



공학석사 학위논문

Design of the Tesla Channel to Maximize the Asymmetry of Bidirectional Flows

양방향 흐름 비대칭 극대화를 위한 최적의 테슬라 채널 설계

2023년 2월

서울대학교 대학원

건설환경공학부 건설환경공학전공

손 석 민

양방향 흐름 비대칭 극대화를 위한 최적의 테슬라 채널 설계

Design of the Tesla Channel to Maximize the Asymmetry of Bidirectional Flows

지도교수 황진환

이 논문을 공학석사 학위논문으로 제출함

2022 년 11 월

서울대학교 대학원

건설환경공학부 건설환경공학전공

손석민

손석민의 공학석사 학위논문을 인준함 2022 년 12 월

위 원	릴 장	 (인)
부 위	원 장	 (인)
위	원	(인)

ABSTRACT

In the coastal area, a hydraulic structure such as a dam or a barrage is required to prevent the problems caused by the seawater from the coastal sea. However, it completely separates the upstream and downstream by its vertical wall then attributes to the ecosystem disconnection and bad water quality. Therefore, an alternative structure is needed with controlling the flow horizontally. It should release the freshwater from the upstream at least and block the seawater from the downstream at most; it implies that the structure should maximize the flow asymmetry in bidirectional flows. To implement the idea, the mechanism of the Tesla valve was grafted which forms different flow structures depending on the flow direction with curved flow-control stages. The present study aims to adapt the hydraulic structures in the open channel with bidirectional flows by numerical modeling, and the performance of their design for the flow asymmetry was evaluated by the total volume difference in two directions. An open-source computational fluid dynamics (CFD) software OpenFOAM solved continuity equations for mean velocities and Reynolds-averaged Navier-Stokes (RANS) equations to simulate the flow with the hydraulic structures, and the volume of fluid (VOF) method was applied to describe the free surface flow.

The preliminary study with the numerical model in a simple straight channel domain confirmed the effectiveness of the structures to make the flow asymmetry in a bidirectional flow by controlling the angle between the channel side and the structures. Also, it proposed a unit structure design that consists of two types of structures. The main model was developed to describe the coastal area and included the structures installed based on the unit design. The grid convergence test verified the reasonable numerical mesh and the laboratory experiments validated the CFD model by measuring the free surface levels and velocity profiles. The simulation results decided the numbers, length, location, and shape of the structures to maximize the flow asymmetry, which was evaluated by the total volume difference in a specific period in a bidirectional flow. It suggests the best design of the hydraulic structures and provides a guideline for future design.

Keywords: <u>Hydraulic structure</u>, Tesla valve, flow asymmetry, numerical modeling, <u>RANS equations, VOF method</u>, <u>OpenFOAM</u>

Student Number: 2021-25858

CONTENTS

3.2 Preliminary CFD modeling	
3.1.2 Computing equipment	
3.1.1 Numerical model description	19
3.1 Numerical methods	19
CHAPTER 3. METHODOLOGY	19
2.3.2 Free surface description	16
2.3.1 Reynolds-averaged Navier-Stokes equations	15
2.3 Governing equations	15
2.2.2 Parameters of the Tesla valve	
2.2.1 Characteristics of the Tesla valve	11
2.2 Tesla valve	11
2.1.2 Energy loss in pipe flow	
2.1.1 Wake regions with eddies	7
2.1 Flow structure	7
CHAPTER 2. THEORETICAL BACKGROUNDS	7
1.2 Objectives	4
1.1 General introduction	1
CHAPTER 1. INTRODUCTION	1
List of Symbols	xiii
List of Tables	xi
List of Figures	v
CONTENTS	iii
ABSTRACT	i

3.2.1 Simulation domain	
3.2.2 Simulation conditions setup	
3.2.3 Simulation cases setup	
3.3 Main CFD modeling	
3.3.1 Simulation domain	
3.3.2 Simulation conditions setup	
3.3.3 Mesh generation	
3.3.4 Experimental validation	
3.3.5 Simulation cases setup	
CHAPTER 4. RESULTS AND DISCUSSION	
4.1 Preliminary results	58
4.1.1 An angle between the wall and structures	
4.1.2 The spacing between the main structures	
4.1.3 The number of structures	65
4.2 Model validation	68
4.2.1 Grid convergence test	
4.2.2 Free surface level	
4.2.3 Velocity profile	
4.3 Main results	77
4.3.1 Without structures	77
4.3.2 The number of structures	79
4.3.3 Sub-structures	
4.3.4 Main structures	
4.3.5 Structure asymmetry	
CHAPTER 5. CONCLUSION	
REFERENCES	
APPENDIX	
국문초록	

List of Figures

Figure 1.1. The horizontal projection of a valvular conduit (Tesla, 1920)
Figure 1.2 Flow chart of study
Figure 2.1. Flow patterns around a bluff body7
Figure 2.2. Flow structure of pipe flow with a contracting cross-section (Haase, 2017)
Figure 2.3. Loss coefficient for the angle of an enlarging cross-section in pipe flow
(Gibson, 1930) 10
Figure 2.4. Illustration of flow in the Tesla valve; (a) blocking directional flow, (b)
unimpeded directional flow (Cmglee)12
Figure 2.5. Illustration of computational cells in the VOF method and volume
fractions of water and air
Figure 3.1. The procedure of numerical modeling with OpenFOAM
Figure 3.2. Execution time for different numbers of cores
Figure 3.3. ARA00's figure in FPIL's server room
Figure 3.4. Domain and boundary illustrations of preliminary simulation
Figure 3.5. Domain illustration of a simulation case: A30_D6_S2.5(C)30
Figure 3.6. Illustration of the Hyeong-san river; (a) Satellite image map to measure
the river width (Google Earth), (b) Moving path of ADCP to measure the river

depth
Figure 3.7. Numerical mesh illustration of Draper's model (Draper, 2011)
Figure 3.8. Domain illustration of the main numerical model; (a) horizontal projection,
(b) schematic
Figure 3.9. Numerical mesh illustration of the channel domain of the numerical model;
(1) a downstream zone, (2) a structure installation zone, (3) an upstream zone,
(4) a relaxation zone
Figure 3.10. Domain and boundaries of the main numerical model
Figure 3.11. Numerical mesh illustrations of entire domains for the grid convergence
test; (a) case 1, (b) case 2, (c) case 3, (d) case 4
Figure 3.12. Photo of the digital water gauge
Figure 3.13. Photos of Vectrino Profiler; (a) a body and a stem fixed out of the water,
(b) four transducers in the water
Figure 3.14. Experimental domains and surface level & velocity profile measuring
points; (a) S3_C, (b) S7_C, (c) S3_E, (d) S7_E51
Figure 3.15. Photos of the experimental setup; (a) S3_E, (b) S7_C, (c) S7_E 52
Figure 3.16. Domain illustration of a simulation case: S7_W0.3_C_L5/6_w0.1_y0.2
Figure 4.1. Comparison of the total outflow volume in 600 seconds for different
angles between the wall and the structures
Figure 4.2. Comparison of the difference in the total outflow volume in 600 seconds

wall and the structures
Figure 4.3. Illustrations of the velocity fields at $z = 0.2$ (m) in the preliminary model;
(a) A30(C), (b) A60(C), (c) A60(E), (d) A30(E)61
Figure 4.4. Illustrations of the streamlines in the preliminary model; (a) A30(C), (b)
A60(C), (c) A60(E), (d) A30(E)61
Figure 4.5. Comparison of the total outflow volume in 600 seconds for different
spacings between the main structures
Figure 4.6. Comparison of the difference in the total outflow volume in 600 seconds
between contracting and expanding structures for different spacings between the
main structures
Figure 4.7. Comparison of the total outflow volume in 600 seconds for different
numbers of the structures
Figure 4.8. Comparison of the difference in the total outflow volume in 600 seconds
between contracting and expanding structures for different numbers of the
structures
Figure 4.9. The horizontal projection of a unit design of the hydraulic structures 68
Figure 4.10. Flow velocity at various points in the channel domain for different
numbers of cells; (a) $t = 0.5Tm$, (b) $t = 0.75Tm$
Figure 4.11. Time series of discharge in the downstream zone of the channel domain
for different numbers of cells
Figure 4.12. Comparison of the surface levels at different points in laboratory

experiments to those in CFD simulations for all experiment cases; (a) S3_C_Q36,

(b) S3_C_Q72, (c) S3_E_Q36, (d) S3_E_Q72, (e) S7_C_Q36, (f) S7_C_Q72,
(g) S7_E_Q36, (h) S7_E_Q72
Figure 4.13. Comparison of velocity profiles in laboratory experiments to those in
CFD simulations; (a) S3_C_Q36, (b) S3_C_Q72, (c) S3_E_Q3674
Figure 4.14. Time series of discharges through the inlet boundary of the basin and in
zone 1 of channel
Figure 4.15. The total volumes of water going downstream and upstream for different
numbers of structures
Figure 4.16. The difference between the total volumes of water going downstream
and upstream for different numbers of structures
Figure 4.17. The difference between the total volumes of water going downstream
and upstream for different numbers of structures relative to the difference for
case S0
Figure 4.18. Illustrations of the velocity fields in the channel domain for different
numbers of the structures; (a) S3, (b) S5, (c) S7, (d) S9 at $t = 0.5Tm$ (going
downstream); (e) S3, (f) S5, (g) S7, (h) S9 at $t = Tm$ (going upstream) 83
Figure 4.19. The total volumes of water going downstream and upstream for different
sub-structure conditions
Figure 4.20. The difference between the total volumes of water going downstream
and upstream for different sub-structure conditions relative to the difference for
case S0

sub-structure conditions; (a) w0.1_y0.1, (b) w0.1_y0.2, (c) w0.2_y0.1, (d)
w0.2_y0.2 at $t = 0.5Tm$ (going downstream); (e) w0.1_y0.1, (f) w0.1_y0.2, (g)
w0.2_y0.1, (h) w0.2_y0.2 at $t = Tm$ (going upstream)
Figure 4.22. The total volumes of water going downstream and upstream for different
main structure conditions
Figure 4.23. The difference between the total volumes of water going downstream
and upstream for different main structure conditions relative to the difference
for case S0
Figure 4.24. Illustrations of the velocity fields in the channel domain for different
main structure conditions; (a) W0.2_NC, (b) W0.3_NC, (c) W0.2_C, (d)
W0.3_C at $t = 0.5Tm$ (going downstream); (e) W0.2_NC, (f) W0.3_NC, (g)
W0.2_C, (h) W0.3_C at $t = Tm$ (going upstream)
Figure 4.25. The total volumes of water going downstream and upstream for different
offsets of the structure design94
Figure 4.26. The difference between the total volumes of water going downstream
and upstream for different offsets of the structure design
Figure 4.27. Illustrations of the velocity fields in the channel domain for different
offsets of the structure design; (a) L0, (b) L1/6, (c) L3/6, (d) L5/6 at $t = 0.5Tm$
(going downstream); (e) L0, (f) L1/6, (g) L3/6, (h) L5/6 at $t = Tm$ (going
upstream)95
Figure 4.28. Domain illustration of the best design of the hydraulic structures:
S7_W0.3_C_L1/6_w0.1_y0.296

Figure A.1. (a) A90, (b) A60(C), (c) A45(C), (d) A30(C), (e) A60(E),	(f) A45(E), (g)
A30(C)	
Figure A.2. (a) A90, (b) A60(C), (c) A45(C), (d) A30(C), (e) A60(E),	(f) A45(E), (g)

A30(C)	106
--------	-----

List of Tables

Table 2.1. Previous studies on the parameters of the Tesla valve 14
Table 3.1. ARA00's detailed specifications 22
Table 3.2. Scale and grid sizes of the domain for preliminary simulation
Table 3.3. Physical properties and turbulence model for preliminary simulation26
Table 3.4. Initial & boundary conditions for preliminary simulation
Table 3.5. Numerical schemes for preliminary simulation 27
Table 3.6. Detailed structure conditions for preliminary simulation
Table 3.7. Preliminary simulation cases
Table 3.8. Types of harmonic constants in the Hyeong-san river and their tidal
amplitudes and period observed at Pohang tidal station
Table 3.9. Adapted prototype hydraulic parameters
Table 3.10. Hydraulic parameters' orders, prototype values, and model values 34
Table 3.11. Wave parameters in the main numerical model
Table 3.12. Initial & boundary conditions for the main simulation 40
Table 3.13. Simulation cases for the grid convergence test 44
Table 3.14. Experiment conditions using Vectrino Profiler
Table 3.15. Experimental conditions for model validation 50
Table 3.16. Experiment cases for model validation 50
Table 3.17. Detailed structure conditions for the main simulation 56

Table 3.18. Main simulation cases	57
Table 4.1. Grid convergence index for all test cases	70
Table 4.2. Root-mean-square errors of the simulation results to the experiment	t results
for the free surface level	72
Table 4.3. Root-mean-square errors of the simulation results to the experiment	t results
for velocities at different points	73

List of Symbols

Latin Uppercase

A_m	tidal amplitude in the main model
A_p	tidal amplitude in the prototype
C_f	skin friction coefficient
Co	Courant number
C _{o,max}	maximum Courant number
C_{μ}	model coefficient
D_m	side length of the basin in the main model
<i>F</i> ₁	function for $k - \omega$ model
F _s	safety factor
Fr	Froude number
Н	order of vertical length
H ₀	initial water depth in the laboratory experiment
H _m	height of the entire domain in the main model

H_z	height of the channel in the preliminary model
K _L	loss coefficient
L	order of longitudinal/transverse length
L_m	length of the channel in the main model
L_{x}	length of the channel in the preliminary model
L_{xx}	length of the geometry
Ν	number of computational cells
N _p	number of points
N_x	number of computational cells in the x -direction
Q	flow discharge
Q_0	initial inflow discharge
Re _x	critical Reynolds number
Т	order of time
T_m	tidal period in the main model
T_p	tidal period in the prototype
U	flow velocity
U ₀	initial flow velocity

