direction in the horizontal plane) velocity magnitudes introduced by the lateral channel for different angles of confluences. With the increasing confluence angle, secondary velocity is a dominant feature in the downstream part of the confluence since they are apparent after the confluence in the recirculation region. The following statements may be listed by looking at the cross-sectional views of the confluent channels.

- 1- Turbulent kinetic energy dominates near the recirculation zone, primarily visible on the 90° confluent channels.
- 2- The turbulent kinetic energy decreases as the angle between the primary and lateral channels decreases.
- 3- The behavior of flow velocities at the confluence's downstream part is similar to the turbulence kinetic energy.
- 4- As the confluence angle decreases, the recirculation zone width and length decrease, as Wang and Cheng (2000) suggested.
- 5- Secondary velocity magnitudes increase as the confluence angle increases from 30° to 90° .

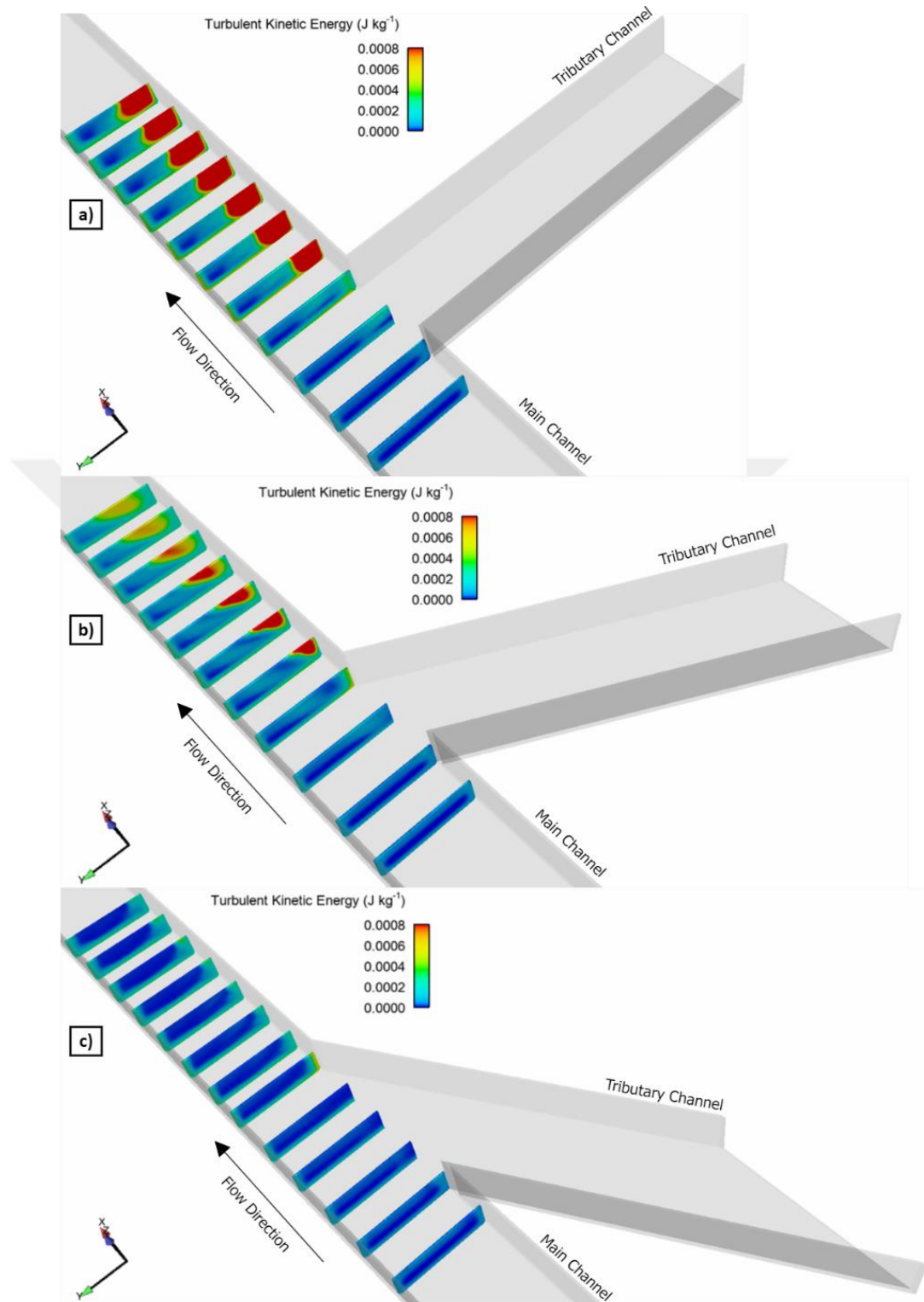


Figure 5.17. The turbulent kinetic energies in the cross-sectional view for a) 90°, b) 60°, and c) 30° confluence.

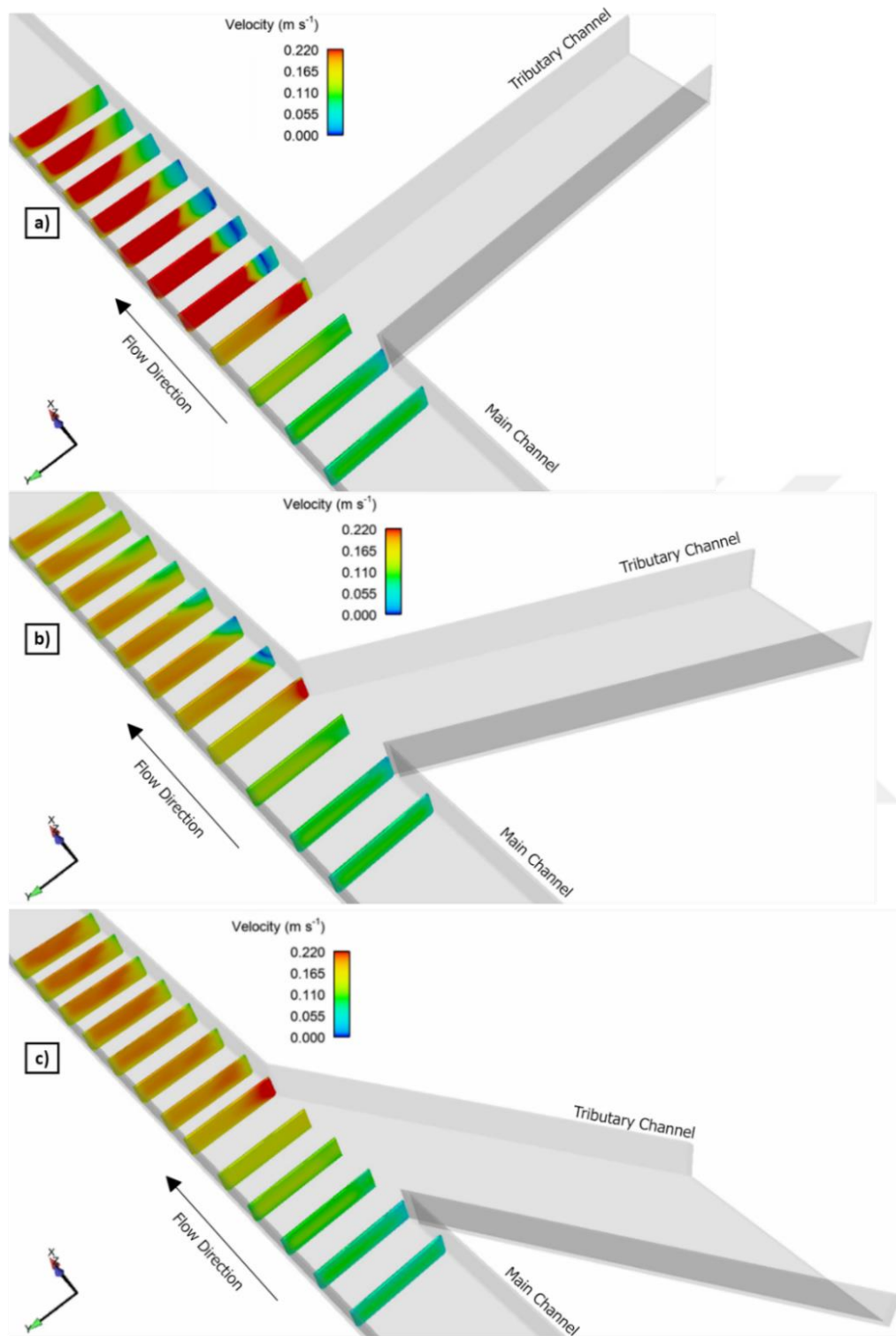


Figure 5.18. Velocities in cross-sectional view for a) 90°, b) 60°, and c) 30° confluence.

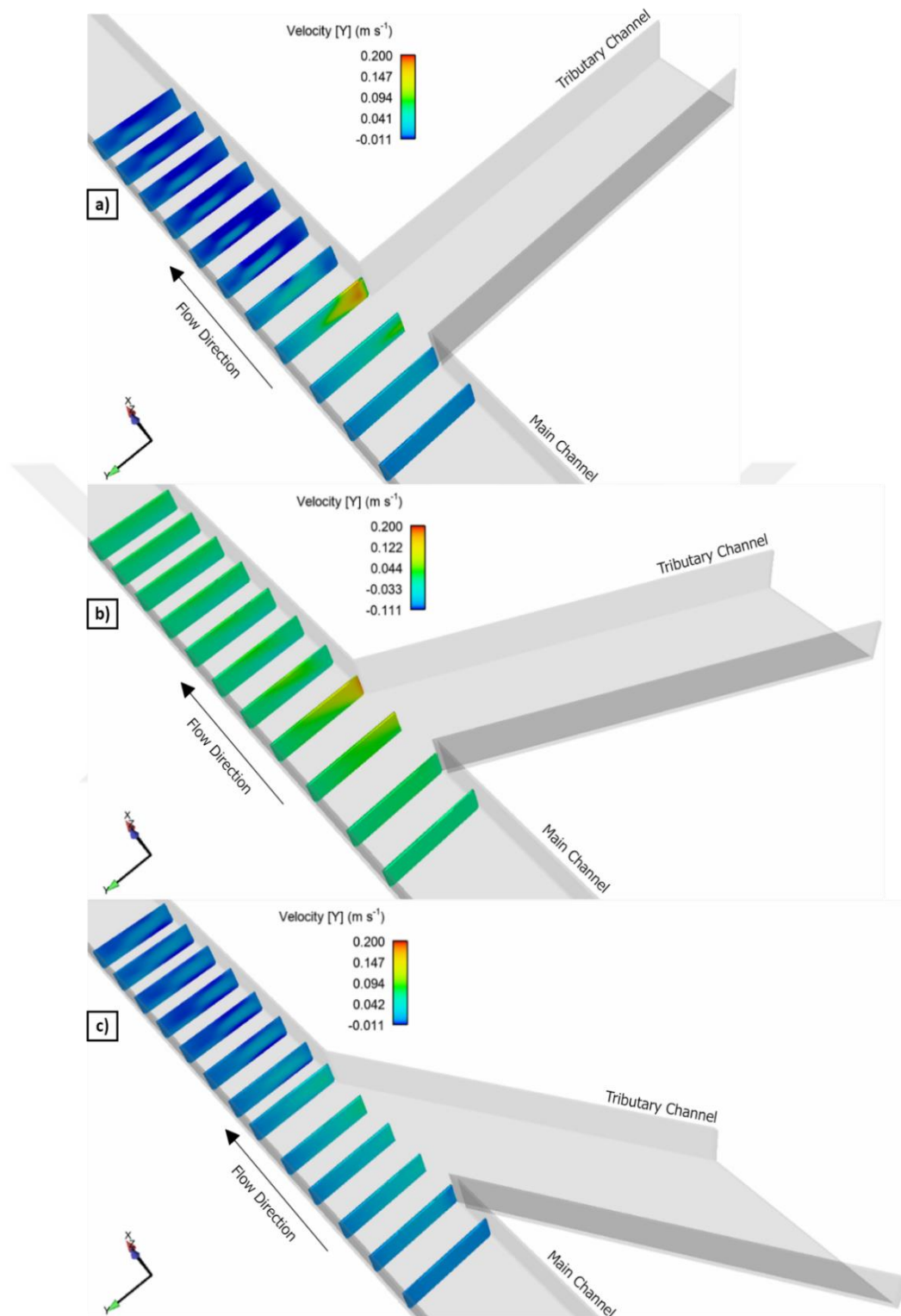


Figure 5.19. Secondary velocities in cross-sectional view for a) 90°, b) 60°, and c) 30° confluence.

5.4.3 Point Location (Probe) Analyses

Flow-3D allows users to generate point value graphs by adding ‘probes’ in the computational domain. The information on each point in the study domain can be obtained quantitatively by adding probes to the desired location in the simulation's preprocessing and postprocessing. Eight probes were used in this section to generate the model results for 90° , 60° , and 30° channels. Figure 5.20 shows the representative schematic for probe locations in the cases where $\theta = 90^\circ, 60^\circ$ and 30° . Q_m and Q_t are the main and tributary discharges, while V_m and V_t are the main and tributary velocities, respectively. Probes were placed on the lines of $y = width/10$ and $y = width/2$ to capture the confluence effect on the flow behavior near this region. Here, width represents the width of the channel, noting that main and tributary channel widths are equal. The distances between probes 1, 2, and 3 located on $y = width/10$, and 5, 6, 7 located on $y = width/2$ in the x-direction was defined as $width/2$. The results of points on the lines of $y = width/10$ and $y = width/2$ were separately investigated. The generated graphs will be presented for velocities, Froude numbers, flow depths, total hydraulic heads, and turbulence kinetic energies.

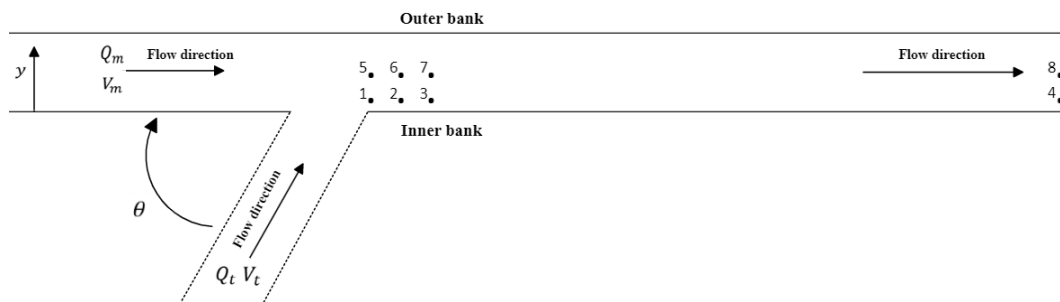


Figure 5.20. Representative confluent channel with probe locations of 8 probes placed where $y = Width/10$, and $y = Width/2$.

In Figure 5.21, the velocity change on these eight points is shown for $\theta = 90^\circ, 60^\circ$, and 30° as a time series. At $y = Width/10$ for points 1-4, it was seen that wall effects are dominant and consistent with the literature where the recirculation zone was mentioned in the literature part. One may check Figure 5.14 for the

representation of velocity distribution in the plan view. The recirculation zone generated due to the confluence affects the hydraulic properties — probes 2 and 3 stay in the recirculation zone where their velocity values are lower than the probe 1 and 4 values for 90° confluence. Probes 1 and 4 have higher velocities than probes 2 and 3 because of the shear layer and recirculation zone. For probe 1, the lateral channel effect is smaller than for probes 2 and 3 due to the incoming main channel flow. Probe 4 is located downstream of the channel, where the velocity distribution is lowered compared to the probe 1 velocity value. For the probes located in the middle of the channel (5, 6, 7, and 8), velocity distributions are close to each other due to the location of these probes.

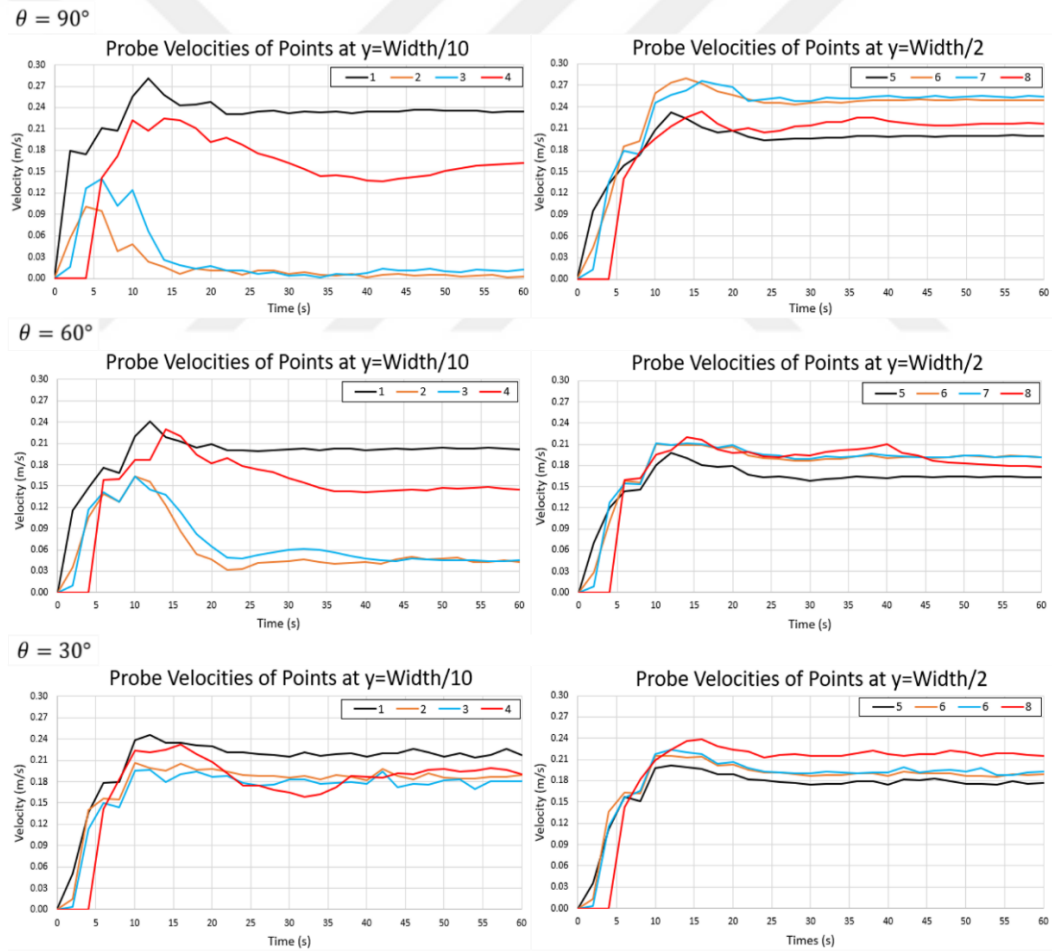


Figure 5.21. Velocities at points near the inner wall and in the middle of the confluence downstream at half of the water depth a) for 90° confluence, b) for 60° confluence, and c) for 30° confluence.

Table 5.2. Velocities of the selected points.

*Velocities of the points located near the wall ($y=width/10$),
and middle of the channel ($y=width/2$)*

	1	2	3	4	5	6	7	8
$\theta=90$	0.235	0.002	0.013	0.162	0.199	0.249	0.254	0.217
$\theta=60$	0.202	0.043	0.045	0.145	0.164	0.191	0.191	0.178
$\theta=30$	0.218	0.189	0.180	0.190	0.177	0.189	0.193	0.215
	<i>difference % of velocities</i>							
<i>btw $\theta=90$ and $\theta=60$</i>	14.2%	-1639.8%	-253.6%	10.3%	17.8%	23.1%	24.8%	17.9%
<i>btw $\theta=90$ and $\theta=30$</i>	7.4%	-7600.0%	-1321.0%	-17.5%	11.3%	24.0%	23.9%	0.7%

Near the inner wall downstream of the confluence, the flow behavior is less coherent when compared to the locations away from the wall. While comparing points 4 and 8, the difference in the velocity values may indicate that the wall is still effective. For $\theta = 60^\circ$, similar behavior can be seen between the points at $y = Width/10$ and $y = Width/2$, where the wall effects are evident at the points near the inner bank. In each graph, higher fluctuations were the cause of flow confluence. However, the angle effect is visible for points 5, 6, 7, and 8 between $\theta = 90^\circ$ and $\theta = 60^\circ$ where the flow in $\theta = 60^\circ$ confluence is nearly uniform for all points at the middle of the channel. The fluctuations and higher values are apparent in $\theta = 90^\circ$ confluence. The uniformity between the point values can be seen as the angle decreases. In $\theta = 30^\circ$ intersection, the velocity values of the points have a slight difference, nearly uniform, since the intersection effect decreases as the angle between the main and tributary channel decreases. Points 2 and 3 have lower velocity values for higher confluence angles.

In the $\theta = 30^\circ$ confluence, the recirculation zone is smaller in width, which allows the incoming flows to accelerate each other rather than creating a stagnation layer. As mentioned in section 5.4, there is a layer formed smaller in width and more significant in length where the turbulence is effective near the inner bank. Due to this turbulence, the downstream flow after the confluence has a fluctuating profile near the internal bank. The shear layer effects are minimal for this case, even near the inner bank.

The simulations were accepted to reach an equilibrium state in 60 seconds since the fluctuations in the point values were small. Point velocities where $t = 60 \text{ seconds}$ were selected to compare the angle effect on velocities. This procedure will be repeated for other hydraulic parameters. The findings are presented in Table 5.2 for a quantitative comparison. The highest and lowest velocity values were seen in $\theta = 90^\circ$ confluent channels. Point 2 has the lowest velocity value since it is located at the recirculation zone. The highest velocity was seen at point 7 due to the accelerating effect of the main channel flow on the flow coming from the tributary channel. In

the 60° confluence case, the highest velocity was observed at point 1 after 60 seconds, while the recirculation zone effects point two again, resulting in the lowest velocity value. Respectively constant values are seen for 30° confluence.

The difference in the values as % can also be seen in Table 5.2. Between $\theta = 90^\circ$ and $\theta = 60^\circ$, at points 1 and 5, velocity values are higher in 90° confluence for 14.2% and 17.8%, respectively. At points 2 and 3, there is a dramatic increase (minus sign represents the increase) in velocity values as the angle decreases. It is due to a reduction in the width of the recirculation zone at 60° confluence. This increase is more visible in the 30° confluence for points 2 and 3. Near the inner bank further downstream, the increase in the velocity for the 30° junction is seen — point 4. The difference in velocity between the 90° and 30° confluence, in the line of $y = Width/2$, for the points of 5, 6, 7, and 8, is similar compared to the 60° confluence case. The only difference in flow behavior was observed in point 4, near the inner bank further downstream. The reason may be due to the flow uniformity since there is a significant decrease in confluence.

In Figure 5.22, the same procedure was repeated. The difference between 90°, 60°, and 30° confluence was investigated for Froude numbers at the defined points. Similar behavior was observed since the Froude number is directly related to the velocity, V of the flow. With the same initial depth, D , and gravitational acceleration, g , velocity is the critical determinant for Froude numbers. The Froude number is a dimensionless quantity utilized in the field of open channel hydraulics that serves as a measure of the ratio between the actual velocity of flow, averaged across the cross-section, V , and the speed at which shallow waves propagate, celerity, c (Hager & Castro-Orgaz, 2017). In Equation 5.1, this ratio is shown.

$$Fr = V/\sqrt{c} \quad 5.1$$

Equation 5.2 shows the relationship that formed celerity where g is the gravitational acceleration, and D is the hydraulic depth.

$$c = (gD)^{1/2} \quad 5.1$$

In the case of V equals to $(gD)^{1/2}$, the Froude number is unity, and the critical flow state occurs. If the Froude number is greater than unity, the flow is in the supercritical state where the inertial forces are dominant; in contrast, under the condition of the Froude number smaller than the unity, the flow is in the subcritical state where the gravity forces are dominant (Chow, V. T., 2009).

In 90° confluence, the recirculation zone effect is apparent again. As the angle decreases, this effect shows respective altery and the points located near the inner bank and middle of the channel have similar Froude number values, as seen in Figure 5.22.

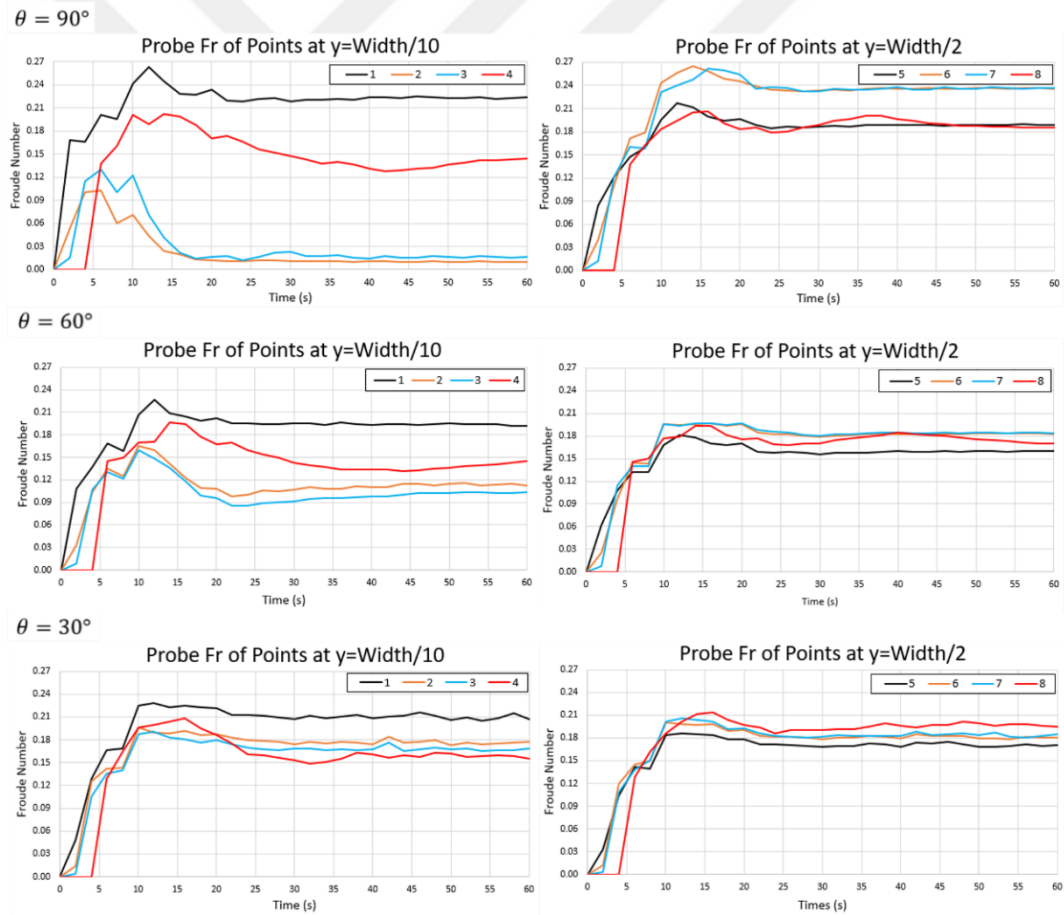


Figure 5.22. Froude Numbers near the inner wall and in the middle of the confluence downstream at half the water depth a) for 90° confluence, b) for 60° confluence, c) for 30° confluence.

Table 5.3. Froude Numbers of the selected points

*Froude Numbers of the Points Located Near the Wall ($y=width/10$),
and Middle of the Channel ($y=width/2$)*

	1	2	3	4	5	6	7	8
$\theta=90$	0.22388	0.01034	0.01651	0.14397	0.18873	0.2357	0.2366	0.18587
$\theta=60$	0.19203	0.11301	0.10349	0.14488	0.15958	0.18249	0.18336	0.16969
$\theta=30$	0.20714	0.17681	0.16809	0.15479	0.17036	0.18079	0.18458	0.19498
	<i>difference % of Froude Numbers</i>							
<i>btw $\theta=90$ and $\theta=60$</i>	14.2%	-992.7%	-526.7%	-0.6%	15.4%	22.6%	22.5%	8.7%
<i>btw $\theta=90$ and $\theta=30$</i>	7.5%	-1609.7%	-917.9%	-7.5%	9.7%	23.3%	22.0%	-4.9%

The Froude numbers of the initial flows were defined as $Fr = 0.101$ since the main and tributary channels are confluence into a smaller scale model. As the angle decreases, the gap in Froude number values for points 1, 2, 3, and 4 closes, and similar characteristics are seen near the wall region. The same behavior is visible for the points in the middle of the channel. However, the variance is not dominant as in the near wall region. For 60° and 30° confluences, the points in the middle of the channel have similar Froude values, as the fluctuations in Froude numbers are small.

At the points located near the inner wall, the effect introduced by the wall is evident since the values of points 1, 2, 3, and 4 differ. In Table 5.3, the behavior change for Froude values is similar to the velocity values. The only difference may be seen while looking at the difference percentage values between $\theta = 90^\circ$ and $\theta = 30^\circ$. This difference may be due to the flow uniformity after the confluence in $\theta = 30^\circ$ case. In a plan view, one may check Figure 5.16, where the Froude number variance is apparent in the downstream part of the main channel after the junction.