$U_{i,p}$	flow velocity at <i>p</i> th point in the case <i>i</i>
U _m	flow velocity in the main model
U _n	velocity normal to the boundary
Up	flow velocity in the prototype
U _x	flow velocity in the x -direction
Uy	flow velocity in the y-direction
Uz	flow velocity in the z-direction
V	total water volume
V ₀	reference volume
V_0^{down}	total volume of water going downstream in case S0
V_0^{up}	total volume of water going upstream in case S0
V _{out}	total outflow volume
V _{out} ^C	total outflow volume in the contracting direction
V_{out}^E	total outflow volume in the expanding direction
V ^{down}	total volume of water going downstream
V^{up}	total volume of water going upstream

ΔV	total volume difference between the downstream and upstream
	flow
ΔV_0	total volume difference between the downstream and upstream
	flow in case S0
ΔV^r	total volume difference between the downstream and upstream
	flow relative to the total volume difference in case S0
W _m	width of the channel in the main model
W_p	width of the channel in the prototype
$W_{\mathcal{Y}}$	width of the channel in the preliminary model

Latin Lowercase

- *d* discretization order
- $f_{\sigma i}$ surface tension
- g gravitational acceleration
- *h* water depth
- h_0 initial water depth in the preliminary model

n_m	water depth of the channel in the main model
h_p	water depth of the channel in the prototype
k	turbulent kinetic energy
k _m	wavenumber in the main model
ł	mixing length scale
n	normal direction
p	pressure
\overline{p}	mean pressure
r	grid refinement ratio
t	time
Δt	time step
u_T	friction velocity
ū	mean velocity
u'	velocity fluctuation
Δx	computational cell size in the x -direction
y_p	distance from the wall to the computational cell center
<i>y</i> ⁺	normalized distance from the wall to the computational cell center

Δy	computational cell size in the y-direction
Δz	computational cell size in z-direction

Greek Lowercase

α	volume fraction
α_{air}	volume fraction of air
α_w	volume fraction of water
β	model constant
β_1	constant for $k - \omega$ model
β_2	constant for $k - \epsilon$ model
β^*	model constant
γ	model constant
γ_1	constant for $k - \omega$ model
γ ₂	constant for $k - \epsilon$ model
E _{RMSRE}	root mean square relative error
η_m	wave displacement in the main model

κ	von-Karman constant
μ	dynamic viscosity
ν	kinematic viscosity
v _{air}	kinematic viscosity of air
ν _T	eddy viscosity
ν _w	kinematic viscosity of water
ρ	density
$ ho_{air}$	density of air
$ ho_w$	density of water
σ_k	model constant
σ_{k1}	constant for $k - \omega$ model
σ_{k2}	constant for $k - \epsilon$ model
σ_{ω}	model constant
$\sigma_{\omega 1}$	constant for $k - \omega$ model
$\sigma_{\omega 2}$	constant for $k - \epsilon$ model
τ	Reynolds-stress
$ au_w$	wall shear stress

φ_{out}	volume flux out of the domain
ϕ	model constant
ϕ_1	constant for $k - \omega$ model
ϕ_2	constant for $k - \epsilon$ model
ϕ_m	wave phase in the main model
ω	specific turbulent dissipation rate
ω_m	wave angular velocity in the main model

CHAPTER 1. INTRODUCTION

1.1 General introduction

The difference in water levels between the upstream and the downstream is the main cause of the open channel flow (e.g. river stream, an estuary). When the sea level rises due to tides, seawater flows upstream of the river and the tidal section appears. Additionally, the seawater intrudes to the upstream near bed due to density difference in a form of a salt wedge (Schijf & Schönfeld (1953)). Such flow appears in a wide range of rivers, and it causes adverse effects on the ecosystem and damages to agricultural land. Also, we cannot rule out the possibility of flooding with the rise of sea level.

To avoid damage, the measure blocking the seawater flow is necessary; a dam is an example of a hydraulic structure controlling bidirectional flows. A hydraulic structure completely separates upstream and downstream so that seawater cannot flow upstream and it secures sufficient upstream water level that alleviates flooding or drought damage. However, it obstructs all the materials and energy from downstream so the free movement of organisms living in the river or ocean is completely blocked. Also, water quality gets worse due to the stagnant pollutants near dams with the rise of the water level. A fishway enables fish to go upstream to some extent, but it cannot be the ultimate solution to the problem. Furthermore, the government of the Republic of Korea has opened 13 barrages of the 16 barrages on the four major rivers since June 2017 and announced the plan for the demolition of several barrages. Therefore, the alternatives of the barrages or estuary banks should be required to prevent the seawater to flow upstream.

Because the adverse effects of the estuary banks are mostly attributed to their vertical walls separating upstream and downstream, a new structure design is needed to control the flow restricting horizontally. With such a design, we can expect the effects of controlling energy with the free exchange of materials. The key point is that the structures obstruct the seawater from downstream at most while releasing the freshwater from upstream. Such a mechanism can be found in the human body, the cardiac valves of the heart. The heartbeat makes the circulation of blood all over the body with the pressure difference between the atria and ventricles. Normal blood flow causes when the pressure in an atrium is higher than in a ventricle; otherwise, the valves do not allow it to flow backward by inducing unidirectional flow with their geometries. Almost 100 years ago, Nicola Tesla designed a valvular conduit, the Tesla valve, by grafting the idea of the cardiac valves' geometry to control the flow with on microfluidic conduit (Figure 1.1) (Tesla, 1920). Tesla valve consists of a main straight channel and several curved flow-control stages, and the principle of the valve will be described in **2.2**. These two examples of flow control by horizontal geometry are generally applied to a pressure-driven flow in which the length scale is in millimeters or centimeters. In the study, the structures are applied to an open channel such as a river estuary by expanding scale up to meters or kilometers, and analogical function with the Tesla valve is anticipated to obstruct the flow from downstream.

To verify the effectiveness of the structures, simulations were conducted using computational fluid dynamics (CFD) modeling using OpenFOAM. The software solved continuity equations for mean velocities and Reynolds-averaged Navier-Stokes (RANS) equations to analyze 3-dimensional (3-D) hydrodynamic behaviors, and the free surface was handled by the volume of fluid (VOF) method (Gopala & van Wachem, 2008). The preliminary study was on the straight channel model, and the main study was on the model describing the coastal area with a basin and a channel. Numerical modeling was on the laboratory scale, so distorted scale analysis with Froude similarity law was carried out to determine the geographic and hydraulic parameters of the model. To implement an ideal model similar to the real case, the simulation domain and initial & boundary conditions should be appropriately set up. The numerical model was validated by the laboratory experiments with a comparison of the free surface levels and velocity profiles of the numerical and experimental results. In the model, simulations were conducted on various conditions with different shapes of the structures and defined the structure design as the case that shows up the greatest difference in discharges between bidirectional flows.



Figure 1.1. The horizontal projection of a valvular conduit (Tesla, 1920)

1.2 Objectives

The study's main purpose is to determine the design of the hydraulic structures to maximize the flow asymmetry in bidirectional flows based on numerical simulations. To achieve the main purpose, the preliminary study and the main study proceeded in sequence, with three detailed steps. The general procedure of the study is shown in **Figure 1.2**.

First, the preliminary study was conducted to check whether the hydraulic structures function to obstruct the specific directional flow. It was evaluated by comparing the outflow discharge with the structures of that without the structures. The preliminary CFD simulations were in the straight-channel model, and the simulation domain and boundary conditions were simply set up. Moreover, a unit design of the structures was decided by simulations for different structure configuration conditions. The unit design was used as the base of the structure design in the main simulation.

Second, the main numerical model was developed. The main study was to find out the hydraulic structure design to maximize the flow asymmetry in the open channel flow where the bidirectional flows occur. The simulation domain was designed with a large basin and a channel to describe the coastal area realistically. Scale analysis was applied to develop a distorted model with Froude similarity. Initial & boundary conditions and numerical methods were carefully implemented for stable bidirectional free surface flows including wave motions. A grid convergence test decided on the reasonable numerical mesh, and physical modeling with laboratory experiments validated the numerical model with a comparison of the free surface levels and the velocity profiles between the results of the two models.

Last, CFD simulations were conducted for different structure configuration conditions installed in the channel domain of the main model. Simulation cases were designed based on the unit structure design found in the preliminary study. The best design was determined as the case with the maximum discharge difference in bidirectional flows. In the design process, the analysis of the hydraulic structures' effects on the flow structure also played an important role.



Figure 1.2 Flow chart of study

CHAPTER 2. THEORETICAL BACKGROUNDS

2.1 Flow structure

2.1.1 Wake regions with eddies

When fluid flows around the bluff body, complicated flow patterns occur in the wake region such as flow separation, reattachment, and vortex generation (**Figure 2.1**) (Roumeas, 2009). The wake region generally appears behind the bluff body where the flow is reversible and rotational, so the main flow is disturbed. The primary cause of the wake regions is viscosity, so it leads to energy dissipation. To maintain the eddies in a wake region, energy should be supplied from the main flow continuously and it converts to eddies' kinetic energy. In conclusion, fluid loses more energy when it passes by the bluff body than it does not, and such energy dissipation causes discharge loss.



Figure 2.1. Flow patterns around a bluff body

2.1.2 Energy loss in pipe flow

It is crucial to consider the flow structure including wake regions and energy dissipation in the open channel, but its theories are not well-established. Instead, the pipe flow analysis was referred to in the study because of its well-known principles. Although some hydraulic characteristics of open channel flow and pipe flow (e.g. driving force, main cause of energy loss) are different, relations between the flow and the wake regions are fairly similar. Therefore, understanding the principles of pipe flow might be helpful to design hydraulic structures in the open channel with several decisive factors.

In pipe flow, local loss occurs with a contraction or enlargement of the crosssection of the pipe. When the fluid flows through a contracting cross-section, it accelerates before passing the cross-section and decelerates after passing it. Also, the fluid's cross-section contracts after passing the pipe entrance, and it expands right away. Then, turbulence zones show up near the entrance and produce local loss (**Figure 2.2**). For an enlarging cross-section, fluid decelerates and produces larger eddies behind the pipe entrance than in the contracting one, then it makes a local loss. Such a local loss at two types of a cross-section of the pipe is associated with the loss coefficient, K_L , which was determined by experiments (Robert et al., 1996). K_L is about one for an enlargement of pipe and much smaller for a contracting one. According to this, it is likely that an expanding cross-section of open channel flow causes more energy dissipation by producing larger eddies. Then, hydraulic structures can obstruct the flow from the coastal sea by narrowing the channel width. Furthermore, the shape of the pipe entrance is also an important factor in the local loss. For a contracting entrance, the local loss is much smaller with a smooth pipe entrance than with an abrupt entrance. This is because a smooth one mitigates the contraction of a cross-section of fluid so that it produces less (Ito, 1960). For an enlarging entrance, the loss coefficient changes with an enlarging angle of a cross-section. According to Gibson, it has a maximum coefficient when the angle is about 60° (Figure 2.3) (Gibson, 1930). There is no guarantee that such a value of an angle is the best for an open channel flow too, but it is reasonable to speculate that an angle is a considerable factor to improve the performance of the hydraulic structures.



Figure 2.2. Flow structure of pipe flow with a contracting cross-section (Haase, 2017)



Figure 2.3. Loss coefficient for the angle of an enlarging cross-section in pipe flow (Gibson, 1930)

2.2 Tesla valve

2.2.1 Characteristics of the Tesla valve

As mentioned in **1.1**, the Tesla valve is a conduit to control bidirectional flow which consists of the main channel and several curved stages. For different directions of the flow, different flow patterns form in the stages due to their geometric structures which resist or assist the main flow. For an unimpeded directional flow, fluid in the main channel (blue arrows in **Figure 2.4**) scarcely gets into the stages and is hardly affected by flow in the stages (**Figure 2.4(b)**). Even if the fluid gets into the stages, it towards the same direction as the fluid in the main channel so it rather assists the main flow. On the contrary, for a blocking directional flow, a large proportion of the fluid in the main channel (black arrows) gets into the stages and forms the circular flow (red arrows) (**Figure 2.4(a)**). The main flow towards the opposite direction to the flow going out of the stages. Then, the main and the circular flow counter each other at the end of the stages and produce the vortex which dissipates the kinetic energy into the heat.

It is quite a hard task to adopt a large-scale of the original Tesla valve with renovating estuary, but it would not be exacting with a more simplified design. Applying the structures controlling the flow by incompatible mechanisms depending on the flow direction, we can expect the effects of the structures on the bidirectional flows without the problems attributed to the separation of upstream and downstream.



Figure 2.4. Illustration of flow in the Tesla valve; (a) blocking directional flow, (b) unimpeded directional flow (Cmglee)

2.2.2 Parameters of the Tesla valve

Parameter determination and optimization are the most important to design a hydraulic structure. Tesla valve is a good object to research a microfluidic structure in a bidirectional flow, so the geometric parameters have been investigated for better performance to direct the flow. Truong and Nguyen (2003) and Gamboa et al. (2005) optimized the shape and size of the Tesla valve in the two-dimensional (2-D) steady flow based on the numerical method. Mohammadzadeh et al. (2013) investigated the number of stages of the Tesla valve on its performance in 2-D steady and unsteady flows, and Thompson et al. (2014) found the most efficient number and spacing of

stages in the three-dimensional (3-D) steady flow. Numerical studies on hydrogen decompression using the Tesla valve were conducted by Jin et al. (2018) and Qian et al. (2019) in the 3-D steady flow. Most studies on the Tesla valve are conducted in microscale and pipe flow models, so the pressure has been investigated as a crucial factor. Diodicity is defined as the ratio of pressure drop in backflow to the pressure drop in forward flow at the same flow rate, and it is a common indicator to evaluate the efficacy of the Tesla valve.