The flow depths at the selected points are presented in Figure 5.23 to time, as in the above cases. The depth profiles in 90° and 60° confluences behave uniformly after some time — Figure 5.23. However, the oscillatory depth profile is apparent in 30° confluence; even a more extended duration was considered. Because of that, by accepting this behavior, the graphs were generated for 60 seconds of simulation time, compatible with other cases. As can be seen from the graphs, the flow depth of the selected points increases due to the effect of the confluence. This increase is sharper at points 2 and 3 for 90° confluence. The intense collision of two flows in a 90° channel confluence may be the reason for it. As seen, the depth decreases afterward, which is the cause of the flow separation zone.

Table 5.4 presents the flow depth values of the selected eight points at $t = 60$ seconds for 90° , 60° , and 30° confluences. By comparing the depth values of points 1 and 5, it was seen that the incoming flow effect is comparably higher at the middle of the channel near the confluence for all cases by looking at the difference between them for each angle.

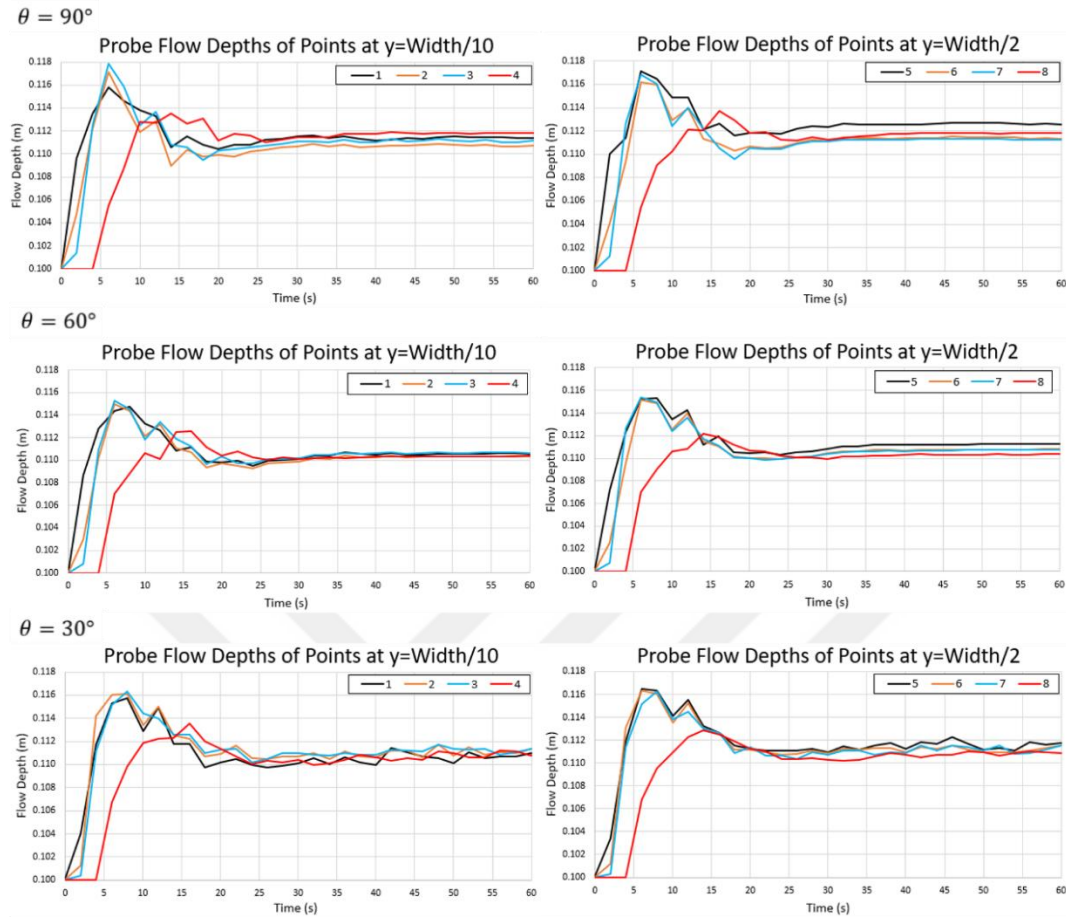


Figure 5.23. Flow depths near the inner wall and in the middle of the confluence downstream a) for 90° confluence, b) for 60° confluence, c) for 30° confluence.

When the depth values of points 2 and 6, 3 and 7, and 4 and 8, were compared, slightly higher depths can be seen in the middle of the channel for each case. Also, as the angle decreases, the difference between these values decreases, as shown in Table 5.4. When comparing the results for 90° and 60° confluences, flow depths at the 90° confluence are higher at each selected location. However, compared to the 30° confluence, depth values are slightly lower in the 90° confluence for points 2, 3, 6, and 7. In 30° confluence, the flow in the main channel behaves uniformly. This may be why these points have slightly low flow depth values. Also, the variance between the depth values of points 2 and 6 is higher while looking at the difference percentage of 90° and 30° confluences; even these points stay in the same vertical line.

Table 5.4 Flow depths of the selected points.

*Flow Depths of the Points Located Near the Wall ($y=width/10$),
and Middle of the Channel ($y=width/2$)*

	1	2	3	4	5	6	7	8
$\theta=90$	0.1114	0.1107	0.1111	0.1118	0.1126	0.1113	0.1112	0.1118
$\theta=60$	0.1106	0.1103	0.1106	0.1104	0.1113	0.1108	0.1107	0.1104
$\theta=30$	0.1110	0.1114	0.1113	0.1108	0.1118	0.1115	0.1115	0.1109
	<i>difference % of flow depths</i>							
<i>btw $\theta=90$ and $\theta=60$</i>	0.7%	0.4%	0.5%	1.3%	1.1%	0.5%	0.4%	1.3%
<i>btw $\theta=90$ and $\theta=30$</i>	0.4%	-0.6%	-0.2%	0.9%	0.7%	-0.2%	-0.3%	0.8%

It may be noted that since point 2 is located near the wall, with a 90° confluence angle, the flow behavior near the confluence may be intense compared to the 30° confluence. In 30° confluence, due to the comparatively uniform flow behavior after the junction, the flow depth values in these points show higher values.

Total hydraulic heads for the selected points were investigated for each confluence angle, 90° , 60° , and 30° . As in the flow depth case, for 90° confluence, at points 2 and 3, a steep increase due to the confluence is apparent. After some time, heads at these points remain lower than the other point head values for 60 seconds of simulation. Since points 2 and 3 stay in the flow separation zone, velocity and flow depth are lower at those locations that introduce a lower hydraulic head corollary.

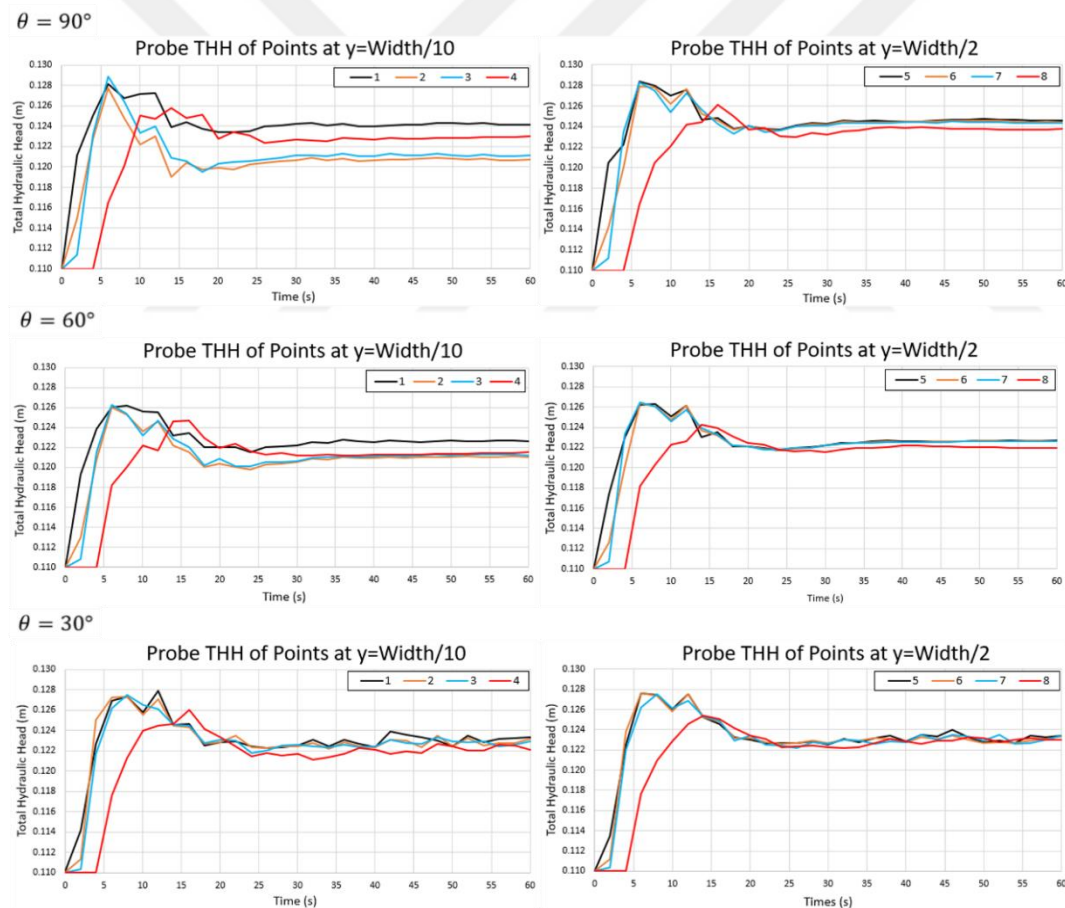


Figure 5.24. Total Hydraulic Heads near the inner wall and in the middle of the confluence downstream at half the water depth a) for 90° confluence, b) for 60° confluence c) for 30° confluence.

Table 5.5. Total hydraulic heads of the selected points.

*Total Hydraulic Heads of the Points Located Near the Wall ($y=width/10$),
and Middle of the Channel ($y=width/2$)*

	1	2	3	4	5	6	7	8
$\theta=90$	0.1242	0.1207	0.1212	0.1230	0.1246	0.1244	0.1243	0.1238
$\theta=60$	0.1226	0.1210	0.1212	0.1215	0.1227	0.1226	0.1226	0.1220
$\theta=30$	0.1233	0.1231	0.1229	0.1221	0.1234	0.1233	0.1234	0.1230
	<i>difference % of total hydraulic heads</i>							
<i>btw $\theta=90$ and $\theta=60$</i>	1.2%	-0.3%	-0.1%	1.2%	1.5%	1.4%	1.4%	1.4%
<i>btw $\theta=90$ and $\theta=30$</i>	0.7%	-2.0%	-1.4%	0.7%	0.9%	0.9%	0.7%	0.6%

For points 5, 6, 7, and 8, the head values are similar when $t = 60 \text{ seconds}$. It may indicate an equilibrium state in the middle of the channel. In the case of 60° confluence, points 2 and 3 have comparatively lower head values than the 90° confluence case. Since the confluence effect is not high compared to the 90° case, this behavior can be expected. Also, when $t = 60 \text{ seconds}$, the total heads of points 2, 3, and 4 have the same value. It may indicate that this behavior is preserved when the flow is in equilibrium on the same horizontal line moving downstream. When the confluence angle is 30° , fluctuating head profile through the simulation period can be seen near the inner wall and in the middle of the channel. The flow separation zone is not formed for 30° confluence, as was apparent in the 90° confluence case.

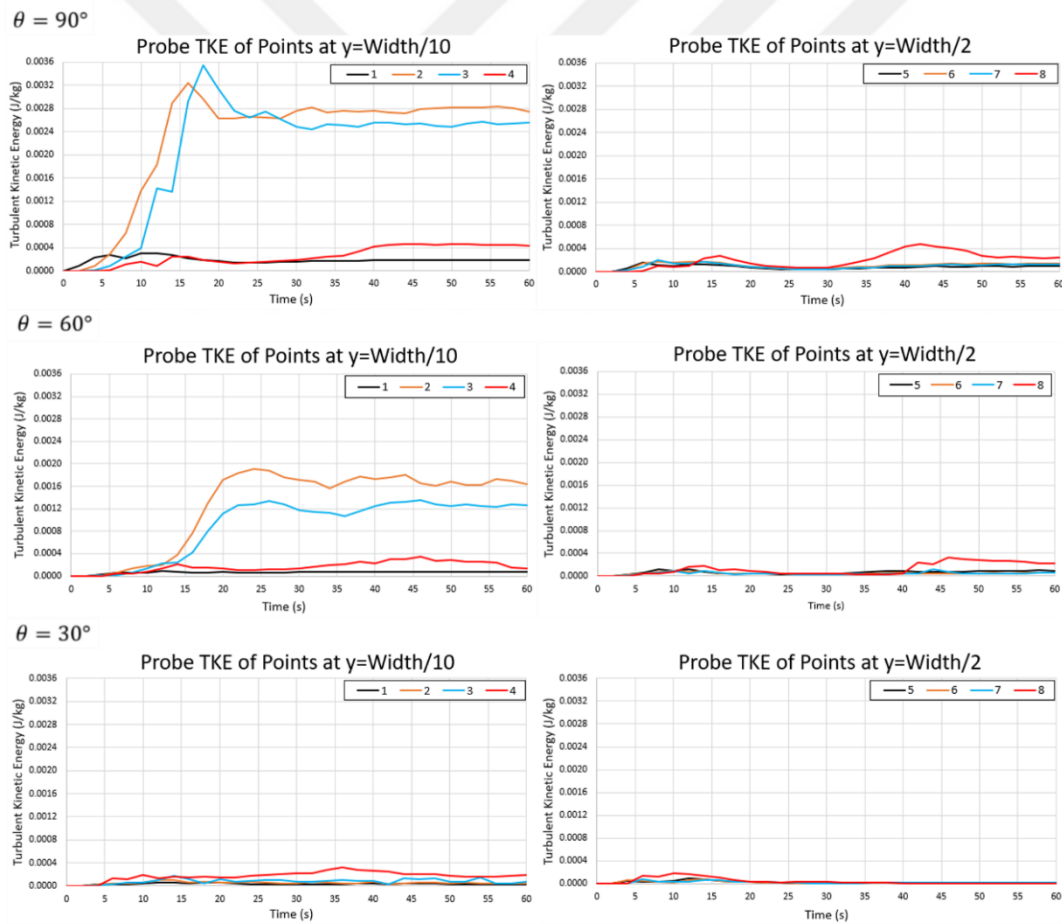


Figure 5.25. Turbulent Kinetic Energies near the inner wall and in the middle of the confluence downstream at half of the water depth a) for 90° confluence, b) for 60° confluence, c) for 30° confluence.

Similar graphs were generated for the selected points' turbulent kinetic energies (TKE) and specified confluence angles — Figure 5.25. While keeping legends the same for all cases, generated turbulence graphs show a strict difference. It was known that the turbulence effect is higher in confluences with more prominent angles. Since only the 90°, 60°, and 30° angled confluences are examined in this study, it will be better to stay in this context. However, for higher confluence angles, one may check for the turbulent flow behavior in the studies of Mosley (1976) and Shakibainia et al. (2010). For 90° confluence, the turbulence effect can be seen for points 2 and 3, while points 1 and 4 have significantly lower TKE values. This is the behavior introduced by the separation or recirculation zone.