However, the present study is on an open channel and pressure is not a significant factor, so something new indicator should be proposed to evaluate the efficiency of hydraulic structures. Dennai et al. (2016)'s study on the performance of the microscale pressure-driven Tesla valve was evaluated by the flow discharge difference depending on the length of the stage. Keizer (2016) applied the large-scale Tesla valve to the open channel flow and investigated its applicability with parameters regardless of the pressure drop by laboratory experiments. The present study investigates the new hydraulic structures on the large-scale open channel in the 3-D steady and unsteady flows by numerical method. The investigation was on the geometric parameters of the structures and the performance of the structures was evaluated by the discharge difference in bidirectional flows. Overall studies on the Tesla valve are summarized in **Table 2.1**.
Reference	Methods	Flow Conditions	Geometric Parameters	Performance
Truong & Nguyen (2003)	Numerical	2-D steady flow	Length of a stage Angle of a stage	Diodicity
Gamboa et al. (2005)	Numerical	2-D steady flow	Width of channel Length of channel Radius of inner curve Angle of a stage	Diodicity
Mohammadzadeh et al. (2013)	Numerical	2-D steady, unsteady flow	Number of stages Width of channel	Diodicity
Thompson et al. (2014)	Numerical	3-D steady flow	Number of stages Stage-to-stage distance	Diodicity
Dennai et al. (2016)	ennai et al. (2016) Numerical 2-D steady flow		Length of internal wall	Flowrate
Keizer (2016)	Keizer (2016) Experimental Steady flow		Number of stages Length of stages	Energy loss Flow velocity Water depth
Jin et al. (2018)	Jin et al. (2018) Numerical 3-D steady flow		Hydraulic diameter Radius of inner curve Angle of a stage	Pressure drop
Qian et al. (2019)	Qian et al. (2019) Numerical 3-D steady flow		Number of stages	Temperature Pressure drop Flow velocity

 Table 2.1. Previous studies on the parameters of the Tesla valve

2.3 Governing equations

2.3.1 Reynolds-averaged Navier-Stokes equations

Navier-Stokes equations are generally employed to analyze the flow motion. Flow structures with separations and wakes are considerable factors in the study and they can be figured out for the mean motion of the flow, so the Reynolds-averaged Simulations (RAS) model is adequate to describe such flow characteristics. In the RAS model, continuity equations for mean velocities and Reynolds-averaged Navier-Stokes equations are the governing equations which are described as

$$\frac{\partial \overline{u_i}}{\partial x_i} = 0, \tag{1a}$$

$$\frac{\partial(\rho\overline{u_i})}{\partial t} + \frac{\partial(\rho\overline{u_i}\,\overline{u_j})}{\partial x_j} = -\frac{\partial\overline{p}}{\partial x_i} + \rho g_i + \mu \frac{\partial^2\overline{u_i}}{\partial x_j^2} + \frac{\partial\tau_{ij}}{\partial x_j},\tag{1b}$$

where $\tau_{ij} = -\rho \overline{u'_i u'_j}$ is Reynolds-stress. To solve the equations, $k - \omega$ SST model is employed for the turbulence closure model to estimate τ_{ij} by calculating v_T with k and ω , which is described as

$$\frac{\partial(\rho k)}{\partial t} + \overline{u_j} \frac{\partial(\rho k)}{\partial x_j} = \tau_{ij} \frac{\partial \overline{u_i}}{\partial x_j} - \beta^* \rho k \omega + \frac{\partial}{\partial x_j} \left[(\mu + \rho \sigma_k \nu_T) \frac{\partial k}{\partial x_j} \right], \tag{2a}$$

$$\frac{\partial(\rho\omega)}{\partial t} + \overline{u_j}\frac{\partial(\rho\omega)}{\partial x_j} = \frac{\gamma}{\nu_T}\tau_{ij}\frac{\partial\overline{u_i}}{\partial x_j} - \beta\rho\omega^2 + \frac{\partial}{\partial x_j}\left[(\mu + \rho\sigma_\omega\nu_T)\frac{\partial\omega}{\partial x_j}\right] + 2\rho(1 - F_1)\sigma_{\omega 2}\frac{1}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial\omega}{\partial x_j},$$
(2b)

where β^* , β , σ_k , σ_{ω} , γ are constant. For any constant ϕ , it can be calculated by

$$\phi = F_1 \phi_1 + (1 - F_1) \phi_2, \tag{3}$$

where F_1 is a function designed to be one in the near wall and zero away from the surface. In other words, ϕ_1 represents the constant for the original $k - \omega$ model and ϕ_2 represents the constant for the $k - \epsilon$ model. Each constant of the two models is used as

$$\beta^{*} = 0.09, \ \beta_{1} = 0.0750, \ \beta_{2} = 0.0828,$$

$$\sigma_{k1} = 0.5, \ \sigma_{k2} = 1.0, \ \sigma_{\omega 1} = 0.5, \ \sigma_{\omega 2} = 0.856,$$

$$\gamma_{1} = \frac{\beta_{1}}{\beta^{*}} - \frac{\sigma_{\omega_{1}}\kappa^{2}}{\sqrt{\beta^{*}}}, \ \gamma_{2} = \frac{\beta_{2}}{\beta^{*}} - \frac{\sigma_{\omega_{2}}\kappa^{2}}{\sqrt{\beta^{*}}},$$
(4)

where κ is the von-Karman constant (=0.41) (Menter, 1993).

2.3.2 Free surface description

To numerically analyze the free surface flow in the open channel, the volume of fluid (VOF) method is widely applicable. The method is implemented for the finite volume method so it is advantageous to calculate the volume fractions of two fluids in the computational grids and to describe the interface of a multiphase of fluids (Hirt & Nichols, 1981). Before solving RANS equations in the computational grids including interface, the density, and kinematic viscosity are calculated with the volume fractions of the fluids with the following equations:

$$\rho = \rho_w \, \alpha_w + \rho_{air} \, \alpha_{air}, \tag{5a}$$

$$\nu = \nu_w \, \alpha_w + \nu_{air} \, \alpha_{air}, \tag{5b}$$

where α_w , α_{air} , ρ_w , ρ_{air} , v_w , v_{air} are volume fraction of water and air, the density of water and air, kinematic viscosity of water and air, respectively. Also, the surface tension should be considered to analyze the motion of the interface. Then, RANS equations are described as

$$\frac{\partial(\rho\overline{u_i})}{\partial t} + \frac{\partial(\rho\overline{u_i}\,\overline{u_j})}{\partial x_j} = -\frac{\partial\overline{p}}{\partial x_i} + \rho g_i + \mu \frac{\partial^2\overline{u_i}}{\partial x_j^2} + \frac{\partial\tau_{ij}}{\partial x_j} + f_{\sigma i},\tag{6}$$

where $f_{\sigma i}$ is surface tension. Additionally, the volume fraction of each phase is calculated by mass conservation law described as

$$\frac{\partial \alpha_i}{\partial t} + \frac{\partial (\alpha_i \overline{u_j})}{\partial x_j} = 0, \qquad i = water, air$$
(7)

The sum of volume fractions of two fluids is constant 1. If the computational cell is filled with water only, $\alpha_w = 1$ and $\alpha_{air} = 0$ (Figure 2.5). The free surface is regarded as located where the volume fraction of water is 0.5.



Figure 2.5. Illustration of computational cells in the VOF method and volume fractions of water and air

CHAPTER 3. METHODOLOGY

3.1 Numerical methods

3.1.1 Numerical model description

Several methods are employed to figure out the flow characteristics like analytical solutions, numerical or physical modeling, and field observation. In this study, we conducted numerical modeling by CFD simulation with OpenFOAM which is an open-source CFD software. To start with OpenFOAM, a specific solver is primarily selected which is suitable to model conditions among a plethora of solvers. Then numerical mesh is generated with reasonable grid sizes, and physical properties (initial & boundary conditions, density, and kinematic viscosity of transport materials) and numerical techniques (turbulence model, time step, simulation time, numerical schemes, equation solvers) are set up. The additional meshes also could be adopted in the simulation domain by importing external object files, and injecting a wave into the domain is possible by installing an additional library, *waves2Foam*. Finally, the software solves governing equations with setup conditions. Such a procedure of numerical modeling with OpenFOAM is described in **Figure 3.1**.



Figure 3.1. The procedure of numerical modeling with OpenFOAM

3.1.2 Computing equipment

The study used OpenFOAM version 4.0 as CFD software, and GSL (GNU Scientific Library) version 2.7.1 was used for scientific computing. To satisfy the software performance for numerical experiments, the study was carried out in the cluster server in Flow Physics and Informatics Laboratory (FPIL) at Seoul National University, which is named ARA00. Its operation system is centOS Linux version 6.8, and it used GCC version 4.9.4 as a compiler. Parallel computing with MPICH version 1.10.2 could significantly reduce the execution time. To optimize the number of cores of the server, parallel testing was carried out. The simulation time was set up to 60 seconds. As a result, the 16 cores showed the minimum execution time, so all simulation cases were conducted with 16 cores (Figure 3.2). ARA00's detailed specification is described in Table 3.1, and its figure is shown in Figure 3.3.



Figure 3.2. Execution time for different numbers of cores

Table 3.1. ARA00's detailed specificatio	ns
--	----

	Intel E5 2680 v4 14 Core 2.4 GHz 35MB Cache × 2 P
DELL	128 GB DDR4 2400 (16 EA × 8 GB)
R730	300 GB SAS 15 K Disk × 2 EA (Mirror)
	8 TB SAS 7.2 K Disk × 4 EA (Data)
OS	CentOS Linux version 6.8
Compiler	GCC version 4.9.4
Parallel Computing	MPICH version 1.10.2



Figure 3.3. ARA00's figure in FPIL's server room

3.2 Preliminary CFD modeling

3.2.1 Simulation domain

In advance of conducting the main study, it is required to verify the effectiveness of hydraulic structures in making a discharge difference in bidirectional flows. Moreover, it could enhance the efficiency of finding the best design for the structures by suggesting a unit design in this step. Preliminary CFD simulations were conducted on a simplified, straight, open-channel flow model to achieve those goals. The domain consists of boundaries of an inlet, an outlet, confined side walls, an atmospheric boundary, and a bottom (**Figure 3.4**). The length of the channel (L_x) was determined not to reach the outlet boundary effect on the wakes generated by the structures. The width (W_y) and height of the channel (H_z) are arbitrarily defined, and the initial depth (h_0) of the channel is set up as half of H_z . The grid sizes ($\Delta x, \Delta y, \Delta z$) were roughly set up to balance the accuracy and economy of the model. The ratio of the length, width, and height of the channel was reflected to set up the grid sizes, and the height of the grid size was decided smaller to treat the free surface more precisely. Those scales are described in **Table 3.2**.



Figure 3.4. Domain and boundary illustrations of preliminary simulation

Table	2 2	Cala	1	1		- f +1	1	£		::			1.4
Table	J.Z.	Scale	and	gria	sizes	of the	domain	IOT	pre	Im	nary	simu	iation
				\mathcal{L}					1		2		

<i>L_x</i> [m]	<i>W</i> _y [m]	<i>H</i> _z [m]	<i>h</i> 0 [m]	Δx [m]	Δy [m]	Δz [m]
24	1	0.8	0.4	0.03	0.025	0.02

3.2.2 Simulation conditions setup

In describing the open channel flow in CFD modeling, two phases of air and water should be included in the domain, which is called multiphase flow. In the study, we selected *interFoam* as a solver which is generally selected for modeling the multiphase flow in OpenFOAM. It is based on the VOF method and applicable to solve the motion equations of two immiscible phases of incompressible and isothermal fluids. The density and kinematic viscosity of fluids were determined at a temperature of 298K, and $k - \omega$ SST model was chosen for the turbulence model (**Table 3.3**). To implement a stable model, constant discharge and velocity were suitable for inlet and outlet boundary conditions, respectively. Constant pressure was applied to the atmospheric boundary of the domain, and a no-slip condition and wall function were applied to the side walls and bottom. To start with the $k - \omega$ SST model, the initial values of k and ω should be determined by the following equations:

$$k = 1.5 \times (0.05|U_0|)^2, \tag{8}$$

$$\omega = \frac{k^{0.5}}{C_{\mu}^{0.25}\ell'}$$
(9)

where U_0, C_{μ}, ℓ are initial flow velocity, model coefficient (=0.09), and mixing length scale, respectively. The Neumann condition is applied to the other boundary conditions with zero gradients. Initial and boundary conditions are described in **Table 3.4**. Numerical schemes and solutions were set up for what were widely employed in interFoam which are described in Table 3.5.

The Courant number (C_o) is a criterion of time step (Δt) in CFD. Applying the Euler scheme to the first time-derivative terms in 3-dimensional analysis, C_o follows

$$C_o = \Delta t \left(\frac{U_x}{\Delta x} + \frac{U_y}{\Delta y} + \frac{U_z}{\Delta z} \right), \tag{10}$$

where U_x, U_y, U_z are flow velocity in x-direction, y-direction, and z-direction, respectively. In the study, the maximum Courant number was set up to 1 ($C_{o,max} = 1$), so the time step was calculated by the following equation:

$$\Delta t = \frac{C_{o,max}}{\left(\frac{U_x}{\Delta x} + \frac{U_y}{\Delta y} + \frac{U_z}{\Delta z}\right)} = \frac{1}{\left(\frac{U_x}{\Delta x} + \frac{U_y}{\Delta y} + \frac{U_z}{\Delta z}\right)}.$$
(11)

The total simulation time was set up to 1200 seconds, and only the last 600 seconds were considered in the analysis because it took enough time for the model to be stable.

Wa	ter	A	Nir	Taabalaas
$ ho_w \ \left[kg/m^3 ight]$	$rac{ u_w}{\left[m^2/s ight]}$	$ ho_{air} \ [kg/m^3]$	v_{air} $[m^2/s]$	model
1000	1×10^{-6}	1	1.48×10^{-5}	$k - \omega$ SST model

Table 3.3. Physical properties and turbulence model for preliminary simulation

	Inlet	Wall	Atmosphere	Outlet
U [m/s]	$Q_0 = 0.05$ (m^3/s)	No slip	$\frac{\partial U}{\partial n} = 0$	0.125
p [Pa]	$\frac{\partial p}{\partial n} = 0$	$\frac{\partial p}{\partial n} = 0$	0	$\frac{\partial p}{\partial n} = 0$
k $[m^2/s^2]$	5.86×10^{-5}	Wall function	$\frac{\partial k}{\partial n} = 0$	5.86× 10 ⁻⁵
ω $[s^{-1}]$	1.16 × 10 ⁻³	Wall function	$\frac{\partial \omega}{\partial n} = 0$	1.16×10^{-3}
$rac{ u_T}{[m^2/s]}$	Calculated	Wall function	Calculated	Calculated

Table 3.4. Initial & boundary conditions for preliminary simulation

 Table 3.5. Numerical schemes for preliminary simulation

Time derivative	Gradient	Divergence	Laplacian	Cell-to-face interpolation
Euler	Linear	Linear / Upwind / VanLeer	Linear corrected	Linear

3.2.3 Simulation cases setup

Hydraulic structures are installed in pairs in the simulation domain by importing external meshes generated in the application Blender. The structures consist of two types of structures, the main structure, and the sub-structure. The main structures are attached to the side walls of the straight channel, and each one's transverse length is equally set to $0.2W_y$. Sub-structures are installed away from the side walls between the main structures and have a half-length of the main structure. Unlike the main structure, sub-structures were necessary to generate circular flows which significantly disturbed the main flow like the Tesla valve. To achieve the goal of the preliminary study, numerical simulations were conducted for different structure conditions: 1) an angle between the wall and structures, 2) a spacing between the main structures, and 3) the number of structures.

First, based on the head loss in the pipe flow, analogous effects that differ from the contracting or enlarging cross-section and its angle were expected in the open channel flow. Therefore, the effectiveness of the structures was verified depending on their directions and the angle with the side walls. Second, the spacing of the main structures was regarded as a considerable factor to affect the flow rate because the size of generated wakes at the back of the structures might be limited by the other structure. Similarly, the number of structures was considered to constrain the expansion of the wakes. Also, it assumed that the presence of the sub-structures was helpful to improve the effects of the structures. To compare the outflow discharge in a bidirectional flow in a single case, simulations were conducted for both the contracting and expanding

directions of the structures. Detailed structure conditions are described in **Table 3.6**. Except for the flow direction, each symbol of the condition consists of a letter and a number, which indicate the type and value of the condition, respectively. Also, the capital and small letters indicate the main structure and sub-structure, respectively. All simulation cases were set up by combining structure conditions (**Table 3.7**). To help understand the expression in the table, **Figure 3.5** describes A30_D6_S2.5(C) in detail. After simulations, the best unit structure design was determined by comparing the total outflow volume in 600 seconds between the contracting and expanding cases.

An angle between the wall and the structures	A90 A60
(A#)	A45 A30
The spacing of the main structures (D#)	D3 D6 D9
The number of structures (S#)	S2 S2.5* S3
The direction of the structures	C (contracting)
(C / E)	E (expanding)

 Table 3.6. Detailed structure conditions for preliminary simulation

(#: the value of the conditions)

* S2.5: 2 pairs of the main structures + 1 pair of the sub-structures

Conditions	Simulat	ion cases
	А	90
An angle between the	A60(C)	A60(E)
wall and the structures	A45(C)	A45(E)
	A30(C)	A30(E)
	A30_D3(C)	A30_D3(E)
The spacing of the main structures	A30_D6(C)	A30_D6(E)
	A30_D9(C)	A30_D9(E)
	A30_D6_S2(C)	A30_D6_S2(E)
The number of structures	A30_D6_S2.5(C)	A30_D6_S2.5(E)
	A30_D6_S3(C)	A30_D6_S3(E)

Table 3.7. Preliminary simulation cases



Figure 3.5. Domain illustration of a simulation case: A30_D6_S2.5(C)

- A30: an angle between the wall and the structures is 30°
- D3: the spacing of the main structures is $6W_y$
- S2.5: 2 pairs of the main structures and 1 pair of the sub-structures
- C: the structures have a contracting cross-section

3.3 Main CFD modeling

3.3.1 Simulation domain

The purpose of the main CFD simulation is to find out the design of the hydraulic structures maximizing the discharge difference in bidirectional flows based on the unit design suggested in the preliminary simulation. This step aimed to design the simulation model with bidirectional flows more realistically, so it was specified as a coastal area consisting of a narrow river and a large ocean with a tide. Unlike the preliminary simulation, hydraulic parameters (e.g. width, depth, tidal characteristics) in the main model should be determined based on the observed data on the prototype object area. In the study, the Hyeong-san river was chosen for the object area. The river width was roughly measured and averaged in Google Earth, and the depth of the river was estimated by averaging the river depth data measured by FPIL (Figure 3.6). Also, it is available to use the observed tidal data in Pohang tidal station shown in Table 3.8. However, it was such an exacting task to consider all of the harmonic constants existing in the Hyeong-san river. To be simple, tidal amplitude and period were estimated as the sum of 4 amplitudes of harmonic constants and the longest tidal period among the constants, respectively. As a result, the parameters were adapted as
 Table 3.9 and such parameters were used as prototype parameters.



Figure 3.6. Illustration of the Hyeong-san river; (a) Satellite image map to measure the river width (Google Earth), (b) Moving path of ADCP to measure the river depth

H	armonic constant	Tidal semi-range [<i>m</i>]	Tidal Period [hr]
M2	Principal lunar semi-diurnal tide	3.1	12.42
S2	Principal solar semi-diurnal tide	0.7	12
K1	K ₁ constituent	4.2	23.93
01	Principal lunar diurnal tide	4.3	25.82

Table 3.8. Types of harmonic constants in the Hyeong-san river and their tidal amplitudes and period observed at Pohang tidal station

Width, W _p	Depth, h _p	Tidal amplitude, <i>A_p</i>	Tidal period, <i>T_p</i>
[m]	[m]	[<i>m</i>]	[<i>hr</i>]
360	2.5	0.125	24

Table 3.9. Adapted prototype hydraulic parameters

However, it was impossible to simulate the model with such large prototype parameters due to the tremendous amount of time and costs, so the scale of the numerical model should be reduced with the law of dynamic similarity. In most free surface flow, Froude similarity is applicable that requires the model's Froude number to have the same number as the prototype's one which is described as

$$F_r = \frac{U_m}{\sqrt{gh_m}} = \frac{U_p}{\sqrt{gh_p}},\tag{12}$$

where F_r , U_m , U_p , h_m , h_p are the Froude number, model flow velocity, prototype flow velocity, model water depth, and prototype water depth, respectively. In the Hyeong-san river with a measured velocity, F_r was about 0.04. Here, the river width is much larger than the river depth, and the longitudinal velocity is dominant in the river. In other words, the order of longitudinal length and that of transverse length are the same as *L*, but that of vertical length *H* is significantly smaller than *L*. Therefore, a distorted model was applied to the main model.

According to the model, the orders of the velocity, tidal amplitude, and tidal period are L/T, H, and T, respectively, where T is the order of the time. The model river length

was set as $L_m = 6 (m)$, the width as $W_m = 0.15 (m)$, and the depth as $h_m = 0.1 (m)$ to follow the laboratory scale. Then, the ratios W_p/W_m and h_p/h_m became 2400 and 25, respectively, and the model tidal amplitude (A_m) was calculated by the Froude similarity following as

$$A_m = A_p \times \frac{h_m}{h_p} = 0.005 \ (m) \tag{13a}$$

The model tidal period (T_m) was arbitrarily set up as 0.05 *hr*. Hydraulic parameters' scales, prototype, and model values are described in **Table 3.10**.

Parameter	Order	Prototype value	Model value
Width [m]	L	360	0.15
Water depth [m]	Н	2.5	0.1
Tidal amplitude [m]	Н	0.125	0.005
Tidal period [hr]	Т	24	0.05

Table 3.10. Hydraulic parameters' orders, prototype values, and model values

The main numerical model was designed to include a channel and a basin referring to Draper's coastal model (Draper, 2011) (**Figure 3.7**). While Draper's model included two large basins adjoined on both sides of a short channel, the present study's model tended to describe the general coastal area so it consisted of only one part of a large ocean and a river part. To generate a 1-dimensional wave from an ocean boundary, the shape of the basin was altered from Draper's semicircular one to the square one. The length of its sides was set up to $D_m = 20W_m$, and the height of the entire domain was uniformly $H_m = 1.5h_m$. The domain is illustrated in **Figure 3.8**. This form of the model has an advantage in boundary conditions. For a straight channel model, water depth is generally fixed to a constant value at an outlet boundary, but it cannot take into account the free surface fluctuation by reflective waves from hydraulic structures. Also, it causes an unphysical abrupt change in the free surface near the boundary. However, a large basin helps diminish the boundary effects on the channel so it is suitable to describe the model with wave motions more stable.



Figure 3.7. Numerical mesh illustration of Draper's model (Draper, 2011)



Figure 3.8. Domain illustration of the main numerical model; (a) horizontal projection, (b) schematic

For a systematic analysis, the channel domain was divided into four zones for the same length: a downstream zone (=zone 1), a structure installation zone (=zone 2), an upstream zone (=zone 3), and a relaxation zone (=zone 4) (Figure 3.9). First, a downstream zone is the nearest zone of the river to the ocean. When the water flows downstream to the ocean, it reaches the zone after passing the structures and being affected by them. The discharge was measured in this zone and compared to evaluate the performance of the structure design. A structure installation zone is where the structures are installed. The structures were designed only in this zone. An upstream zone is a zone further from the ocean than the structure installation zone. Although the flow goes through this zone before reaching the structures, the discharge in this zone also changes due to the structures. Finally, a relaxation zone is the farthest zone of the river from the ocean. In this zone, the waves from the ocean are damped or pass out without any reflection. This zone was not considered in the analysis.



Figure 3.9. Numerical mesh illustration of the channel domain of the numerical model; (1) a downstream zone, (2) a structure installation zone, (3) an upstream zone, (4) a relaxation zone

3.3.2 Simulation conditions setup

In the main numerical domain, bidirectional flows are generated by the wave into the ocean's boundary in a specific period. However, OpenFOAM cannot solve the RANS equations with a wave motion by *interFoam*. Therefore, a solver *waveFoam* was applied to the main simulation which is also based on the principle of *interFoam* using the VOF method. It enabled input of the wave through the boundary by setting wave parameters and damping the wave near another boundary in a relaxation zone. For this reason, *waveFoam* was regarded as a suitable solver for the main study. As a boundary condition, the wave was injected through the left side of the basin domain (inlet boundary). The wave equation was modeled by the equation:

$$\eta_m = A_m \cos(k_m x - \omega_m t + \phi_m), \tag{17}$$

where $\eta_m, k_m, \omega_m, \phi_m$ are a wave displacement, a model wavenumber, a model angular frequency, and a model phase, respectively. Each value is shown in **Table 3.11**, some of which were determined by Froude similarity in **3.3.1**. The basin's other 3 sides were set up to an advective boundary condition which is described as

$$\frac{\partial \varphi_{out}}{\partial t} + U_n \frac{\partial \varphi_{out}}{\partial n} = 0, \tag{18}$$

where φ_{out} , U_n , $\partial/\partial n$ are the volume flux going out of the domain, the velocity normal to the boundary, and partial operation normal to the boundary, respectively. This type of boundary condition is a non-reflective boundary condition that is applied to let the wave go out of the domain through the boundary without any effects of wave reflection. This boundary condition was also applied to both side walls of zone 4 in the channel domain because this zone was implemented to diminish the wave effects, too. Also, there should be no flux on the outlet boundary to implement the relaxation zone in the rightest part of the channel domain, so it was set up to U = 0. Similar to the preliminary model, a no-slip boundary condition and wall function were applied to the side walls of other zones in the channel domain and the bottom of the entire domain, and the constant pressure was applied to the atmospheric boundary.

The density and the kinematic viscosity of the fluids were set up the same as the preliminary model, and the main study employed $k - \omega$ SST model as a turbulence model. The initial k and ω were calculated as $3.674 \times 10^{-4} (m^2/s^2)$ and $3.499 \times 10^{-1} (s^{-1})$ by Eq.(8) and Eq.(9), respectively. All simulation conditions are described in **Table 3.12**, and the boundaries are illustrated in **Figure 3.10**. Except for the limiter of divergence schemes altered from the van Leer limiter to the MUSCL limiter, all numerical schemes in the main model were set up identically to the preliminary model. The total simulation time was set up to 450 seconds, but the analysis was in the last 360 seconds which corresponded to the 2 wave periods.