To better understand the flow behavior, one may recheck Figure 5.13. As mentioned in Figure 5.14, turbulent kinetic energy is high in the 90° confluence case, and as the angle decreases, the turbulence effect decreases as well after the confluence. That is why the generated graphs are highly different. For the 60° confluence, the turbulence effect can be seen clearly, but the magnitude is lower than the 90° confluence case — Figure 5.13. Turbulence can be seen in the 30° confluence near the inner and outer banks; however, no significant effect accumulates on a specific part of the downstream channel. In this confluence case, turbulence kinetic energy behavior is more likely to be expanded through near the wall regions. It may be helpful to indicate that the separation zone location varies for each confluent channel case. The turbulence effect may not be clear from these graphs since the generated graphs were for the same points selected for each confluent case. However, the general flow behavior of the turbulence kinetic energy can be obtained from these graphs — Figure 5.25.

It can be noted that the turbulent kinetic energies decrease with the decreasing confluence angle, which is consistent with the literature. As seen from Figure 5.25, in 90° confluence, points 2 and 3 have higher values than the other points. It also indicates that this behavior continues even in equilibrium, meaning the flow separation near the inner wall continues through the simulation period.

The turbulence at points 4 and 8 is comparatively more apparent than the other point values. In the 60° confluence, the turbulence effect at points 2 and 3 is higher but lower than in the 90° confluence case. In 30° confluence, turbulence is distributed to the channel width, nearly uniform, slightly higher at the near wall regions. Table 5.7 shows the TKE values of the selected points where $t = 60 \text{ seconds}$ for 90° , 60° , and 30° confluent channels. The TKE values of each point in the 90° confluence have higher values when compared to the point values of 60° and 30° confluences. As the angle difference increases, the difference in the flow behavior introduced by TKE also increases. For all the selected points, the most apparent discrepancy can be observed between the 90° and 30° confluence at points 2 and 3, where the turbulence effects are dominant.

Table 5.6. TKEs of the selected points

*Turbulent Kinetic Energies of the Points Located Near the Wall ($y=width/10$),
and Middle of the Channel ($y=width/2$)*

	1	2	3	4	5	6	7	8
$\theta=90$	0.00019	0.00275	0.00256	0.00043	0.00009	0.00014	0.00013	0.00024
$\theta=60$	0.00007	0.00164	0.00126	0.00013	0.00009	0.00006	0.00005	0.00022
$\theta=30$	0.00003	0.00004	0.00007	0.00018	0.00003	0.00002	0.00002	0.00001
	difference % of velocities							
<i>btw $\theta=90$ and $\theta=60$</i>	62.1%	40.4%	50.7%	68.9%	1.4%	59.3%	58.7%	10.7%
<i>btw $\theta=90$ and $\theta=30$</i>	82.9%	98.4%	97.4%	56.9%	70.6%	85.5%	87.4%	95.7%

5.5 Discharge Ratios

Another factor that affects the flow behavior in confluent channels is the discharge ratio studied by many researchers. Previous research findings have shown that various attributes of an open-channel junction flow, including the shape index of the separation zone, the flow angle of the lateral channel, contraction coefficients, and the location of the stagnation point, strongly rely on the discharge ratio (Zhang et al., 2009). Considering the substantial effect of the discharge ratio on downstream flow and flow zone characteristics in confluent channels, this section will investigate the effects of the discharge ratios. Four discharge ratios will be presented for the flow characteristics, including turbulent kinetic energy, velocity, Froude Number, flow depth, and total hydraulic head. The discharge ratios that will be presented were determined by the following equation, Equation 5.2.

$$q^* = \frac{\text{Lateral flow}}{\text{Combined total flow}} \quad 5.2$$

Until now, three turbulence models and three confluence angles were investigated with the initial velocity values of 0.1 m/s in main and lateral channels where the discharges were $4 \times 10^{-3} \text{ m}^3/\text{s}$, respectively. Now, with different lateral channel velocities, four scenarios will be examined. The following study will be conducted by keeping the main channel discharge value the same for each case. The only difference from the previous simulations will be the lateral velocities directly affecting the discharge values. Also, the simulations performed for this section have a duration of 120 seconds since the difference in discharge values brings a dynamic behavior for the downstream region of the confluence. Even with a longer simulation duration, small oscillations are visible for all investigated flow characteristics. Because of that, the graphs representing the probe points mentioned in section 5.4.1 will be generated for the same simulation duration, which is 60 seconds, since these oscillations can be accepted as negligible. Table 5.7 was prepared to show the investigated four discharge ratios and their related lateral velocities and discharge values in a more compact form.

Table 5.7. Discharge ratios investigated in this study.

<i>Lateral Flow Velocity (m/s)</i>	<i>Discharge (m³/s)</i>	<i>Discharge ratio (q^*)</i>
0.075	0.003	0.429
0.10	0.004	0.500
0.15	0.006	0.600
0.20	0.008	0.667

The generated figures for this section are all in plan view, where only the water surface is visible, as in the turbulence model analysis. The first flow characteristic that will be investigated in this section is velocity. Velocity change introduced by the discharge ratio difference was investigated. Figure 5.26 illustrates velocity values in the plan view for the four different discharge ratios. As can be seen from the figure, an increase in the discharge value results in an increase in the velocities near and after the confluence showing relatively higher values through the outer wall of the main channel. This expected behavior results from the difference in discharge values at the primary and lateral channels and the lateral flow influence.

Due to the confluent flows, the generated regions mentioned in the Literature Review section are also visible. The location of the stagnation point, which determines the minimum velocity, varies depending on the discharge ratio (Zhang et al., 2009). This region is more visible in the small q^* cases. As the q^* increases, this zone becomes thinner, shifting from the intersection corner of the two flows to the outer bank of the main channel. The flow separation zones generated just downstream of the confluence are getting thinner and longer with the increased q^* values, the shear layer between the two meeting flows shifts towards the left side of the main channel, resulting in a smaller contracted flow region in the vena contracta (Zhang et al., 2009). The reduced contraction of the main channel flow leads to lower velocities in the junction region, generating a thinner and longer separation zone. The separation zone in the near-confluence region has a more extended profile in large q^* cases. Outside the separation zone, velocities are significantly higher. Consistent with the literature, a low-velocity region is also observed near the upstream corner of the branch channel entrance (Zhang et al., 2009).

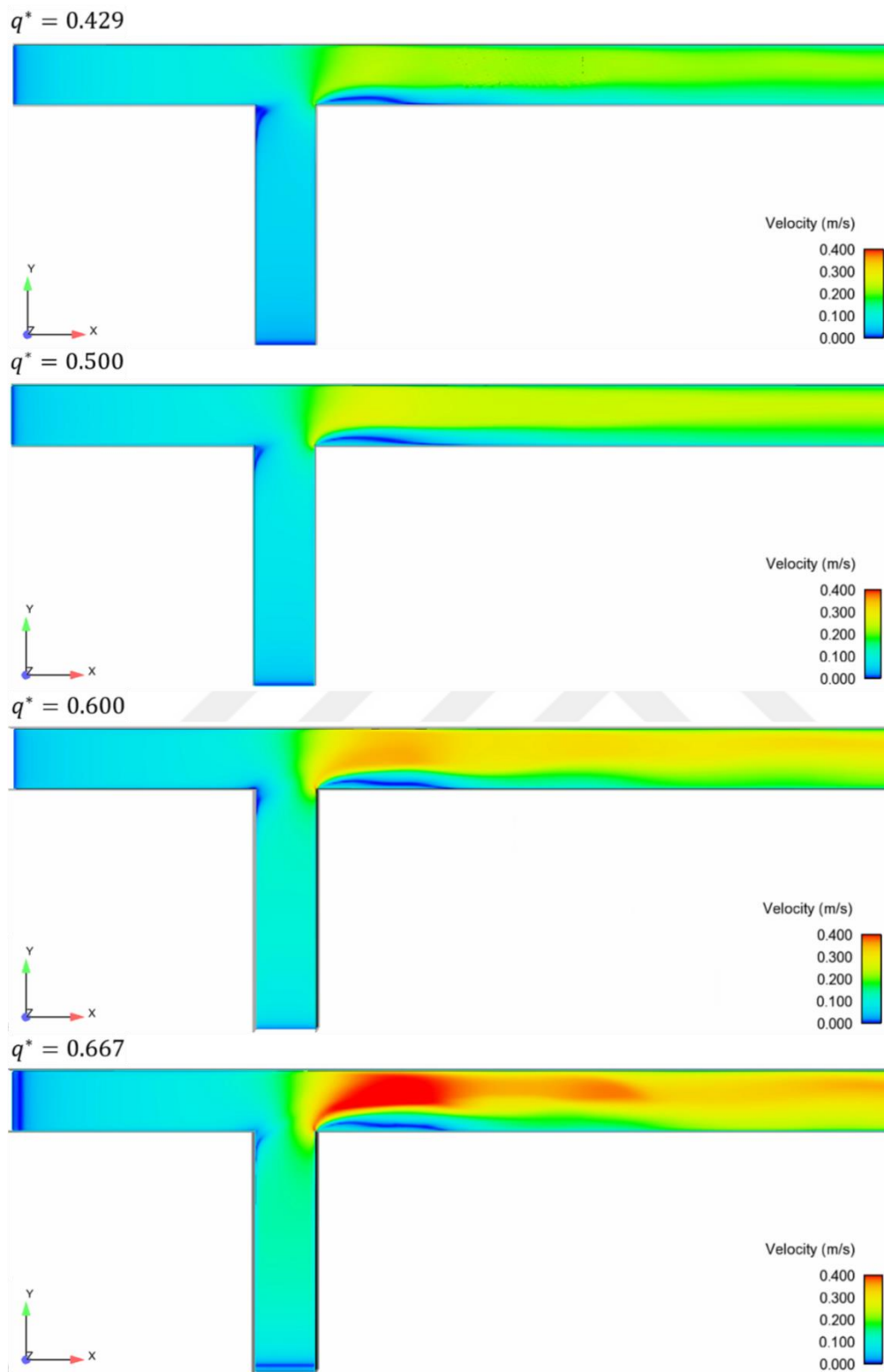


Figure 5.26. Velocity profiles for different discharge ratios.
in plan view.

As in Section 5.4.3, the probe values were used to generate similar graphs to investigate each flow characteristic quantitatively. Eight points were selected again at the same locations in Section 5.4.3 to understand better the flow behavior near the inner wall and in the middle of the channel with changing discharge ratios. Figure 5.27 presents the velocity values at the selected locations near the confluence and far away from the confluence.

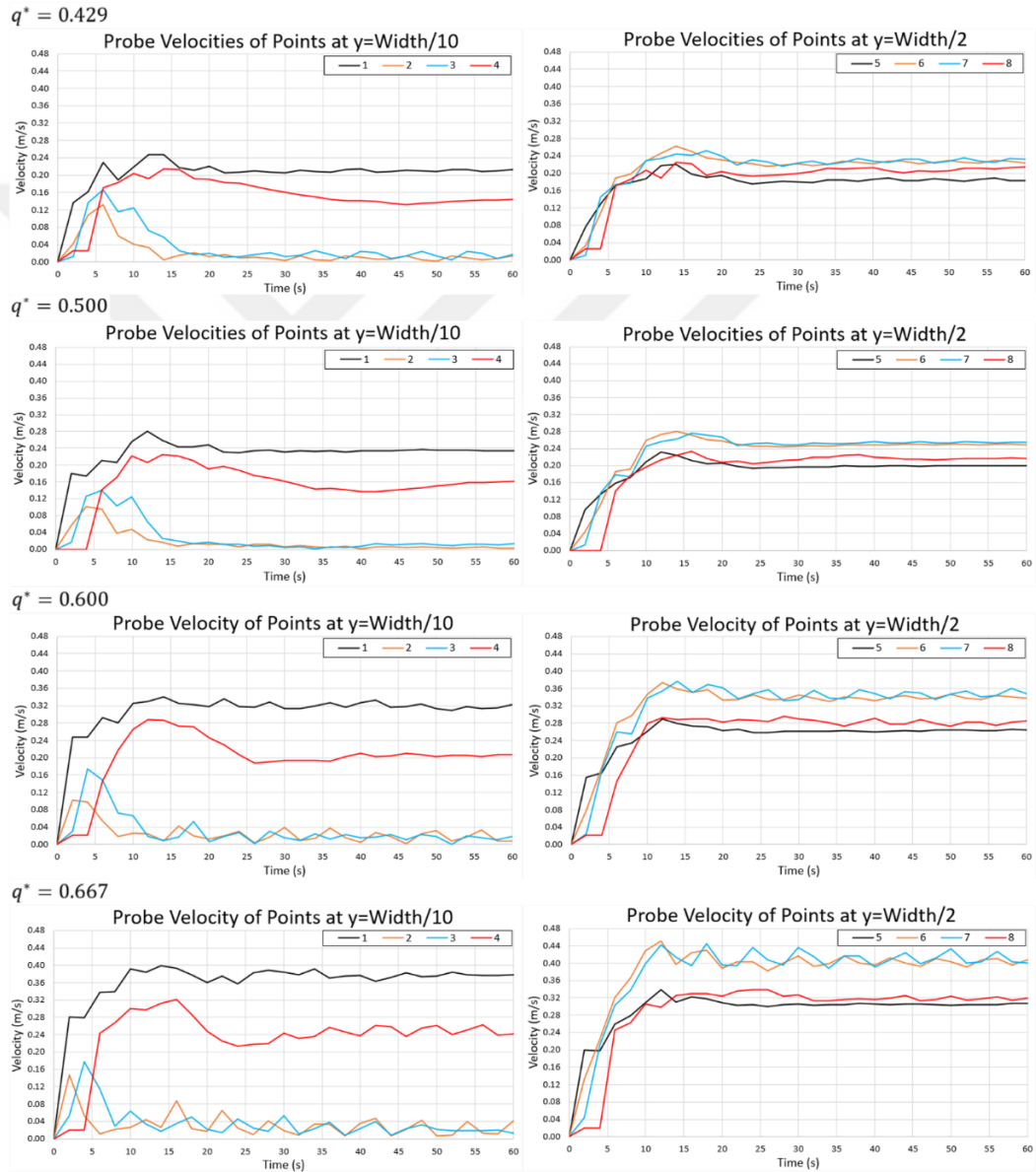


Figure 5.27. Velocity values of the selected probe points near the inner wall (width/10), and in the middle of the channel (width/2) for different discharge ratios.