<i>A_m</i> [m]	<i>T_m</i> [s]	k_m $[m^{-1}]$	ω_m $[s^{-1}]$	ϕ_m
0.005	180	3.52×10^{-2}	3.49×10^{-2}	0

 Table 3.11. Wave parameters in the main numerical model

	Inlet	Basin side	Channel side	Outlet
U [m/s]	Wave inlet	Advective	No slip	0
р [Ра]	$\frac{\partial p}{\partial n} = 0$	$\frac{\partial p}{\partial n} = 0$	$\frac{\partial p}{\partial n} = 0$	$\frac{\partial p}{\partial n}=0$
k $[m^2/s^2]$	$\frac{\partial k}{\partial n} = 0$	$\frac{\partial k}{\partial n} = 0$	Wall function	$\frac{\partial k}{\partial n} = 0$
ω $[s^{-1}]$	$\frac{\partial \omega}{\partial n} = 0$	$\frac{\partial \omega}{\partial n} = 0$	Wall function	$\frac{\partial \omega}{\partial n} = 0$
$rac{ u_T}{[m^2/s]}$	Calculated	Calculated	Wall function	Calculated

 Table 3.12. Initial & boundary conditions for the main simulation



Figure 3.10. Domain and boundaries of the main numerical model

3.3.3 Mesh generation

The present study is interested in the change in the discharge affected by the structures installed in the river and needed to focus on it, not on the ocean. Therefore, the numerical mesh was generated finely in the channel domain, and those in the basin domain were comparatively larger. In the study, the maximum grid size of Δx and Δy in the basin were set up as 10 times larger than Δx and Δy in the channel, respectively. Also, the grid size in a relaxation zone was not required to be fine as other zones in the river, so Δx of the relaxation zone was set up to be 5 times bigger than that of the other zones. Δz should be enough small to treat the free surface precisely in open channel flow modeling, so it was equally set up to 0.001 *m* to both the channel and the basin domains.

When external meshes are put into the domain, if their sizes are smaller than the computational cell size, deformation may occur in the domain. To put the structures without deformation in the channel domain, Δy of the channel domain should be smaller than the transverse length of the structures, so it was set up to 0.01 m. To confirm that such Δy is appropriate, it was checked whether y^+ (a normalized distance from the wall to the center of the computational cell) was in a range of $30 < y^+ < 200$, with the following procedure:

$$1) Re_x = \frac{UL_{xx}}{v_w}$$
(14a)

2)
$$C_f = [2\log(Re_x) - 6.5]^{-2.3}$$
 (14b)

$$3) \tau_w = \frac{C_f}{2} \rho U^2 \tag{14c}$$

4)
$$u_T = \sqrt{\frac{\tau_w}{\rho}}$$
 (14d)

5)
$$y^+ = \frac{u_T y_p}{v} = \frac{u_T \Delta y}{2v}$$
, (14e)

where $Re_x, L_{xx}, C_f, \tau_w, u_T, y_p$ are the critical Reynolds number, length of the geometry, skin friction coefficient, wall shear stress, friction velocity, and distance from the wall to the center of the computational cell, respectively. Eq.(14a) is an initial guess and Eq.(14b) is an empirical formula to calculate the skin friction coefficient (Schlichting, 1979). Eq.(14c) and Eq.(14d) are the equations to obtain the wall shear stress and the friction velocity, respectively, which are required to calculate y^+ . As a result of Eq.(14e), $y^+ = 63.5075$; therefore, it was used for the grid size in the simulations.

To determine the size of Δx , a grid convergence test was conducted for different numbers of cells (*N*) in the same domain. *N* was set for 204,000, 420,750, 841,500, and 1,683,000 which were set differently by the cell size and the number of cells in *x*-direction (*N_x*) (**Table 3.13**). Each simulation was conducted for a wave period *T_m*, and the flow velocities at 0.5*T_m* and 0.75*T_m* were compared among four test cases at different points in the channel. The grid convergence was evaluated by the grid convergence index (GCI) based on Richardson extrapolation which is calculated by

$$GCI = F_s \frac{\varepsilon_{RMS}}{r^d - 1'} \tag{15}$$

where F_s , ε_{RMS} , r, d are the safety factor, root-mean-square error, grid refinement ratio, and discretization order, respectively (Roache, 1994). In the study, r = 2 and d = 2 were set up, and the comparison was for more than three grids so F_s was determined as 1.25 (Roache, 1998). The finest grid case, case 4, was regarded as the reference case, then ε_{RMSRE} was calculated by

$$\varepsilon_{RMS} = \sqrt{\frac{1}{N_p} \sum_{p=1}^{N_p} \left(\frac{U_{i,p} - U_{4,p}}{U_{4,p}} \right)},$$
(16)

where N_p and $U_{i,p}$ are the number of points used in the test and the flow velocity at *p*th point in case *i* (= 1, 2, 3, 4). The case with the *GCI* less than 0.05 was determined as the suitable numerical grid. Numerical meshes of all cases are illustrated in **Figure 3.11**.

Simulat	tion cases	Δx [m]	N _x	N
	Basin	0.1 – 1	7	
Case 1	Channel (Zone 1, 2, 3)	0.1	45	204,000
	Channel (Zone 4)	0.1 - 0.5	6	
	Basin	0.05 - 0.5	15	
Case 2	Channel (Zone 1, 2, 3)	0.05	90	420,750
	Channel (Zone 4)	0.05 - 0.25	12	
Case 3	Basin	0.025 - 0.25	30	
	Channel (Zone 1, 2, 3)	0.025	180	841,500
	Channel (Zone 4)	0.025 - 0.125	24	
	Basin	0.0125 - 0.125	60	
Case 4	Channel (Zone 1, 2, 3)	0.0125	360	1,683,000
	Channel (Zone 4)	0.0125 - 0.0625	48	

 Table 3.13. Simulation cases for the grid convergence test



Figure 3.11. Numerical mesh illustrations of entire domains for the grid convergence test; (a) case 1, (b) case 2, (c) case 3, (d) case 4

3.3.4 Experimental validation

Before the simulation, the numerical model should be validated by laboratory experiments. The experiments aimed at the flow characteristics in the channel of the numerical simulation domain, so they were conducted in a flume of 6.5 m long, 0.15 m wide, and 0.3 m in the Fluid Mechanics Laboratory, Bldg. #35, Seoul National University. The experimental domain mimicked the downstream region, structure installing region, and upstream region in the numerical channel part. The inlet boundary was set for a constant discharge generated by a pump, and the outlet boundary was controlled by the sluice gate located at the end of the flume which fixes the surface level. Both sides and the bottom of the flume were made of acrylic boards, and the bottom of the structure installing region was made of floral foam blocks. The structures were made of 2 mm-thick acrylic boards and installed above the floral foam blocks. To avoid the effects of boundaries, the observation was only on near the structure installing region.

A digital water gauge was used to measure the free surface level (**Figure 3.12**). The gauge displays a distance from the set origin to the end of the needle in mm-scale to the third decimal place. First, set down the needle of the gauge to the bottom of the flume and measure the distance. Then, set up the needle to the free surface and measure the distance again. Finally, the difference between the two distances is the free surface level. This procedure was repeated 10 times for each measuring point. An acoustic Doppler velocimeter (ADV, Vectrino Profiler – fixed stem, Nortek, Vangkroken 2, N-1351 RUD, Norway) was used to measure the velocity profile

(Figure 3.13). It can measure the instantaneous 3-dimensional velocity with its four transducers. First, the transducers transmit sound waves through the water, and they are bounced from the particles in the water. Then, the transducers receive the waves and calculate the velocity for each time step. ADV can measure the velocity of the particles in a sampling volume with the number of cells set up, but the sampling volume should be located in the middle of the water. Because it can measure the velocity of at least $4 \ cm$ from the transducers and they should be submerged in the water, measurement of the velocity near the free surface is impossible. Also, the information obtained near the bottom is not reliable due to the disturbance of the waves.



Figure 3.12. Photo of the digital water gauge



Figure 3.13. Photos of Vectrino Profiler; (a) a body and a stem fixed out of the water, (b) four transducers in the water

In the study, the free surface level was about 90 mm, so the profile range was limited from 40 mm to 60 mm (sampling volume: 20 mm). Cell size was set up to 4 mm so it had 6 cells in the sampling volume. The sampling rate, the velocity range, and the speed of sound were 60 Hz, 0.4 m/s, and $1.4796 \times 10^3 m/s$, respectively. Such setup conditions are described in Table 3.14.

Profile range [<i>mm</i>]	40 - 60	
Cell size [<i>mm</i>]	4.0	
Number of cells	6	
Sampling rate [<i>Hz</i>]	60	
Velocity range [<i>m/s</i>]	0.4	
Speed of sound [m/s]	1.4796×10^{3}	

 Table 3.14. Experiment conditions using Vectrino Profiler

The experiment cases were decided based on the unit structure design defined in the preliminary study, which is implemented by connecting several unit structures. Case **S3** consists of 3 pairs of the main structures and 2 sub-structure, and the other case **S7** consists of 7 pairs of the main structures and 6 sub-structure. Also, unlike the main model, the experiment in the Fluid Mechanics Laboratory cannot generate the wave, so a bidirectional flow cannot show up in a single case. Therefore, the experiments were conducted for both contracting cross-sections of structures (**C**) and expanding ones (**E**). Furthermore, the velocity varies in a case so we decided to conduct the experiments for two cases of inflow discharge, 36 L/m and 72 L/m (**Q36**, **Q72**), which were based on the maximum Froude number and surface level in
the main numerical model. All of the experimental conditions are described in **Table 3.15**. With the combinations of three experimental conditions, 8 experiment cases were set up (**Table 3.16**). For each case, the surface level and velocity profile were measured at 4 positions which were all located at the center of the flume's width. Numerical domain illustrations are shown in **Figure 3.14**, and **Figure 3.15** shows the photos of experiment cases in the laboratory.

The number of structures (S#)	S3, S7
The direction of the structures (C / E)	C (contracting) E (expanding)
Inflow discharge (Q#)	Q36, Q72

 Table 3.15. Experimental conditions for model validation

 Table 3.16. Experiment cases for model validation

Experiment cases				
S3_C_Q36	S7_C_Q36			
S3_C_Q72	S7_C_Q72			
S3_E_Q36	S7_E_Q36			
S3_E_Q72	S7_E_Q72			



Figure 3.14. Experimental domains and surface level & velocity profile measuring points; (a) S3_C, (b) S7_C, (c) S3_E, (d) S7_E



Figure 3.15. Photos of the experimental setup; (a) S3_E, (b) S7_C, (c) S7_E

To compare the results of the laboratory experiments to those of the numerical simulations, a new numerical model was implemented with the same cell size and structure scales as the main numerical model, and conditions were set up similarly to the laboratory experiments. The domain of the new model was designed as a straight channel like the preliminary model, so the solver and types of boundary conditions were identically applied to the preliminary model, too. Initially, the new model was set up to fit the surface level in the experiment for each case. Then, simulations were conducted for 300 seconds for Q36 cases and 240 seconds for Q72 cases, and only the data in the last 180 seconds were analyzed because the model needed enough time to be stable. Finally, the time-averaged velocity profiles of the model in 180 seconds were compared to those of the laboratory experiments and evaluated the validity of the numerical model.

3.3.5 Simulation cases setup

In the main study, the hydraulic structures were installed in zone 2 of the channel domain based on the unit structure design determined in the preliminary study. The simulation cases were devised by combining and modifying the unit design to make the best-performing design on the asymmetry in discharge. Unlike the preliminary cases, the structures in the main numerical model were installed toward the ocean for all simulation cases and only one simulation was conducted for each case. Particularly, this step focused on generating the flow structure in the channel similar to the flow in the Tesla valve. To be specific, the blocking directional flow should form a circular flow by the structures so that it could disturb the main flow. For design, several conditions were considered for CFD simulations: 1) the number of structures, 2) the location and the length of the sub-structures, 3) the shape and the length of the main structure, and 4) the offset of the structure design.

First, the preliminary study found that the spacing between the main structures and the number of structures are considerable factors in the structure design. The present study tended to know how the number of structures in a specified length of the channel affected the flow and which case could make a discharge difference in bidirectional flows the largest. Of course, the spacing between the structures also changed depending on the number of structures, so the combined effect in these simulations could be identified. Second, the effectiveness of the sub-structures was also recognized in the preliminary simulations, then it was figured out how to make the circular flow thoroughly by modifying the sub-structures in this step. The substructures were set up further away from the wall, or longer ones were installed. The main structures were also modified for the circular flow, too. It assumed that the curved structures could easily induce circular flow, so the plate-shaped structures connected with the curved structures were installed in the simulation domain. Also, to reflect the effects of the sub-structures figured out in previous simulations, longer main structures were regarded as more suitable for increasing the discharge difference with the sub-structures. Finally, the offset of the structure design was applied to the simulation to imitate the form of the Tesla valve. The longitudinal distance from the

structures on one side to those on the other side varied and the best distance for the maximum discharge difference was determined by the simulations. Detailed structure conditions are described in **Table 3.17**, and the simulation cases were set up by the combinations of such conditions (**Table 3.18**). **Figure 3.16** illustrates the domain of one of the simulation cases (S7_W0.3_C_L5/6_w0.1_y0.2) and describes it in detail.

For all simulation cases, discharge passing zone 1 in 2 wave periods was measured and flow directions were distinguished by their signs (positive or negative). Then, the total volume of each flow was respectively calculated, and the difference was obtained. This total volume difference in bidirectional flows in 2 periods was evaluated as an indicator of the performance of the hydraulic structure design.