For 60 seconds simulation duration, the change in flow velocities is shown through time. As shown in Figure 5.27, as the discharge ratio increases, the velocity values of the selected points also increase. The lower velocity values can be observed for $q^* = 0.429$ since the lateral flow decreases as the lateral channel velocity decreases. Generally speaking, the points located near the inner wall of the main channel ($width/10$) have lower velocity values when compared to the points located in the middle of the channel. In section 5.4.3, it was mentioned that the points that stay in the flow separation zone have lower velocity values due to the recirculating motion in that region. This behavior is consistent with the literature. Also, the oscillatory motion that was mentioned above can be seen from the graphs mainly on the higher discharge ratios, $q^* = 0.600$, and $q^* = 0.667$. In the case of $q^* = 0.500$, since the same discharge flows come from the primary and lateral channels, the oscillations generated downstream of the confluence are smaller than the other cases.

Froude Number values in plan-view with changing discharge ratio were investigated and presented in Figure 5.28. The shear layer generated between the meeting flows apparent with higher discharge ratios. The stagnation zone is visible in lower discharge ratios and disappears as the flow ratio increases. The length of the flow separation or recirculation region increases with an increased flow ratio

Figure 5.29 presents the probe Froude Number values that were located similarly at the middle of the channel ($y = width/2$) and near the inner bank ($y = width/10$). As in the velocity case, the oscillatory behavior is apparent for higher discharge ratios. Smaller Froude Numbers result from the recirculation zone since the points near the inner bank (points 1, 2, and 3) stay in this zone.

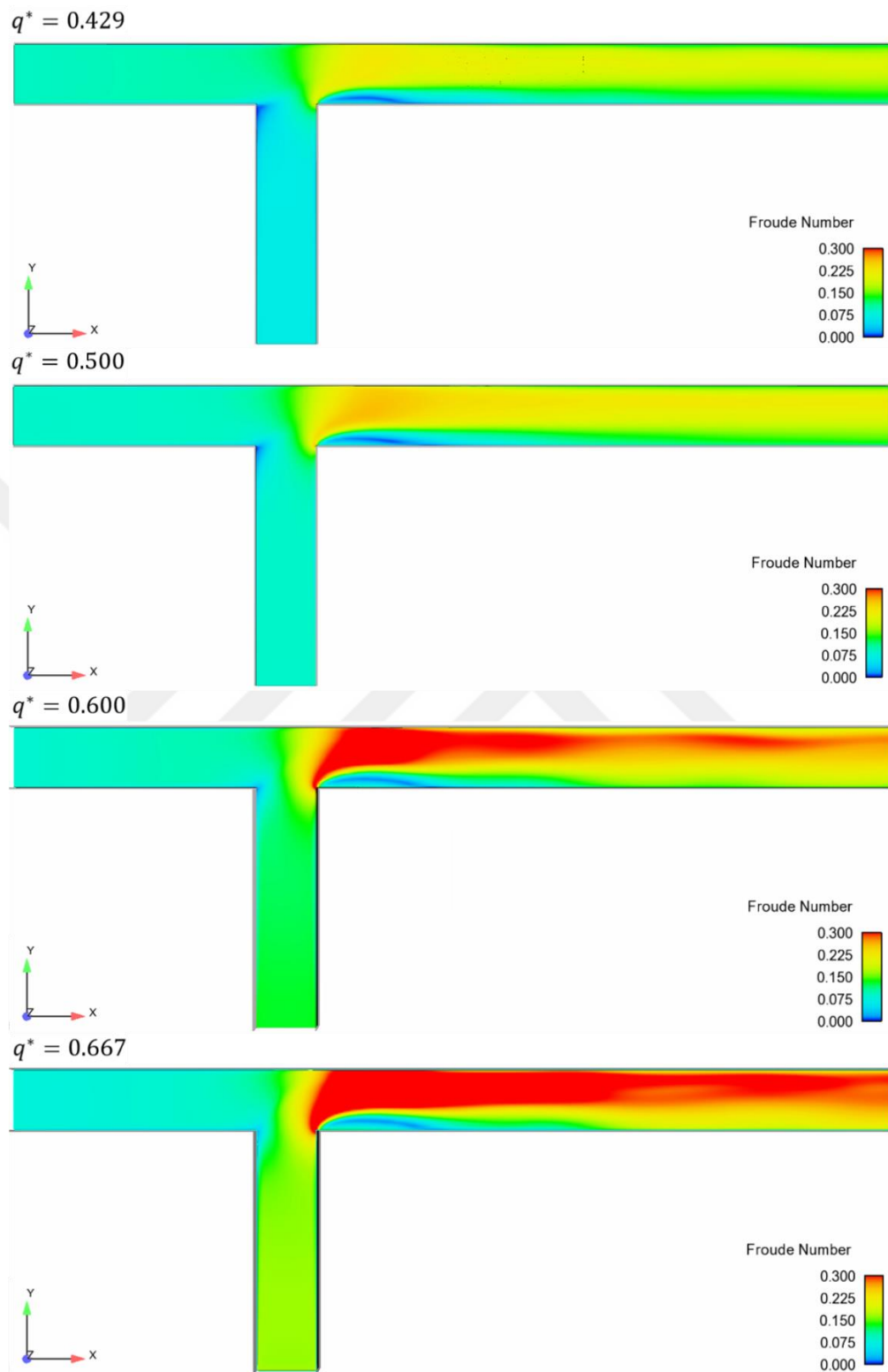


Figure 5.28. Froude Number profiles for different discharge ratios in plan view

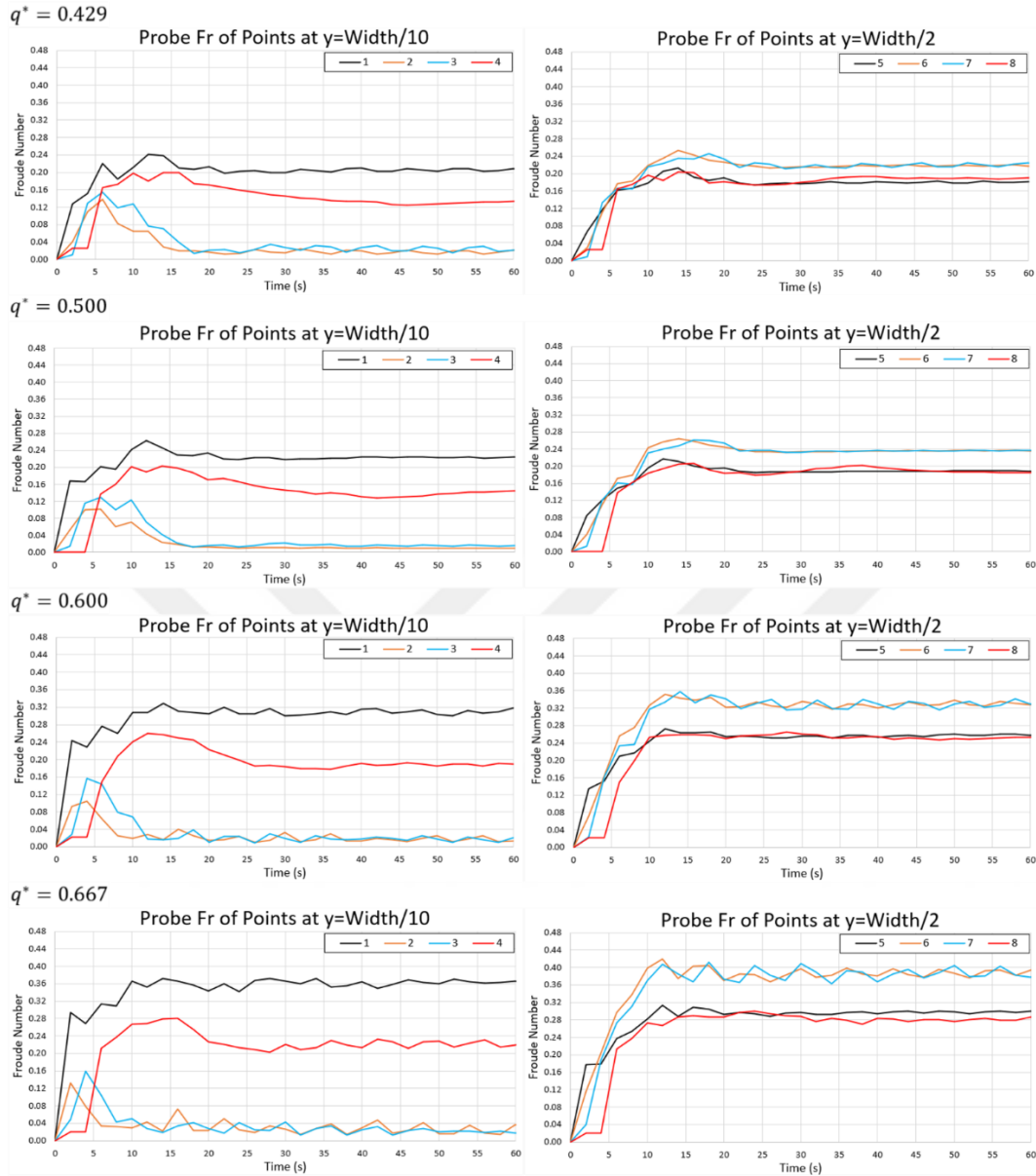


Figure 5.29. Froude Number values of the selected probe points near the inner wall (width/10), and in the middle of the channel (width/2) for different discharge ratios.

Another comparison was made for the turbulent kinetic energies (TKE) with different discharge ratios, as demonstrated in Figure 5.30. TKE increases after the confluence as the discharge ratio increases. Also, the higher TKE moves downstream, lengthening the recirculation zone through the inner bank. From the stagnation zone to the recirculation zone, a separated layer is generated in higher discharge ratios in the flow direction where the oscillatory behavior is still apparent.

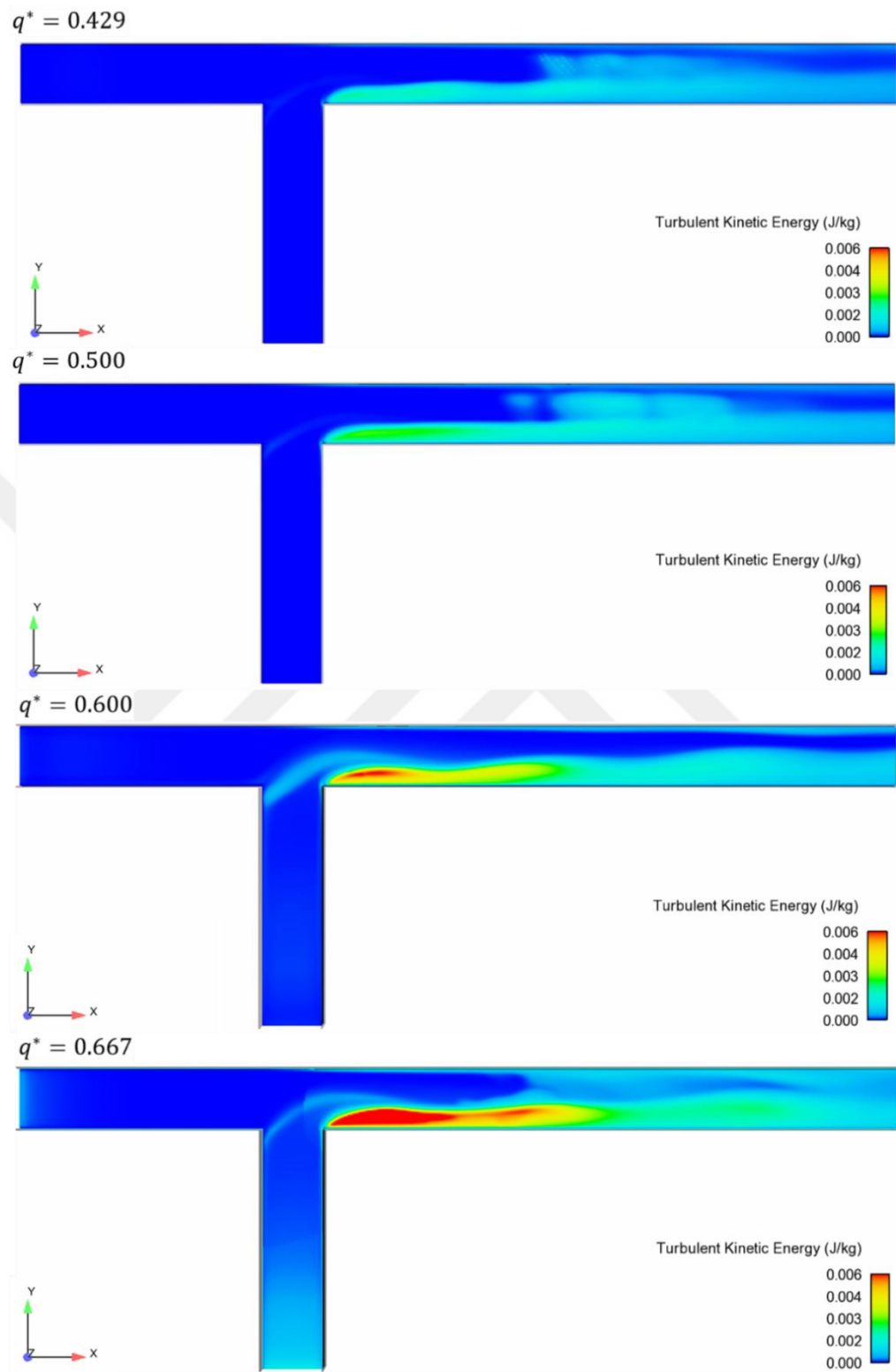


Figure 5.30. Turbulent Kinetic Energy profiles for different discharge ratios in plan view

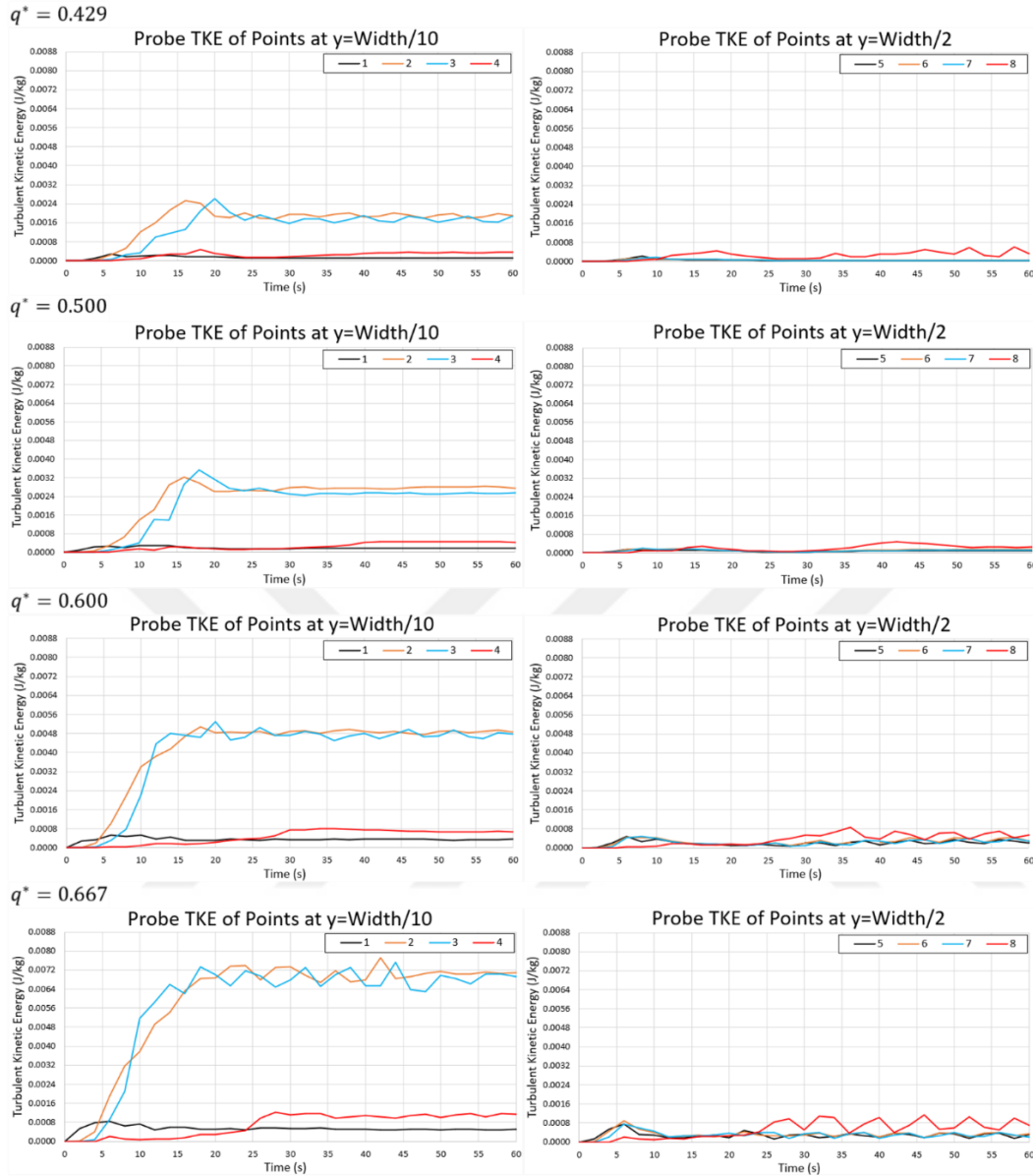


Figure 5.31. TKE values of the selected probe points near the inner wall (width/10), and in the middle of the channel (width/2) for different discharge ratios.

The graphs generated for TKE values show the difference in each point with different discharge ratios. Higher TKE values can be observed in the recirculation region, where the flow dynamically circulates through the simulation period - Figure 5.31. With increasing flow ratios, TKE values also increase, specifically near the inner bank region ($y = Width/10$), and the oscillatory motion causes the turbulent characteristics to spread through the downstream region of confluence.

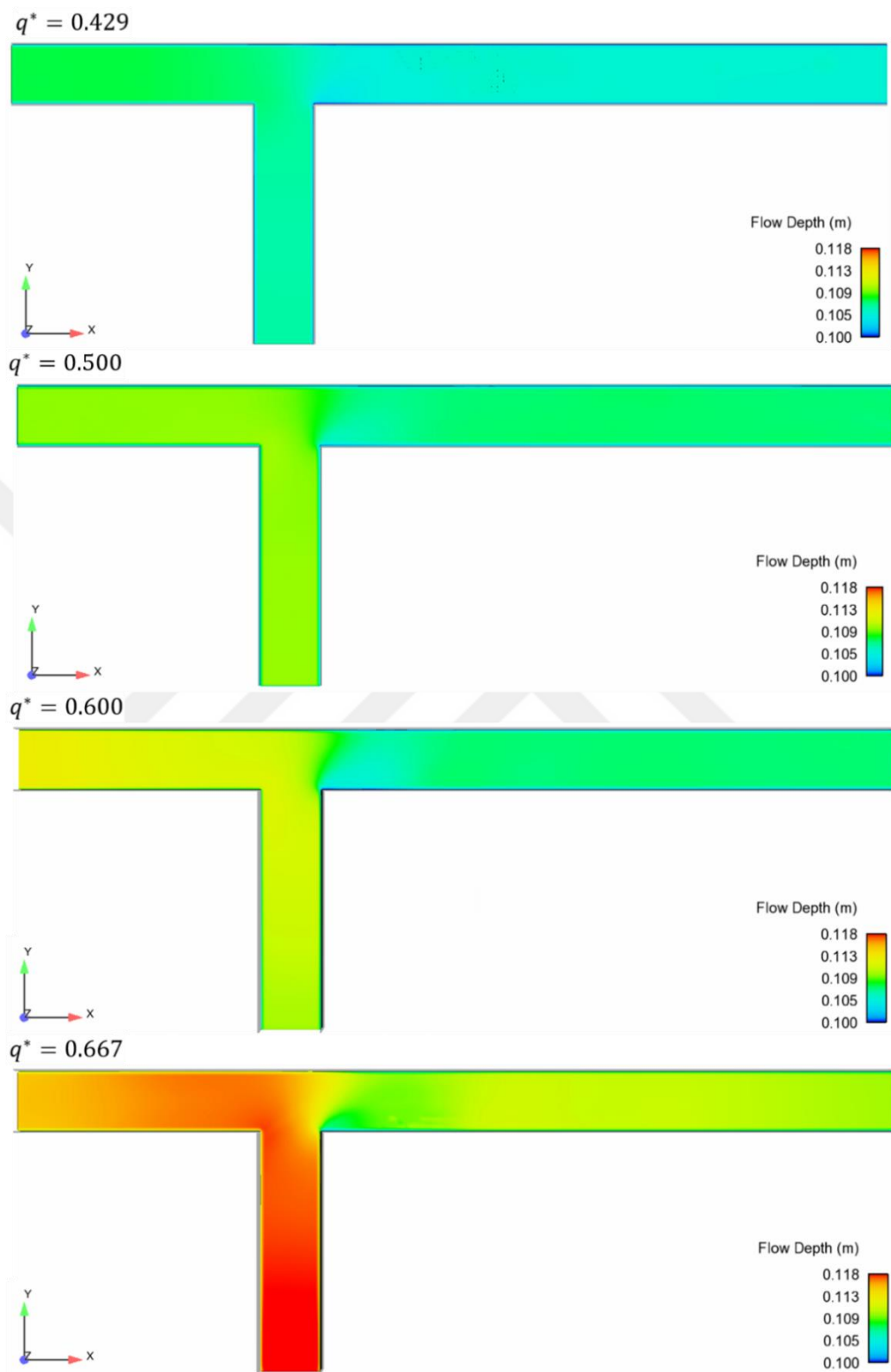


Figure 5.32. Flow depth profiles for different discharge ratios in plan view.

Representative flow depths with changing discharge ratios are shown in Figure 5.32. Initial flow depths were defined as 0.10 *cm*. After the plan views were generated, it was noticed that the flow depths proportionally increased with higher velocities close to the confluence. Lower depths can be seen at the confluence, specifically at the recirculation region.

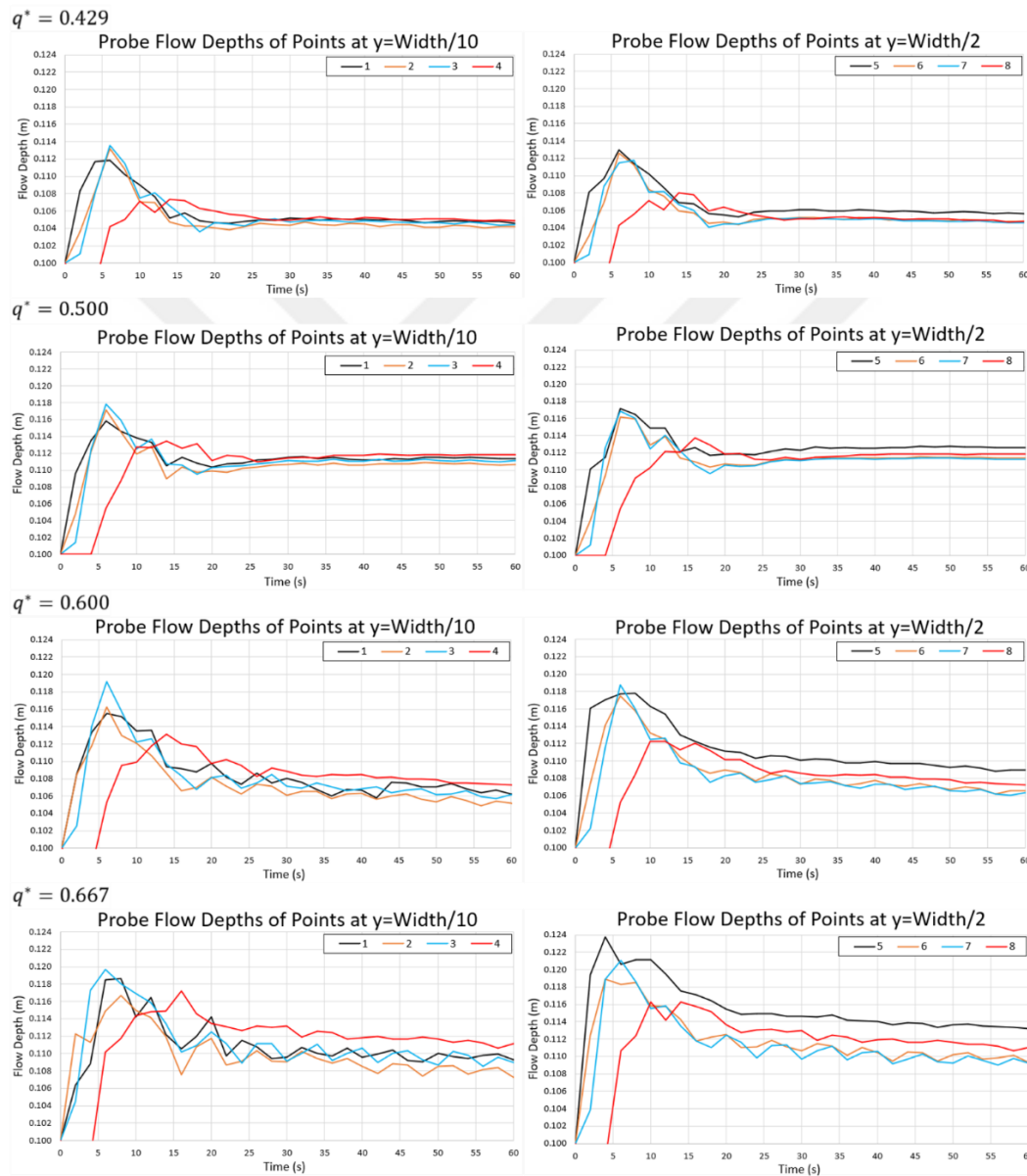


Figure 5.33. Flow depth values of the selected probe points near the inner wall (width/10), and in the middle of the channel (width/2) for different discharge ratios.

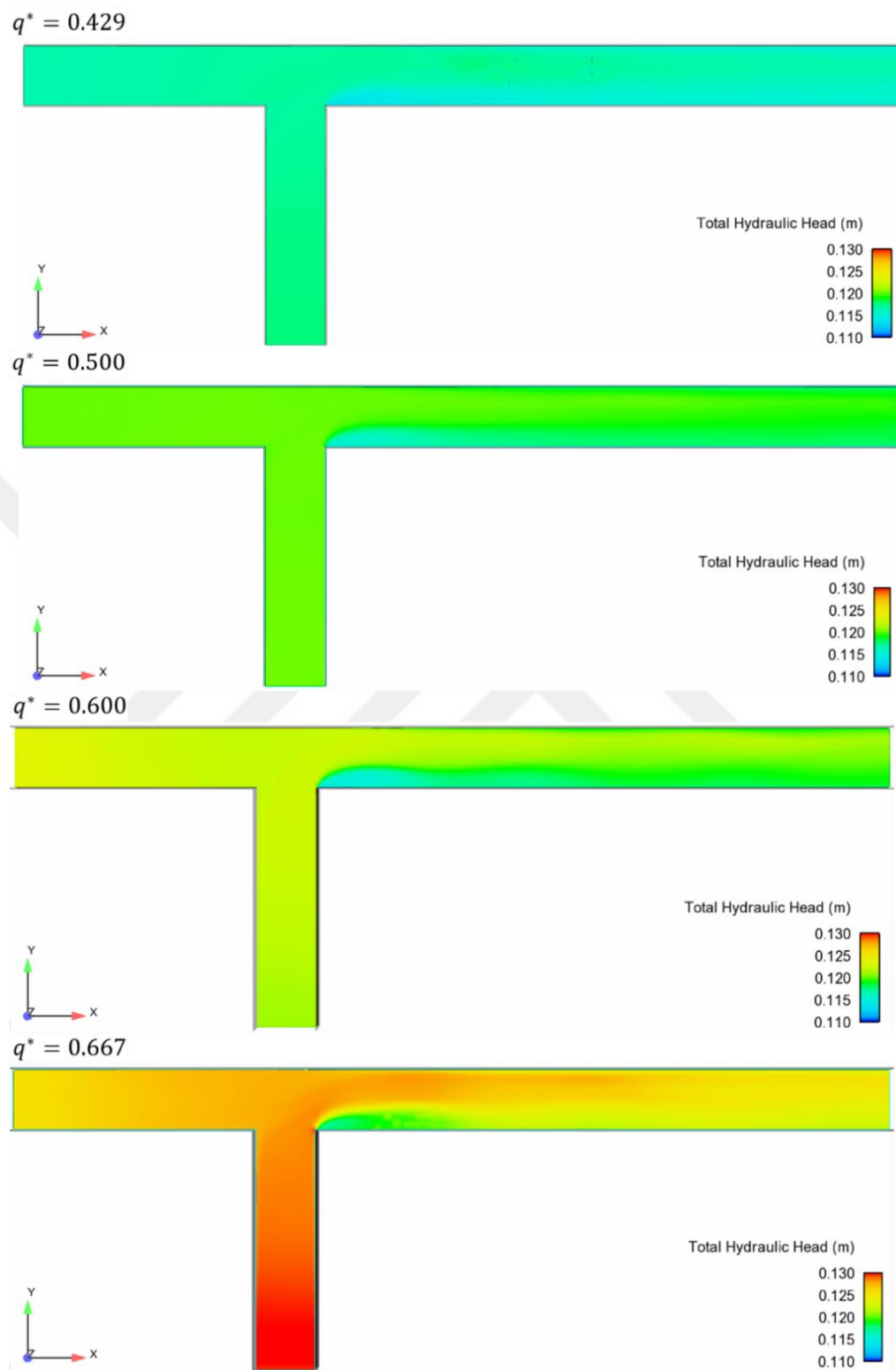


Figure 5.34. Total hydraulic head profiles for different discharge ratios in plan view

Figure 5.34 presents confluent channels' total hydraulic heads (THHs) with different discharge ratios. The total hydraulic head, which depends on the velocity and flow depth, increases as the velocity and flow depth increase. Since the increase was apparent in these values, as mentioned above, the total heads also increased with the increasing discharge ratio. As the velocity and flow depth increase, a dramatic increase in THH values can be seen clearly.