The number of structures (S#)	S0 S6	S3 S7	S4 S8	S5 S9
A transverse length of the sub-structures (w#)		w0.1	w0.2	
The spacing between the wall and the sub- structures (y#)		y0.1	y0.2	
A transverse length of the main structures (W#)		W0.2	W0.3	3
The shape of the main structures (NC / C)	NC (Not Curved) C (Curved)			
The spacing between the structures on one side and the structures on the other side (L#)	L L3	0 L1. 6/6 L4	/6 L2 4/6 L	2/6 5/6

Table 3.17. Detailed structure conditions for the main simulation

(#: the value of the conditions)

Conditions	Simulation cases			
The number of	S0	S3	S4	S5
structures	S6	S7	S8	S9
The length &	S7_w0.1_y0.1		S7_w0.1_y0.2	
sub-structures	S7_w0.2_y0.1		S7_w0.2_y0.2	
The length & shape of the main structures	S7_W0.2_NC_w0.1_y0.1		S7_W0.3_NC_w0.1_y0.2	
	S7_W0.2_C	C_w0.1_y0.1	S7_W0.3_C_w0.1_y0.2	
The offset of the structure design	S7_W0.3_C_L0_w0.1_y0.2		S7_W0.3_C_L1/6_w0.1_y0.2	
	S7_W0.3_C_L2/6_w0.1_y0.2		S7_W0.3_C_L3/6_w0.1_y0.2	
	S7_W0.3_C_L	4/6_w0.1_y0.2	S7_W0.3_C_L	.5/6_w0.1_y0.2

 Table 3.18. Main simulation cases



Figure 3.16. Domain illustration of a simulation case: S7_W0.3_C_L5/6_w0.1_y0.2

- S7: 7 pairs of the main structures
- W0.3: a transverse length of the main structures is $0.3W_m$
- C: curved main structures
- L5/6: the spacing between the structures in a pair is $5/6W_m$
- w0.1: a transverse length of the sub-structures is $0.1W_m$
- y0.2: the spacing between the wall and the sub-structures is $0.2W_m$

CHAPTER 4. RESULTS AND DISCUSSION

4.1 Preliminary results

4.1.1 An angle between the wall and structures

First, to check whether hydraulic structures can make an asymmetry in the discharge, preliminary CFD simulations were conducted with a pair of the main structures and V_{out} (the total outflow volume in 600 seconds through the outlet boundary) was compared with the structure directions. The reference volume was set up to $V_0 = 30 \ (m^3)$, which was calculated as the total inflow volume through the inlet boundary with an inflow discharge $Q_0 = 0.05 \ (m^3/s)$ in 600 seconds of the simulation time.

Simulation results for V_{out} of each contracting and expanding case for different angles are shown in **Figure 4.1**. First of all, V_{out} is smaller than V_0 for all simulation cases; it explains that the hydraulic structures have effects on declining the outflow discharge depending on the structure direction and the angle. Comparing the results based on the structure directions, V_{out} is larger in contracting directional structures than in expanding ones for the same angle case. In other words, the structures can block the flow more with their expanding direction than their contracting direction. Comparing the results based on the angle between the wall and the structures shown in **Figure 4.2**, V_{out} gets larger as the angle gets smaller in contracting cases. On the contrary, V_{out} gets smaller as the angle also gets smaller in expanding cases. As a result, A30 shows the largest difference in V_{out} among several angle cases.

Energy dissipation is one of the considerable reasons for such structures' blocking effect. In Figure 4.3, the velocity fields show that the flow gets faster after passing between the structures. In expanding cases, however, the flow velocity gets much higher and persists for a longer distance than in contracting cases. Because the friction is proportional to the velocity magnitude, the larger friction occurs in expanding cases. Then, the flow's kinetic energy is converted to heat energy and causes a larger decrease in the discharge for expanding cases than for contracting cases. Eddy size is also related to energy dissipation. According to the streamlines in Figure 4.4, eddies are generated back to the structures, and they are larger in expanding cases than in contracting cases. The eddies keep in existence by converting the energy in the flow to their kinetic energy. Then, the larger eddies in expanding cases convert more energy than in contracting cases, and they make the discharge smaller. To compare the angle cases, A30 shows the least increase in the flow velocity and smaller eddies behind the structures in a contracting structure direction. Conversely, in an expanding structure direction, A30(E) shows a faster flow persisting for a long distance and larger eddies comparing compared to the other cases.



Figure 4.1. Comparison of the total outflow volume in 600 seconds for different angles between the wall and the structures



Figure 4.2. Comparison of the difference in the total outflow volume in 600 seconds between contracting and expanding structures for different angles between the wall and the structures



Figure 4.3. Illustrations of the velocity fields at z = 0.2 (m) in the preliminary model; (a) A30(C), (b) A60(C), (c) A60(E), (d) A30(E)



Figure 4.4. Illustrations of the streamlines in the preliminary model; (a) A30(C), (b) A60(C), (c) A60(E), (d) A30(E)

In conclusion, the simulation results verify that the hydraulic structures with an angle to the wall can lead to flow asymmetry as the present study intended. Also, the angle plays a key role in making outflow discharge difference and it becomes the largest with an angle of 30 $^{\circ}$. Based on the results, such plate-shaped structures were applied to the subsequent simulations and identically have an angle of 30 $^{\circ}$. Illustrations of the velocity field and streamlines for all simulations are attached to **Appendix A.1** and **Appendix A.2**, respectively.

4.1.2 The spacing between the main structures

In the previous simulations, it was noticed that friction and eddy size are considerable factors in the efficiency of structures to obstruct the flow, so the preliminary study focused on figuring out the unit structure design based on the sizes of the eddies generated by the structures. It assumed that the eddy size depends on the spacing between the structures, so the CFD simulations were conducted for different spacings to prove it.

According to the simulation results in **Figure 4.5**, overall V_{out} s are comparatively small to those in 4.1.1. In other words, installing 2 pairs of the main structures can reduce the outflow discharge effectively rather than 1 pair. In the contracting direction, the case with a spacing of $9W_y$ shows the largest V_{out} , and the case with a spacing of $6W_y$ shows the smallest V_{out} among the simulation cases. The results indicate that V_{out} gets smaller with the increase of the spacing between the structures until a specific spacing, and it gets bigger for a much larger spacing. This is because the effects of the two main structures interact with each other and make the structures more resistant to flow. However, if the spacing exceeds the specific distance, the main structures play their role individually without any interaction and let the flow more easily. Some results in expanding cases are different from the contracting ones. Vout for the case D3 is larger than for the other two cases which have similar values of Vout. It can be explained by the eddies generated by the structures. When the spacing is small, the eddies cannot stretch longer between the structures. According to the illustrations of the streamlines, eddies behind the first pair of structures for D3 are smaller than for D6 and D9 in both directions. Then, the area where the eddies dissipate the energy is smaller and the discharge remains higher with a smaller spacing. Comparing D6 and D9 of the cases with the larger spacings, V_{out} in the expanding direction is slightly higher for D6, but they are not much different. This is because the eddies can stretch to their maximum without any disturbance to the structures, so the two cases have similar sizes of eddies behind the structures. It leads to a similar amount of energy dissipation and a decline in the outflow discharge. Figure 4.6 shows the difference in V_{out} in both directions for different spacings and indicates that the structures with a spacing of $9W_{\nu}$ make the maximum difference in the outlet discharge.



Figure 4.5. Comparison of the total outflow volume in 600 seconds for different spacings between the main structures



Figure 4.6. Comparison of the difference in the total outflow volume in 600 seconds between contracting and expanding structures for different spacings between the main structures

In summary, with a small spacing of the structures, the eddies are restricted to the structures and cannot stretch more so energy dissipation gets less and the discharge is higher than with a larger one. As a result of the simulation, the case with the largest spacing shows the best performance in making the outflow discharge difference. However, less number of structures can be installed with a larger spacing so the study on the effects of the structures was conducted depending on the number of them.

4.1.3 The number of structures

To find out the best unit design of the structures, several combinations of the main structures and the sub-structures were set up. The structure installation was based on the spacing between the main structures which allows the eddies to stretch. Also, the sub-structures were included in the design to make the circular flow or generate more eddies so that it can impede the flow. **Figure 4.7** shows the simulation results for V_{out} for the cases S2, S2.5, and S3. In the contracting direction, V_{out} has a smaller value with a large number of structures. Comparing the velocity fields of 3 simulation cases, the area with a higher flow velocity is the largest with 3 pairs of the main structures, which leads to more friction and energy dissipation. Also, the streamlines show that there are more eddies in the domain of the S3 which implies that the flow converts its energy to the eddies' kinetic energy. In the expanding direction, the S2.5 case, which includes the sub-structures, has the smallest V_{out} . According to the velocity distribution, the higher speed of the flow persists longest in S2.5 after passing

the last pair of the main structures due to the sub-structures. Related to it, the eddies behind the last pair of the main structures are also the largest among 3 cases. Such a higher velocity flow and the large eddies cause energy dissipation and the reduction of the outflow discharge. To consider both directions of the structures and compare them, the case with the sub-structures has the maximum V_{out} difference among the cases (**Figure 4.8**). It proposes that the unit design of the structures includes the substructures.

To conclude the preliminary study, it identified that the plate-shaped hydraulic structures can play a role in controlling the discharge and it is possible to make a discharge difference in bidirectional flows by changing an angle between the wall and the structures. The CFD simulation results show that 30 $^{\circ}$ is the best angle for the performance of the structure design. Also, other simulations were conducted with varying the spacing between the mains structures and the number of the structures, and they found that sufficient spacing should be secured and the sub-structures must be contained in the structure design. Based on these results, unit structure design could be determined: 2 pairs of the main structures and 1 pair of the sub-structured with an angle of 30° (Figure 4.9). However, it was hard to certainly decide the specific spacing because the combined effects of the spacing and the number of structures should be considered. Further, the main study includes more CFD simulations with more complicated cases based on the unit design structures.



Figure 4.7. Comparison of the total outflow volume in 600 seconds for different numbers of the structures



Figure 4.8. Comparison of the difference in the total outflow volume in 600 seconds between contracting and expanding structures for different numbers of the structures



Figure 4.9. The horizontal projection of a unit design of the hydraulic structures

4.2 Model validation

4.2.1 Grid convergence test

In advance of the main simulations, suitable grid size in the *x*-direction was determined by considering both the economy and accuracy of the study. The test was conducted with the higher and lower flow velocity and each case was evaluated independently. **Figure 4.10** shows the flow velocity at various points for different numbers of cells at $t = 0.5T_m$ and $t = 0.75T_m$. According to it, case 1 is greatly inaccurate in flow velocity compared to the other cases. The velocity profiles in case 2 are slightly different from those in cases 3 and 4, and case 3 has very similar velocity profiles to the reference case. The present study was interested in the discharge in the channel, so it was checked and compared for all cases in a wave period (**Figure 4.11**). Similar to the flow velocity, all cases have similar discharge except for case 1. **Table 4.1** shows GCIs for all mesh cases. At $t = 0.5T_m$, with a high-velocity flow, GCI is less than 5 (%) in cases 2 and 3. However, GCI is less than 5 (%) only in case 3 at $t = 0.75T_m$ with a low-velocity flow. As a result, case 3 was selected as a numerical mesh for the main CFD simulations.



Figure 4.10. Flow velocity at various points in the channel domain for different numbers of cells; (a) $t = 0.5T_m$, (b) $t = 0.75T_m$



Figure 4.11. Time series of discharge in the downstream zone of the channel domain for different numbers of cells

Case	N	ε _R	\mathcal{E}_{RMS}		GCI (%)	
	IN	$t = 0.5T_m$	$t = 0.75T_m$	$t = 0.5T_m$	$t = 0.75T_m$	
1	204,000	0.3710	0.8419	15.46	35.08	
2	420,750	0.0316	0.2425	1.32	10.10	
3	841,500	0.0054	0.0762	0.22	3.17	

Table 4.1. Grid convergence index for all test cases

4.2.2 Free surface level

To validate that the numerical model describes the real flow appropriately, laboratory experiments were conducted for 8 cases and their results were compared to simulation results for the surface level and the velocity profile. **Figure 4.12** shows the free surface levels measured by both simulations and experiments. For all cases, the two results are very similar in the downstream, but the results are slightly different in the upstream. Implementing the numerical model, the downstream surface level was calibrated as the measured level in the experiment, so it should be more accurate near the outlet boundary. However, the surface levels in CFD simulations are underestimated in the upstream because the numerical simulation does not completely reflect the effects of the structures on making the surface level difference between upstream and downstream.



Figure 4.12. Comparison of the surface levels at different points in laboratory experiments to those in CFD simulations for all experiment cases; (a) S3_C_Q36, (b) S3_C_Q72, (c) S3_E_Q36, (d) S3_E_Q72, (e) S7_C_Q36, (f) S7_C_Q72, (g) S7_E_Q36, (h) S7_E_Q72

For validation, the numerical model was evaluated by the root mean square errors of the simulation results to the experiment results, which are shown in **Table 4.2**. According to the table, all cases have ε_{RMS} s less than 0.1 which is regarded as high accuracy, and they are less than 0.05 except for S7_C_Q72. In detail, the cases of lower discharges are more accurate than those of higher ones. This is because the faster flow makes the surface level difference larger and these effects are not entirely applied to the numerical model. Also, the model has the lowest error in the case S3_C_Q36, and the highest in S7_C_Q72. In conclusion, the experiment results verify that the numerical model can describe the free surface of the real flow with high accuracy.

Case	S3_C_Q36	\$3_C_Q72	S3_E_Q36	S3_E_Q72
ϵ_{RMS}	0.0328	0.0414	0.0375	0.0418
Case	S7_C_Q36	S7_C_Q72	S7_E_Q36	S7_E_Q72
Ермс	0.0353	0.0540	0.0398	0.0379

 Table 4.2. Root-mean-square errors of the simulation results to the experiment results

 for the free surface level

4.2.3 Velocity profile

Velocity profiles were also measured and compared at 4 points for each case, which are shown in **Figure 4.13**. The profiles are only from z = 0 to $z = 0.3H_0$ because ADV can measure the velocity at least 5 *cm* far from it and the free surface level is only about 9 *cm*. Also, the measurement is inaccurate near the bottom due to the disturbance of the sound wave so a point the nearest to the bottom was not counted. Likewise, the root-mean-square error was used for the evaluation of the numerical model. According to **Table 4.3**, the results are inaccurate with the expanding direction structures. Also, the cases with lower discharge have higher errors because it is more sensitive to a minute change than with high-velocity flow. In detail, the cases S3_E_Q36 and S7_E_Q36 show errors higher than 0.1 and lower than 0.2, and the others' ε_{RMS} s are lower than 0.1. The case S7_C_Q72 represents the most accurate result, and S3_E_Q36 has the worst result. Overall, the model is appropriate to simulate real-world flow velocity.

Case	S3_C_Q36	S3_C_Q72	S3_E_Q36	S3_E_Q72
ε _{RMS}	0.0814	0.0654	0.1570	0.0931
Case	87_C_Q36	S7_C_Q72	S7_E_Q36	S7_E_Q72
ε _{RMS}	0.0738	0.0531	0.1352	0.0835

Table 4.3. Root-mean-square errors of the simulation results to the experiment results

 for velocities at different points



Figure 4.13. Comparison of velocity profiles in laboratory experiments to those in CFD simulations; (a) S3 C Q36, (b) S3 C Q72, (c) S3 E Q36



Figure 4.13. (d) S3_E_Q72, (e) S7_C_Q36, (f) S7_C_Q72 (cont'd)



Figure 4.13. (g) S7_E_Q36, (h) S7_E_Q72 (cont'd)

4.3 Main results

4.3.1 Without structures

In the main study, several types of conditions were regarded as important factors in the design of the hydraulic structures, and the best conditions were figured out with the maximum discharge difference in bidirectional flows. All conditions were compared to case S0 which does not include any structure. Before the numerical simulations for various structure conditions, the analysis of discharge in the channel domain was conducted in case S0.

In the main study, the flow discharge going downstream was set as positive and the discharge going upstream as negative. In other words, the flow from the channel to the basin was set up as a positive flow, and a negative flow in opposite directional flow. According to the simulation result of S0, the discharge in zone 1 of the channel domain fluctuates with the same period of the wave and it is proportional to the discharge through the inlet boundary of the basin, which is shown in **Figure 4.14**. When the wave comes into the basin domain, most of it goes out of the domain through the open boundaries of the basin and a small amount of it propagates to the channel domain. Therefore, bidirectional flows were generated in the channel by the wave and the channel discharge strongly depends on the basin discharge. Also, it allows the study to focus only on the discharge in the channel domain. The simulation results are not stable and inaccurate at the initial time, so the analysis was only in the last two wave periods, from 90 seconds to 450 seconds of the simulation time.

The total volumes of water going downstream and upstream in 2 periods, V^{down} and V^{up} , are estimated by the sums of the water volume from positive discharge and negative discharge, respectively. The difference in the volumes is calculated by subtracting the former from the latter, which is described as $\Delta V = V^{down} - V^{up}$. In the case of S0, such different total volumes without any structures are expressed as V_0^{down}, V_0^{up} , and ΔV_0 . In the study, all the total water volumes were normalized by V_0^{down} , and ΔV_s for all cases were compared to ΔV_0 to evaluate the structure design. As a result, the indicator of the performance of the structure design was defined as a normalized ΔV relative to ΔV_0 which is described as

$$\frac{\Delta V^r}{V_0^{down}} = \frac{\Delta V - \Delta V_0}{V_0^{down}}.$$



Figure 4.14. Time series of discharges through the inlet boundary of the basin and in zone 1 of channel

4.3.2 The number of structures

The preliminary study could not decide the specific spacing and number of the structures. The effects of the spacing and the number of structures are correlated, so a reasonable number of structures should be decided for the best performance of the structure design. Considering this interacting effect, 7 simulation cases were set up to find out the best numbers of the structures in the stated length of the channel based on the unit structure design defined in **4.1**. The simulation results for V^{down} and V^{up} are shown in **Figure 4.15**. According to the simulation result of S0, V_0^{down} is

smaller than V_0^{up} , and it means that the total amount of water going upstream is more than that going downstream. Such a result is due to the geometric asymmetry of the entire domain that the size of the basin domain is considerably larger than that of the channel domain. To compare the total volume for each direction, V^{down} is similar for all cases except for S0, but V^{up} varies in the number of structures. It represents that the number of structures hardly affects the flow discharge going downstream, but it plays an important role in that going upstream. To be specific, the discharge going upstream decreases with more structures until case S7, but it increases with the increasing number of the main structures pairs exceeding 7. Following these discharge trends in the number of structures, ΔV increases with more structures with less than 7 pairs and decreases with more structures with more than 8 pairs (**Figure 4.16**). As a result, ΔV^r has the maximum value in the case of S7 (**Figure 4.17**).

To explain the simulation results, velocity fields in the channel domain were analyzed for different cases (**Figure 4.18**). According to the velocity distributions, water going downstream passes the structure installation zone through the center of the channel width with a higher velocity, and the distributions of the high velocity are similar for all illustrated cases. Then, it leads to similar amounts of energy dissipation and reduction of the flow rate for different numbers of structures. For the water going upstream, however, the velocity increases after passing the structures but rapidly decreases before passing the next structures. It shows that the structures cause higher friction with energy dissipation and also make the water hard to pass the structures. These are shown in (e), (f), and (g) of **Figure 4.18**, but not in (h). In other words, the structures well-function with installing structures, but they do not effectively block the flow with too many structures due to their small spacings.

In summation, the study identified the best number of structures to increase ΔV^r as 7 pairs of the main structures and 6 pairs of the sub-structures in zone 2 of the channel domain. This number of structures was applied to the following study on the other conditions of the structure design. Incidentally, it was found that the flow discharge going upstream strongly depends on the number of hydraulic structures, but it does not significantly affect that going downstream.



Figure 4.15. The total volumes of water going downstream and upstream for different numbers of structures



Figure 4.16. The difference between the total volumes of water going downstream and upstream for different numbers of structures



Figure 4.17. The difference between the total volumes of water going downstream and upstream for different numbers of structures relative to the difference for case S0



Figure 4.18. Illustrations of the velocity fields in the channel domain for different numbers of the structures; (a) S3, (b) S5, (c) S7, (d) S9 at $t = 0.5T_m$ (going downstream); (e) S3, (f) S5, (g) S7, (h) S9 at $t = T_m$ (going upstream)

4.3.3 Sub-structures

The main study focused on generating a circular flow like the Tesla valve by modifying the sub-structures. However, such sub-structures hardly generate circular flow because of their short distance from the walls. Therefore, to verify the effects of the sub-structures on the flow structure, two spacing between the wall and the substructures were set up. Also, it was assumed that the short length of the sub-structures does not affect the water flow near the wall. So, the simulation cases were set up with longer sub-structures to induce the flow between the wall and the sub-structures to generate circular flow. Combining these two sub-structure conditions, the simulations were conducted for 4 cases and their results were compared (Figure 4.19). It shows that V^{up} is larger in case y0.2 than in case y0.1, which means that the sub-structures near the walls block the flow going upstream more effectively than those far from the walls. To compare the cases w0.1 and w0.2, V^{up} is smaller with the longer substructures than the shorter ones, and it is also the same for V^{down} , too. In other words, the effects of the longer sub-structures on blocking the flow are larger than shorter ones both in the flows going upstream and downstream, so they do not contribute to an increase in discharge difference in bidirectional flows. As a result, the case with the shorter sub-structures near the walls, w0.1 y0.1, shows the best performance for the flow asymmetry (Figure 4.20).

The velocity distributions in **Figure 4.21** shows the simulation results. For the flow going upstream, the sub-structures far from the walls allow the water to flow between the walls and the sub-structures, but it does not form a circular flow. Rather, it joins

the main flow in the same direction and assists the discharge increase. For the longer sub-structures, they contract the cross-sectional area of the main flow so that they obstruct both bidirectional flows a lot. In addition, a reduced cross-section leads to an increase in the flow velocity which causes large friction and energy dissipation.

In conclusion, the case with sub-structures far from the walls has larger V^{up} by dispersing the flow and reducing the friction. Those with a longer length greatly block both bidirectional flows so it does not increase the discharge difference between them. As a result, ΔV^r is the maximum in case w0.1 y0.1.

However, the study found that a larger spacing between the wall and the substructures could induce the fluid to flow near the walls. Even though the near-wall flow in this step joins the main flow in the same direction and assists the main flow, this suggested that it is possible to form the circular flow by modifying other conditions of the structures.


Figure 4.19. The total volumes of water going downstream and upstream for different sub-structure conditions



Figure 4.20. The difference between the total volumes of water going downstream and upstream for different sub-structure conditions relative to the difference for case S0



Figure 4.21. Illustrations of the velocity fields in the channel domain for different sub-structure conditions; (a) w0.1_y0.1, (b) w0.1_y0.2, (c) w0.2_y0.1, (d) w0.2_y0.2 at $t = 0.5T_m$ (going downstream); (e) w0.1_y0.1, (f) w0.1_y0.2, (g) w0.2_y0.1, (h) w0.2_y0.2 at $t = T_m$ (going upstream)

4.3.4 Main structures

Based on the previous study, it is hard to form a circular flow like the Tesla valve just by controlling the spacing between the walls and the sub-structures. To achieve it, the main structures should be modified. In this step, two factors were considered to develop the simulation cases. First, the length of the main structure. It was found that the flow between the wall and the sub-structures joined the main flow going upstream in the same direction, so the sub-structures didn't work on blocking the flow going upstream. It was assumed that the circular flow can be formed with the longer main structures and it joins the main flow in the opposite direction. In this case, the spacing between the wall and the sub-structures was identically set up as $0.2W_m$. Second, the shape of the main structures. The two main plate-shaped structures are connected by a curve-shaped structure to form the circular flow more obviously. The curve-shaped structure is long toward the downstream and short toward the upstream. These two types of conditions were combined and 4 cases of the simulation were conducted, of which results are shown in Figure 4.22. To compare the cases W0.2 and W0.3, both V^{down} and V^{up} are much larger in W0.2 than in W0.3, which means that the longer main structures block both bidirectional flows much more than the shorter ones. For the shape of the main structures, V^{down} and V^{up} with curved structures are very similar to those without them except for V^{up} for W0.3 cases. Therefore, the curve-shaped structures do not significantly affect the discharge. According to Figure 4.23, W0.3 NC and W0.3 C show similar ΔV^r s which are larger than for the cases W0.2 NC and W0.3 C. It indicates that the longer main

structures reduce the flow discharge in both directions, but it is more effective for the flow going upstream so its discharge becomes smaller than the discharge going downstream.

Figure 4.24 illustrates the velocity fields for different cases in bidirectional flows. Generally, the flow velocity for cases W0.3 is much larger than for cases W0.2 for all flow directions regardless of the presence of curved structures. This is analogous to the longer sub-structures in 4.3.3; the structures reduce the cross-section of the main flow so it becomes much faster and hard to pass between the main structures. Also, the flow in W0.2s cannot form the circular flow because of the sub-structures near the walls, but it is formed in W0.3s of the flow going upstream between the walls and the sub-structures. The velocity distributions of W0.3_NC and W0.3_N are very similar regardless of the curved structures, and it leads to the similar V^{down}, V^{up} , and ΔV^r .

In summary, the longer main structures reduce both the flow discharges going downstream and upstream more than the shorter ones by their circular flow and the contracted cross-section of the main flow. However, it's more effective for the flow going upstream and it makes ΔV^r much larger. On the other side, curved structures do not play a key role in controlling the discharge. As a result, cases W0.3_NC and W0.3_C represent higher values of ΔV^r compared to W0.2_NC and W0.2_C. Considering the study to imitate the Tesla valve, W0.3_C has a similar form to the Tesla valve and looks more aesthetic than W0.3_NC, so W0.3_C was selected as the best design in this step.



Figure 4.22. The total volumes of water going downstream and upstream for different main structure conditions



Figure 4.23. The difference between the total volumes of water going downstream and upstream for different main structure conditions relative to the difference for case S0



Figure 4.24. Illustrations of the velocity fields in the channel domain for different main structure conditions; (a) W0.2_NC, (b) W0.3_NC, (c) W0.2_C, (d) W0.3_C at $t = 0.5T_m$ (going downstream); (e) W0.2_NC, (f) W0.3_NC, (g) W0.2_C, (h) W0.3_C at $t = T_m$ (going upstream)

4.3.5 Structure asymmetry

The design of the structures was improved to apply the principle of the Tesla valve and follow its geometry with specific directional curved stages. The stages are deployed asymmetrically along the flow direction, and such characteristic was also applied to the design in the last step of the study. To find the best design of the structures with the offset, simulations were set up and conducted for different spacings between the structures in a pair. The simulation results for V^{down} and V^{up} are shown in Figure 4.25. For V^{down} , it is the largest in the case L2/6 among the cases with a value of $0.8184V_0^{down}$, and the smallest in L0 with $0.8056V_0^{down}$, but the differences among the cases are quite marginal. However, the maximum V^{up} is $0.7800V_0^{down}$ in L3/6, and the minimum one is $0.7223V_0^{down}$ in L1/6, and their difference is much larger than that of V^{down} . The trends in V^{down} and V^{up} for the cases are similar, but the change is more abrupt in V^{up} . It indicates that the offset of the structure design has a significant influence on V^{up} rather than V^{down} . As a result, ΔV^r is the largest for the case L1/6 with a value of $0.2210V_0^{down}$, which shows the best performance in the present study (Figure 4.26). According to the figure, a slight offset is advantageous to increase the discharge difference, but it is unfavorable to the larger spacing between the structures in a pair.

Figure 4.27 illustrates the velocity fields of different cases in the channel domain. Comparing the flows going downstream, the velocity distributions are similar in that the water flows between the structures with high velocity, but the flow becomes slower at the entrance of the structures with offset. It leads to the reduction of friction and a slightly larger discharge. For the flows going upstream, the cases L0 and L1/6 with slight asymmetry, represent several high-velocity areas between the structures which make the friction larger. In contrast, the flows in cases L3/6 and L5/6 are unlikely to be disturbed by the structures and pass between the structures directly.

To conclude the main CFD simulations, S7 W0.3 C L1/6 w0.1 y0.2 shows the maximum discharge difference in bidirectional flows, which is illustrated in Figure **4.28**, and it is selected as the best design of hydraulic structures. The structure design includes 7 pairs of main structures and 6 pairs of sub-structures. The main structures are connected to the curved structures which are favorable to the formation of circular flows so that the main flow can be blocked by it. The sub-structures also assist this mechanism. The geometry of the design was derived from the Tesla valve so it looks similar. However, the general Tesla valve is completely asymmetric but the structure design is not. The disparity could stem from the geometry of the channel. The present study is intended for the large-scale open channel flow, and it is impossible to make the channel identical to the Tesla valve. Then, the installation of the structures was proposed as an alternative. Unlike the Tesla valve, it remains the main channel that is dominant to the flow so the effects of the structures to obstruct the flow are not as considerable as the stages in the Tesla valve. Nevertheless, the performance of the hydraulic structures designed in the study is verified by various CFD simulations, and they can be applied to the coastal area in the real world.