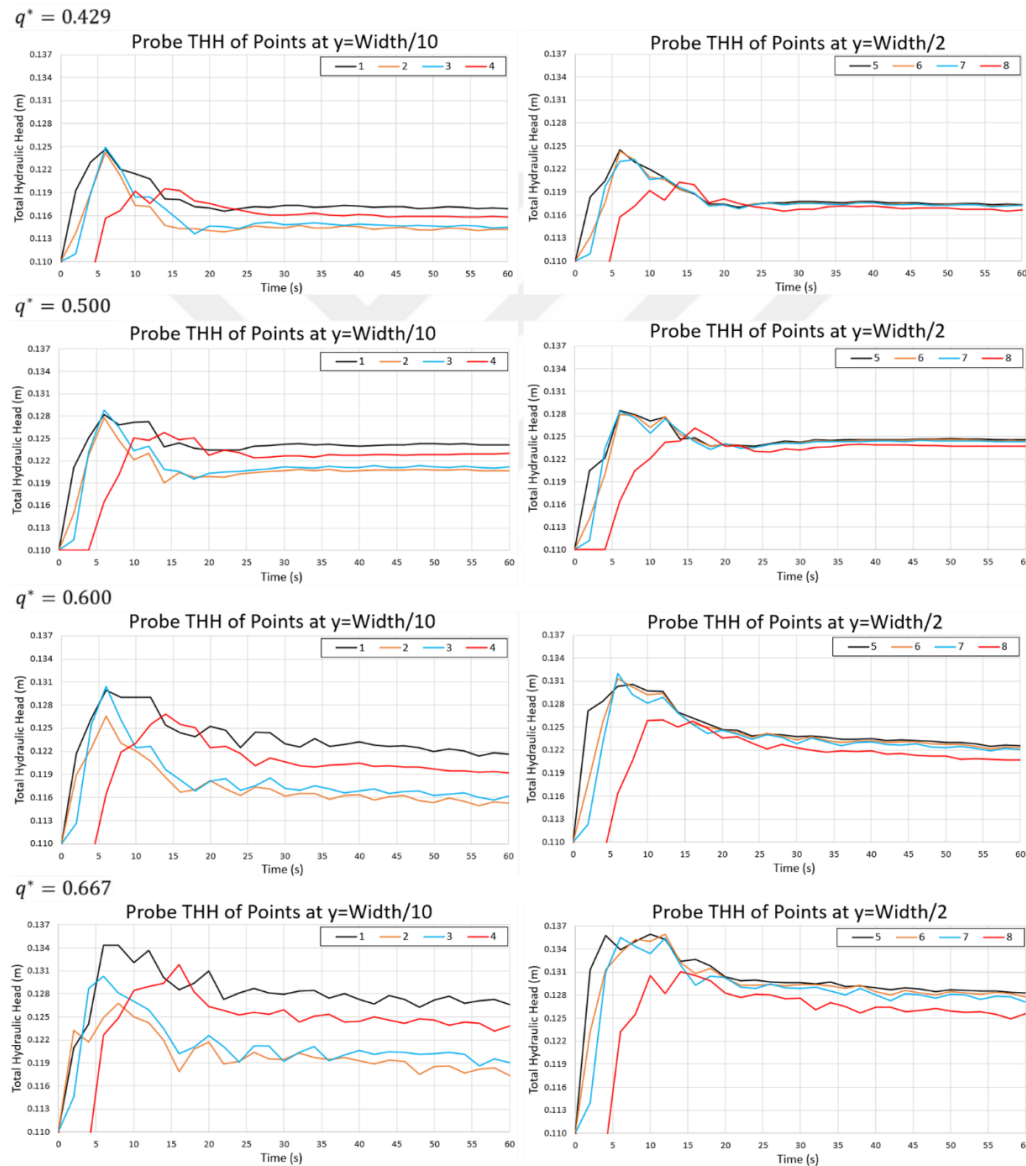


Figure 5.35. Total hydraulic head values of the selected probe points near the inner wall (width/10), and in the middle of the channel (width/2) for different discharge ratios.

5.6 Concluding Remarks

In this chapter of the thesis, open channel confluences were studied, where the effects of confluence angle and discharge ratio were investigated. Since significant characteristics are formed downstream of the two merging flows (Rice et al., 2008), it is crucial to enhance the understanding of the factors that affect downstream characteristics, as also suggested by Shen et al. (2022).

The numerical study performed in this thesis was conducted with a widely used CFD software, Flow-3D. The channel geometries were generated in Autodesk Inventor Pro, and the meshes were created in Flow-3D. After mesh independence was established for 10 mm mesh, the capability of three turbulence models was investigated. *Standard $k - \varepsilon$* model was chosen considering the computational time required to achieve an equilibrium-state solution and the capability to capture the dynamic nature of the flow in confluent channels. Significant findings can be listed as follows:

- The most dynamic streamlined profile was achieved with $k - \omega$ turbulence model when compared to the other models. *Standard $k - \varepsilon$* model gave a partially dynamic profile, whereas *RNG $k - \varepsilon$* model had a stable profile during the simulation.
- For the surface velocities, the results generated by *standard $k - \varepsilon$* and *RNG $k - \varepsilon$* models were similar, where the reattachment lengths were generated almost on the same plane.
- The depth-averaged velocity distribution in $k - \omega$ model was oscillatory due to the dynamic nature of $k - \omega$ model. The solution was always dynamic with moving vortex cores, and the simulation duration was higher when compared with the durations by *standard $k - \varepsilon$* and *RNG $k - \varepsilon$* models.

The difference in flow dynamics introduced by confluence angles was investigated with 30°, 60°, and 90° confluence angles by analyzing different model parameters. Significant findings can be listed as follows:

- When the confluence angle decreased from 90° to 30°, the TKE value at the flow surface also decreased.
- The higher surface velocity was observed in a 90° channel confluence across the outer all and progressed in that region. Due to the vortex formation near the junction, the velocity magnitude falls close to 0 m/s as a thin layer in that region at larger angles. In the 30° channel confluence, the velocity magnitude spreads through the channel width showing an organized motion.
- Depth-averaged velocities differed near the outer and inner wall layers near the junction, creating a shear layer at larger angles. Due to this shear layer, Froude Numbers near and after the confluence are also affected, increasing through the outer bank of the main channel.

Another analysis was performed for cross-sectional views for the same confluence angles. It was seen that the shear layer is visible in a 90° confluence. With the increasing confluence angle, secondary velocity was a dominant feature in the downstream part of the confluence since they were apparent downstream of the confluence in the recirculation region. It was also seen that when the confluence angle decreased, the recirculation zone width and length also decreases. And secondary velocity magnitudes increased as the confluence angle increases from 30° to 90°.

Another analysis was performed by selecting eight points on the study domain to generate the model results for 90°, 60°, and 30° channels. Probes were placed on the lines of $y = width/10$ (near the inner bank) and $y = width/2$ (in the middle of the channel) to capture the confluence effect on the flow behavior near this region. Significant findings can be listed as follows:

- The wall just downstream and far away from the confluence affected the points located near the inner bank region. In the middle of the channel, the behavior introduced by the confluence was lower. The recirculation or flow separation zones were apparent since some selected points stayed in this region.

- The fluctuations and higher values were apparent in $\theta = 90^\circ$ confluence. In $\theta = 30^\circ$ intersection, the velocity values of the points had a slight difference, nearly uniform, since with the decreased confluence angle, the confluence effect also decreases.
- In the $\theta = 30^\circ$ confluence, the recirculation zone was smaller in width, which allows the incoming flows to accelerate each other rather than creating a stagnation layer. Also, due to the turbulence, the downstream flow after the confluence had a fluctuating profile near the inner bank.
- The highest and lowest velocity values were observed in $\theta = 90^\circ$ confluent channels. In 90° confluence, Froude number values got closer for the points at the same horizontal lines. However, the variance was not dominant in the middle of the channel, as in the near-wall region.
- The depth profiles in 90° and 60° confluences showed a uniform behavior after some time; however, the oscillatory depth profile was apparent in 30° confluence.
- TKE was higher in the 90° confluence case, and as the angle decreased, the turbulence effect decreased as well downstream of the confluence.

Another analysis was performed to investigate the effect of different discharge ratios, q^* , that was the ratio between the incoming lateral flow discharge and the combined total flow discharge. Discharge ratios selected for this study were 0.429, 0.500, 0.600, and 0.667. Important findings can be listed as follows:

- An increase in the discharge ratio induced an increase in the velocities near and after the confluence showing relatively higher values through the outer wall of the main channel.
- The location of the stagnation zone varied depending on the discharge ratio. The flow separation zones generated just downstream of the confluence were getting thinner and longer with the increased q^* values, the shear layer between the two meeting flows shifted towards the outer wall, resulting in a smaller contracted flow region.

- The separation zone in the near-confluence region had a more extended profile in large q^* cases.
- The maximum velocity region became apparent with an increased q^* . The length of the flow separation or recirculation region increased with an increased q^* , having an oscillatory behavior more clearly.
- An increase in TKE values was observed after the confluence as the discharge ratio, q^* increases. Also, the higher TKE moved downstream, lengthening the recirculation zone through the inner bank. From the stagnation zone to the recirculation zone, a separated layer was generated in higher discharge ratios in the flow direction where the oscillatory behavior was still apparent.
- The total hydraulic head, which depends on the velocity and flow depth, increased with the increased velocity and flow depth. Since the increase was apparent in these values, the total heads also increased with the increasing q^* .

CHAPTER 6

SUMMARY AND CONCLUSIONS

This study focused on investigating river confluences utilizing a 1D hydraulic model, HEC-RAS, and channel confluences utilizing a 3D CFD model, Flow-3D.

The first part of the study involved employing the HEC-RAS model to analyze the Ohio-Mississippi confluence. A sensitivity analysis was conducted by perturbing various model parameters and generating hypothetical cases. The aim was to understand the behavior of the large-scale model and assess its capability in simulating confluence dynamics. A case study was also performed on a nearly 9 km Cache Creek reach, utilizing observed inputs. This allowed for an evaluation between the model's simulation and the real-world scenario, providing insights into the model's accuracy and performance.

In the second part of the study, the 3D CFD model, Flow-3D, was used for a hypothetical junction problem to investigate flow dynamics near river junctions. First, a mesh independence analysis was performed to determine the appropriate mesh size for accurate simulations. Subsequently, the performance of different turbulence models, namely *standard* $k - \varepsilon$, $k - \omega$, and *RNG* $k - \varepsilon$, was investigated using a confluence channel geometry with a 90° angle. Once the most suitable turbulence model and mesh size were determined for this study, the flow behavior at different confluence angles, precisely 90° , 60° , and 30° , was examined in the equilibrium flow state. The effects of flow ratio on flow dynamics were then explored, focusing on a 90° confluence angle. Various variables were analyzed, including turbulent kinetic energy, surface velocity, depth-averaged velocity, Froude number in plan view flow analyses, velocity magnitude, secondary velocities, and turbulent kinetic energies in cross-section flow analyses; velocity, Froude Number,

flow depth, total hydraulic head, and turbulent kinetic energy in point location analyses.

The study's findings align with existing literature, indicating that as the confluence angle decreases, the length and width of the recirculation zone also decrease. Furthermore, the magnitudes of secondary velocities become more pronounced as the confluence angle rises from 30° to 90°. Additionally, the study reveals that the flow ratio impacts the characteristics of the recirculation zone and influences the flow dynamics downstream of the confluence.

Overall, this study provides valuable insights into the behavior of river/channel confluences by employing both a 1D hydraulic model and a 3D CFD model. The combination of these approaches offers a comprehensive understanding of the flow dynamics, sediment transport/deposition, and bed morphology at junctions, thereby contributing to effective river management and conservation strategies.

In this study, numerical simulations were conducted considering the river confluences in 1D and channel confluences in 3D, investigating the flow dynamics near the confluence region. In the future, laboratory experiments should be performed to calibrate and validate the numerical simulations. Since only three turbulence models, three confluence angles, and four discharge ratios were investigated in this study, the performance of other turbulence models, other confluence angles, including a reverse one, and also other discharge ratios should be examined qualitatively in the future.

REFERENCES

- Akan, A. O. (2006). *Open Channel Hydraulics*, Elsevier, New York.
- Akram Abbaspour, & Kia, S. (2014). Numerical investigation of turbulent open channel flow with semi-cylindrical rough beds. *Ksce Journal of Civil Engineering*, 18(7), 2252–2260.
- Almohammadi, K. M., Ingham, D. B., Ma, L., & Pourkashan, M. (2013). Computational fluid dynamics (CFD) mesh independency techniques for a straight blade vertical axis wind turbine. *Energy*, 58, 483–493.
- Ardıçlıoğlu, M., & Kuriqi, A. (2019). Calibration of channel roughness in intermittent rivers using HEC-RAS model: case of Sarimsakli creek, Turkey. *SN Applied Sciences*, 1(9).
- Barkau, R. L. (1982). *Simulation of the July flood along the Salt River. Report for CE695Bv. Special Problems in Hydraulics*. Collins, CO. Colorado State University. Department of Civil Engineering pp. 231.
- Bates, P. D., Lane, S. N., & Ferguson, R. I. (2005). *Computational fluid dynamics: applications in environmental hydraulics*. J. Wiley.
- Best, J. (1987). *Flow Dynamics at River Channel Confluences: Implications for Sediment Transport and Bed Morphology*.
- Best, J. L., & Reid, I. (1984). Separation Zone at Open-Channel Junctions. *Journal of Hydraulic Engineering*, 110(11), 1588–1594.
- Blanch, R. G. (2017). *Effect of the sub-grid geometry on two-dimensional river flow models*. Department of Civil Engineering Delft.

- Brahimi, M., & Afzalimehr, H. (2018). Effect of submerged vegetation density on flow under favorable pressure gradient. *SN Applied Sciences*, 1(1).
- Brunner, G. W. (1994). *HEC River Analysis System (HEC-RAS)*.
- Burden, R. L., J. Douglas Faires, & Burden, A. M. (2015). *Numerical Analysis*. Cengage Learning.
- Butt, M. J., Umar, M., & Qamar, R. (2012). Landslide dam and subsequent dam-break flood estimation using HEC-RAS model in Northern Pakistan. *Natural Hazards*, 65(1), 241–254.
- Cayan, D. R., Luers, A. L., Franco, G., Hanemann, M., Croes, B., & Vine, E. (2008). Overview of the California climate change scenarios project. *Climatic Change*, 87(S1), 1–6.
- Chow. (1959). *Open-Channel Hydraulics*. McGraw-Hill Book Co., Inc., New York, N.Y., 10–11.
- Chow, V. T. (2009). *Open-channel hydraulics*. Caldwell: Blackburn Press.
- Chow, Maidment, D. R., & Mays, L. W. (2013). *Applied hydrology*. Mcgraw-Hill Professional ; London.
- Domagalski, J. L., Alpers, C. N., Slotton, D. G., Suchanek, T. H., & Ayers, S. M. (2004). Mercury and methylmercury concentrations and loads in the Cache Creek watershed, California. *Science of the Total Environment*, 327(1-3), 215–237.
- Drake, J., Bradford, A., & Joy, D. (2010). Application of HEC-RAS 4.0 temperature model to estimate groundwater contributions to Swan Creek, Ontario, Canada. *Journal of Hydrology*, 389(3-4), 390–398.

- Dronkers, J. J. (1964). *Tidal Computations in Rivers and Coastal Waters*. North-Holland Pub. Co.; Interscience Publishers, Amsterdam, New York, 1964.
- El Bilali, A., Taleb, A., & Boutahri, I. (2021). Application of HEC-RAS and HEC-LifeSim models for flood risk assessment. *Journal of Applied Water Engineering and Research*, 9(4), 336–351.
- Fan, C., Ko, C.-H., & Wang, W.-S. (2009). An innovative modeling approach using Qual2K and HEC-RAS integration to assess the impact of tidal effect on River Water quality simulation. *Journal of Environmental Management*, 90(5), 1824–1832.
- Ferziger, J. H., Milovan Perić, & Street, R. L. (2019). *Computational Methods for Fluid Dynamics*. Springer.
- Fischer, I. (2014). *Flood Frequency Analysis of Future Climate Projections for Cache Creek Watershed* (Order No. 1585068). Available from ProQuest Dissertations & Theses Global. (1665309170). In <https://www.proquest.com/dissertations-theses/flood-frequency-analysis-future-climate/docview/1665309170/se-2>.
- Flow Science, Inc. (2019). *FLOW-3D Version 12.0 Users Manual (2018)*.
- Gerald, C. F., & Wheatley, P. O. (1984). *Applied numerical analysis*. Addison-Wesley.
- Ghanbarpour, M. R., Saravi, M. M., & Salimi, S. (2014). Floodplain Inundation Analysis Combined with Contingent Valuation: Implications for Sustainable Flood Risk Management. *Water Resources Management*, 28(9), 2491–2505.
- Gilany, N., & Iqbal, J. (2020). Geospatial analysis and simulation of glacial lake outburst flood hazard in Shyok Basin of Pakistan. *Environmental Earth Sciences*, 79(6).

- Goldman, C. R., & Wetzel, R. G. (1963). A Study of the Primary Productivity of Clear Lake, Lake County California. *Ecology*, 44(2), 283–294.
- Goudarzizadeh, R., & Musavi-Jahromi, S. H. (2020). Experiment-Based CFD Simulation of Flow Pattern at Open-Channel Junctions. *Iranian Journal of Science and Technology, Transactions of Civil Engineering*, 44(S1), 427–436.
- Graziani, A., Lippert, M., Uystepuyst, D., & Laurent Keirsbulck. (2017). Scaling and flow dependencies over forward-facing steps. *International Journal of Heat and Fluid Flow*, 67, 220–229.
- Gül, G. O., Harmancıoğlu, N., & Gül, A. (2009). A combined hydrologic and hydraulic modeling approach for testing efficiency of structural flood control measures. *Natural Hazards*, 54(2), 245–260.
- Gurram, S. K., Karki, K. S., Hager, W. H., & Fellow. (1997). Subcritical junction flow. *Journal of Hydraulic Engineering*, 123(5), 447-455.
- Hager, W. H. (1989). Transitional Flow in Channel Junctions. *Journal of Hydraulic Engineering*, 115(2), 243–259.
- Hager, W. H., & Castro-Orgaz, O. (2017). William Froude and the Froude Number. *Journal of Hydraulic Engineering*, 143(4), 02516005.
- Hicks, F. E., & Peacock, T. (2005). Suitability of HEC-RAS for Flood Forecasting. *Canadian Water Resources Journal*, 30(2), 159–174.
- Higgins, S. A., & Kamman, G. R. (2010). *Watershed-Based Assessment of Hydrologic and Geomorphic Conditions in Cache Creek through Capay Valley*. Prepared for the Yolo County Resource Conservation District.
- Hijmans, R. J. (2023). The Raster Package. In *Spatial Data Science* (p. 49). r-spatial.org

- I. Chirica, Angheluta, C. M., Perijoc, S. D., Hobjilă, A., & M. Frătita. (2019). Mesh independence of a transient multiphase fluid-solid interaction. *Journal of Physics*, 1297, 012026–012026.
- Iehisa Nezu, & Nakagawa, H. (1989). Turbulent Structure of Backward-Facing Step Flow and Coherent Vortex Shedding from Reattachment in Open-Channel Flows. *Springer EBooks*, 313–337.
- Joshi, N., Lamichhane, G. R., Rahaman, Md. M., Kalra, A., & Ahmad, S. (2019). Application of HEC-RAS to Study the Sediment Transport Characteristics of Maumee River in Ohio. *World Environmental and Water Resources Congress 2019*.
- Kim, B.-J., Hwang, J.-H., & Kim, B. (2022). FLOW-3D Model Development for the Analysis of the Flow Characteristics of Downstream Hydraulic Structures. *Sustainability*, 14(17), 10493.
- Kiss, T., Nagy, J., Fehérváry, I., & Vaszkó, C. (2019). (Mis) management of floodplain vegetation: The effect of invasive species on vegetation roughness and flood levels. *Science of the Total Environment*, 686, 931–945.
- Kumar, N., Lal, D., Sherring, A., & Issac, R. K. (2017). Applicability of HEC-RAS & GFMS tool for 1D water surface elevation/flood modeling of the river: a Case Study of River Yamuna at Allahabad (Sangam), India. *Modeling Earth Systems and Environment*, 3(4), 1463–1475.
- Leigh, C., Boulton, A. J., Courtwright, J. L., Fritz, K., May, C. L., Walker, R. H., & Datry, T. (2015). Ecological research and management of intermittent rivers: an historical review and future directions. *Freshwater Biology*, 61(8), 1181–1199.
- Leite Ribeiro, M., Blanckaert, K., Roy, A. G., & Schleiss, A. J. (2012). Flow and sediment dynamics in channel confluences. *Journal of Geophysical Research: Earth Surface*, 117(F1).

- Liang, Q., & Borthwick, A. G. L. (2009). Adaptive quadtree simulation of shallow flows with wet–dry fronts over complex topography. *Computers & Fluids*, 38(2), 221–234.
- Lin, F., Chen, X., & Yao, H. (2017). Evaluating the Use of Nash-Sutcliffe Efficiency Coefficient in Goodness-of-Fit Measures for Daily Runoff Simulation with SWAT. *Journal of Hydrologic Engineering*, 22(11), 05017023.
- Liu, Z., & Merwade, V. (2018). Accounting for model structure, parameter and input forcing uncertainty in flood inundation modeling using Bayesian model averaging. *Journal of Hydrology*, 565, 138–149.
- Lopes, P. M. B. (2013). *Free-surface flow interface and air-entrainment modelling using OpenFOAM* (Doctoral Dissertation).
- Mandal, S. P., & Chakrabarty, A. (2016). Flash flood risk assessment for upper Teesta River basin: using the hydrological modeling system (HEC-HMS) software. *Modeling Earth Systems and Environment*, 2(2).
- Mei, C. C. (2001). *Advanced Environmental Fluid Mechanics*. MIT.
- Menter, F. R. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*, 32(8), 1598–1605.
- Mondal, I., Bandyopadhyay, J., & Paul, A. Kr. (2016). Estimation of hydrodynamic pattern change of Ichamati River using HEC RAS model, West Bengal, India. *Modeling Earth Systems and Environment*, 2(3).
- Mosley, M. P. (1976). An Experimental Study of Channel Confluences. *The Journal of Geology*, 84(5), 535–562.
- Mueller, Z. P. (2008). *Feasibility Study of Removing the Grand Rapids-Providence Dams, Maumee River (Nw Ohio) Based on Hec-Ras Models* (p. 253) (Thesis).

- Papaioannou, G., Vasiliades, L., Loukas, A., & Aronica, G. T. (2017). Probabilistic flood inundation mapping at ungauged streams due to roughness coefficient uncertainty in hydraulic modelling. *Advances in Geosciences*, 44, 23–34.
- Pope, S. B. (2000). *Turbulent flows*. Cambridge: Cambridge University Press.
- Ramesh, R., Datta, B., Bhallamudi, S. M., & Narayana, A. (2000). Optimal Estimation of Roughness in Open-Channel Flows. *Journal of Hydraulic Engineering*, 126(4), 299–303.
- Remo, J. W. F., Carlson, M., & Pinter, N. (2011). Hydraulic and flood-loss modeling of levee, floodplain, and river management strategies, Middle Mississippi River, USA. *Natural Hazards*, 61(2), 551–575.
- Rice, S. P., Roy A. G., & Rhoads, B. L. (2008). *River confluences, tributaries, and the fluvial network*. Wiley.
- Saidi, M. S., Rismanian, M., Monjezi, M., Zendehbad, M., & Fatehiboroujeni, S. (2014). Comparison between Lagrangian and Eulerian approaches in predicting motion of micron-sized particles in laminar flows. *Atmospheric Environment*, 89, 199–206.
- Saleh, F., Ducharne, A., Flipo, N., Oudin, L., & Ledoux, E. (2013). Impact of river bed morphology on discharge and water levels simulated by a 1D Saint–Venant hydraulic model at regional scale. *Journal of Hydrology*, 476, 169–177.
- Shabani, A., Woznicki, S. A., Mehaffey, M., Butcher, J., Wool, T. A., & Whung, P. (2021). A coupled hydrodynamic (HEC-RAS 2D) and water quality model (WASP) for simulating flood-induced soil, sediment, and contaminant transport. *Journal of Flood Risk Management*.
- Shakibainia, A., Reza, M., & Zarrati, A. R. (2010). *Three-dimensional numerical study of flow structure in channel confluences*. 37(5), 772–781.

- Shen, X., Li, R., Cai, H., Feng, J., & Wan, H. (2022). Characteristics of secondary flow and separation zone with different junction angle and flow ratio at river confluences. *Journal of Hydrology*, 614, 128537–128537.
- Shumate, E. D. (1998). *Experimental Description of Flow at an Open-Channel Junction*.
- Speziale, C. G., & Thangam, S. (1992). Analysis of an RNG based turbulence model for separated flows. *International Journal of Engineering Science*, 30(10), 1379-IN4.
- Strelkoff, T. (1970). Numerical Solution of Saint-Venant Equations. *Journal of the Hydraulics Division*, 96(1), 223–252.
- Szomorová, L., & Halaj, P. (2015). Numerical simulations in the Mala Nitra stream by 1D model. *Acta Scientiarum Polonorum Formatio Circumiectus*, 14(2), 185–194.
- Taylor, E. (1944). Flow characteristics at rectangular open channel junction. *Journal of Hydraulic Engineering*, 10((6)), 893–902.
- Toro, E. F., & Garcia-Navarro, P. (2007). Godunov-type methods for free-surface shallow flows: A review. *Journal of Hydraulic Research*, 45(6), 736–751.
- Versteeg, H. K., & Malalasekera, W. (2007). *An introduction to computational fluid dynamics: the finite volume method*. Pearson Education.
- Villarini, G., Taylor, S. H., Wobus, C. W., Vogel, R. M., Hecht, J., White, K., Baker, B., Gilroy, K., Olsen, J. R., & Raff, D. (2018). Floods and Nonstationarity: A Review. In *ResearchGate*.
- Vojtek, M., Petroselli, A., Vojteková, J., & Asgharinia, S. (2019). Flood inundation mapping in small and ungauged basins: sensitivity analysis using the EBA4SUB and HEC-RAS modeling approach. *Hydrology Research*, 50(4), 1002–1019.

- Vreugdenhil, C. B. (1994). *Numerical Methods for Shallow-Water Flow*. Springer Science & Business Media.
- Wang, X. F., & Cheng, L. (2000). Three-dimensional simulation of a side discharge into a cross channel flow. *Computers & Fluids*, 29(4), 415–433.
- Warner, J. C., Brunner, G. W., Wolfe, B. C., Piper, S. S., & Marston, L. (2021). HEC-RAS Applications Guide. In *HEC-RAS Documentation*.
- Wernecke, J. (2008). *The KML Handbook*. Pearson Education.
- Wilcox, D. C. (2010). *Turbulence modeling for CFD*. La Cañada, Calif. Dcw Industries.
- Yakhot, V., & Orszag, S. A. (1986). Renormalization group analysis of turbulence. I. Basic theory. *Journal of Scientific Computing*, 1(1), 3–51.
- Yamani, K., Hazzab, A., Sekkoum, M., & Slimane, T. (2016). Mapping of vulnerability of flooded area in arid region. Case study: area of Ghardaïa-Algeria. *Modeling Earth Systems and Environment*, 2(3).
- Yılmaz, K., Dinçer, A. E., Kalpakçı, V., & Öztürk, Ş. (2022). Debris flow modelling and hazard assessment for a glacier area: a case study in Barsem, Tajikistan. *Natural Hazards*.
- Zellou, B., & Rahali, H. (2016). Assessment of reduced-complexity landscape evolution model suitability to adequately simulate flood events in complex flow conditions. *Natural Hazards*, 86(1), 1–29.
- Zhang, T., Wang, W., Gao, T., & An, B. (2021). Simulation and Assessment of Future Glacial Lake Outburst Floods in the Poiqu River Basin, Central Himalayas. *Water*, 13(10), 1376.

Zhang, T., Xu, W., & Li, V. C. (2009). *Effect of Discharge Ratio on Flow Characteristics in 90° Equal-Width Open-Channel Junction*. 21(4), 541–549.

Zhang, Z., & Chen, Q. (2007). Comparison of the Eulerian and Lagrangian methods for predicting particle transport in enclosed spaces. *Atmospheric Environment*, 41(25), 5236–5248.
<https://engineering.purdue.edu/~yanchen/paper/2007-1.pdf>

Zhang, Z., & Johnson, B. E. (2014). *Application and evaluation of HEC-RAS-nutrient simulation module (NSM I)*. EMRRP Technical Notes Collection. ERDC TN-EMRRP-SR-47.

Zimoch, I., & Bartkiewicz, E. (2018). Process of hydraulic models calibration. *E3S Web of Conferences*, 59, 000.