Figure 4.25. The total volumes of water going downstream and upstream for different offsets of the structure design



Figure 4.26. The difference between the total volumes of water going downstream and upstream for different offsets of the structure design



Figure 4.27. Illustrations of the velocity fields in the channel domain for different offsets of the structure design; (a) L0, (b) L1/6, (c) L3/6, (d) L5/6 at $t = 0.5T_m$ (going downstream); (e) L0, (f) L1/6, (g) L3/6, (h) L5/6 at $t = T_m$ (going upstream)



Figure 4.28. Domain illustration of the best design of the hydraulic structures: S7_W0.3_C_L1/6_w0.1_y0.2

CHAPTER 5. CONCLUSION

The present study aimed to find the design of hydraulic structures to maximize the asymmetry of the flow discharge in bidirectional flows. It was ascribed to the necessity of an alternative to the current hydraulic structures such as a dam or a barrage. The idea of the alternative structure was inspired by the Tesla valve which directs the flow in a specific direction. The study was conducted by numerical modeling using OpenFOAM.

The preliminary study in the model of a straight channel domain shows that the plate-shaped structures can decrease the discharge and its amount depends on the angle between the side wall of the channel and the structures. An angle of 30° represents better performance compared to that of 45°, 60°, and 90°. Case A30 shows a higher velocity flow between the structures and larger eddies back to the structures that occur the larger energy dissipation and reduction of the discharge. The structures with a large spacing let the eddies stretch longer so the energy dissipation gets larger and it allows less discharge. Installation of the sub-structures between the main structures is important to increase the discharge difference in contracting and expanding directions, and the design including two types of structures is more effective than the design with only the main structures. The preliminary study proposed the unit structure design which consists of 2 pairs of main structures and 1 pair of the sub-structures.

The main CFD model was developed as a distorted model with Froude similarity law, and it describes the stable bidirectional flows by its numerical domain of a basin and a channel and its boundaries with wave inlet and open boundary conditions. The grid convergence test and the wall function allow the reasonable grid size in the xand y directions, respectively, and that in the z-direction was set up much smaller to treat the free surface more precisely. To validate the model, laboratory experiments were conducted to measure the free surface level and velocity profile using a digital water gauge and Velocity Profiler, respectively. The CFD model was calibrated by the measured free surface level with the root-mean-square errors of 0.0328 ~ 0.0540, and the model showed reasonable velocity profiles with the RMSEs of 0.0531 ~ 0.1570.

The simulation results show that the design with 7 pairs of the main structures and 6 pairs of the sub-structures in zone 2 has the effective number of the structures for the flow asymmetry in bidirectional flows, with the value of $0.1508V_0^{down}$. A large spacing between the wall and the sub-structures allows the water to flow between them, but it does not decline the difference in discharge because it does not form circular flows like the Tesla valve. The longer sub-structures are also not effective for the asymmetry in the flow discharge. Such circular flow could be generated with the longer main structures and the curved-shaped structures, and it also increases the discharge difference by $0.2100V_0^{down}$. The design with the longer main structures without curved-shaped structures has a very similar difference to that with curved-shaped structures, but it was not selected as the design in consideration of following the shape of the Tesla valve. Finally, a slight offset of the structure design helps increase the discharge difference, but the larger one is not unfavorable to it, rather. As a result, the best design of the hydraulic structures shows a discharge difference of $0.2210V_0^{down}$, which is the maximum value among all simulation cases. In addition, this design also looks similar to the Tesla valve.

There are a tremendous number of factors to consider when optimizing the configuration of the structures such as the curvature of the curved structures and the additional structures different from the two types of structures. Also, the 3-dimensional factors could be included in the design such as the height of the structures. However, the design proposed in the present study already yields fine performance to make flow asymmetry in bidirectional flows, and it also might be the guideline for future design to improve its performance.

REFERENCES

- Corallo, M., Sheridan, J., & Thompson, M. C. (2015). Effect of aspect ratio on the near-wake flow structure of an Ahmed body. *Journal of Wind Engineering and Industrial Aerodynamics*, 147, 95-103.
- Dennai, B., Belboukhari, M., Chekifi, T., & Khelfaoui, R. (2016). Numerical investigation of flow dynamic in mini-channel: Case of a mini diode tesla. *Fluid Dynamics & Materials Processing*, 12(3), 102-110.
- de Vries, S.F., Florea, D., Homburg, F.G., & Frijns, A.H. (2017). Design and operation of a Tesla-type valve for pulsating heat pipes. *International Journal of Heat and Mass Transfer, 105,* 1-11.
- Draper, S. (2011). *Tidal Stream Energy Extraction in Coastal Basins*. [Doctor of Philosophy's thesis, the University of Oxford]. St.Catherine's College.
- Gamboa, A. R., Morris, C. J., Forster, F. K. (2005). Improvements in fixed-valve micropump performance through shape optimization of valves. *Journal of Fluids Engineering*, 127(2), 339-346.

Gibson, A. H. (1930). Hydraulics and its Applications (4th ed). Constable, limited.

Gopala, V. R., & van Wachem, B. (2008). Volume of fluid methods for immisciblefluid and free-surface flows. *Chemical Engineering Journal, 141*(1-3), 204-221.

- Greenshields, C. J. (2019). *OpenFOAM User Guide version 7*. The OpenFOAM Foundation.
- Haase, M. (2017). Numerical analysis and design of hydrostatic thrust bearing for the laboratory test rig. [Master's thesis, Gdansk University of Technology].
 ResearchGate.
- He, P., Wang, P., Liu, L., Ruan, X., Zhang, L., & Xu, Z. (2022). Numerical investigation of Tesla valves with a variable angle. *Physics of Fluids*, *34*, 033603.
- Hirt, C. W., & Nichols, B. D. (1981). Volume of fluid (VOF) method for the dynamics of free boundaries. *Journal of Computational Physics*, 39, 201-225.
- Hulsman, P., Martinez-Tossas, L. A., Hamilton, N., & Kuhn, M. (2020). Modelling and assessing the near-wake representation and turbulence behaviour of controloriented wake models. *Journal of Physics*, 1618(2), 022056.
- Hu, P., Wang, P., Liu, L., Ruan, X., Zhang, L., & Xu, Z. (2022). Numerical investigation of Tesla valves with a variable angle. *Physics of Fluids*, 34. 033603.
- Itō, H. (1960). Pressure losses in smooth pipe bends. *Journal of Fluids Engineering*, 82(1). 131-140.
- Jin, Z., Goa, Z., Chen, M., & Qian, J. (2018). Parameter study on Tesla valve with reverse flow for hydrogen decompression. *Journal of Hydrogen Energy*, 43. 8888-8896.
- Keizer, K. (2016). Determination whether a large-scale Tesla valve could be

applicable as a fish passage. [Additional thesis, Delft University of Technology].

- Lesics. (2020, September 25). *Tesla Valve* | *The complete physics* [Video]. Youtube. https://www.youtube.com/watch?v=suIAo0EYwOE
- Menter, F. R. (1993). Zonal two equation k- ω turbulence models for aerodynamic flows. In 23rd fluid dynamics conference.
- Mohammadzadeh, K., Kolahdouz, E. M., Shirani, E., & Shaffi, M. B. (2013). Numerical investigation on the effect of the size and number of stages on the Tesla microvalve efficiency. *Journal of Mechanics*, 29(3). 527-534.
- Qian, J., Wu, J., Gao, Z., Wu, A., & Jin, Z. (2019). Hydrogen decompression analysis by multi-stage Tesla valves from hydrogen fuel cell. *International Journal of Hydrogen Energy*, 44, 13666-13674.
- Roache, P. J. (1994). Perspective: A method for uniform reporting of grid refinement studies. *Journal of Fluids Engineering*, *116(3)*. 405-413
- Roache, P. J. (1998). Verification of codes and calculations. *AIAA Journal*, 36(5). 696-702
- Robert, L. S., Gary, Z. W., & John, K. V. (1996). *Elementary Fluid Mechanics* (7th ed). John Wiley & Sons, Inc.
- Roumeas, M., Gillieron, P., & Kourta, A. (2009). Analysis and control of the nearwake flow over a square-back geometry. *Computers & Fluids, 38*. 60-70.
- Ruan, X., Zhang, X., Wang, P., Wang, J., & Xu, Z. (2020). Numerical investigation 102

of the turbulent wake-boundary interaction in a translational cascade of airfoils and flat plate. *Energies, 13*, 4478.

- Schijf, J.B., & Schönfeld, J.C. (1953). Theoretical considerations on the motion of salt and fresh water. In 5th IAHR World Congress.
- Schlichting, H. (1979). Boundary Layer Theory (7th ed). McGraw-Hill, New York.

Tennekes, H., & Lumley, J. L. (1972). A First Course in Turbulence. The MIT Press.

Tesla, N. (1920). Valvular conduit (U.S, Patent No. 1,329,559A). U.S. Patent Office.

- Thompson, S. M., Paudel, B. J., Jamal, T., Walters, D. K. (2014). Numerical investigation of multistaged Tesla valves. *Journal of Fluid Engineering*, 136. 081102.
- Truong, T-Q., Nguyen, N-T. (2003). Simulation and Optimization of Tesla valves. Nanotechnology, 1. 178-181.
- 강은지.(2021년1월18일). 금강 영산강 보 처리방안 확정…해체 시기는 미정. *동아일보*.

https://www.donga.com/news/Society/article/all/20210118/104980696/1

APPENDIX

A.1 Illustration of the velocity fields in the preliminary CFD model domain



Figure A.1. (a) A90, (b) A60(C), (c) A45(C), (d) A30(C), (e) A60(E), (f) A45(E), (g) A30(C)



Figure A.1. (h) A30_D3(C), (i) A30_D6(C), (j) A30_D9(C), (k) A30_D3(E), (l) A30_D6(E), (m) A30_D9(E) (cont'd)



Figure A.1. (n) A30_D6_S2(C), (o) A30_D6_S2.5(C), (p) A30_D6_S3(C), (q) A30_D6_S2(E), (r) A30_D6_S2.5(E), (s) A30_D6_S3(E) (cont'd)

A.2 Illustration of the streamlines in the preliminary CFD model domain



Figure A.2. (a) A90, (b) A60(C), (c) A45(C), (d) A30(C), (e) A60(E), (f) A45(E), (g) A30(C)



Figure A.2. (h) A30_D3(C), (i) A30_D6(C), (j) A30_D9(C), (k) A30_D3(E), (l) A30_D6(E), (m) A30_D9(E) (cont'd)



Figure A.2. (n) A30_D6_S2(C), (o) A30_D6_S2.5(C), (p) A30_D6_S3(C), (q) A30_D6_S2(E), (r) A30_D6_S2.5(E), (s) A30_D6_S3(E) (cont'd)

국문초록

양방향 흐름 비대칭 극대화를 위한 최적의 테슬라 채널 설계

서울대학교 대학원

건설환경공학부

손 석 민

염수의 역류가 빈번히 일어나는 해안 지역에서는 이를 막기 위해 댐과 같은 수리 구조물의 설치가 필수적이다. 하지만 이러한 구조물은 상류와 하류를 완전히 분리해 생태계를 단절시키는 동시에 수질 악화 문제를 야 기한다. 따라서 이에 대한 새로운 대안이 필요한데, 이는 흐름을 기존 수 리 구조물처럼 상류와 하류의 흐름을 차단하는 것이 아닌 수평 구조를 통해 흐름을 조절해야 기존의 문제를 해결할 수 있다. 새로운 구조물은 하류로 향하는 흐름은 많이 흘러 보내는 동시에 상류로 거슬러 올라오는 흐름은 최대한으로 막아 양방향 흐름에서의 흐름 비대청을 극대화해야 한다. 이를 구현하기 위해 테슬라 밸브의 메커니즘을 활용하였는데, 이는 흐름 방향에 따라 루프 형태의 부품 내의 흐름 형태가 달라져 주 흐름을 도와주거나 방해하는 장치이다. 본 연구에서는 이러한 원리를 활용하여 간단한 형태의 구조물을 양방향 흐름이 일어나는 개수로에 적용시키고자 했고, 수치 모델링을 이용하여 이를 구현한 후 방향에 따른 흐름 총량의 차이를 비교하여 최적의 효율을 보이는 구조물 배치 설계를 결정하였다. 구조물에 따른 흐름 모사를 위해 오픈소스 전산 유체 프로그램 OpenFOAM을 사용하여 레이놀즈 평균 나비에-스톡스 방정식을 풀었고, 유체 부피 방법을 이용하여 자유 표면 흐름을 나타내고자 하였다.

직선형 수로 영역을 나타내는 수치 모델을 이용한 예비 연구는 수로와 구조물이 이루는 각도를 적절히 조절함에 따라 구조물을 이용하여 양방 향 흐름에서의 흐름 비대칭을 만들어낼 수 있다는 것을 확인하였다. 또한, 서로 다른 두 종류의 구조물을 포함한 단위 구조물 설계를 제시하여 본 연구의 효율을 증대하였다. 본 연구에서는 해안 지역 모사를 위한 파의 유입과 비반사 경계조건을 적용하여 모델을 개발하였고, 예비 연구에서 얻은 단위 구조물 설계를 바탕으로 한 다양한 구조물을 포함하여 수치 모의가 진행되었다. 격자 수렴성 평가를 통해 적절한 메시가 결정되었고, 수표면과 속도 분포 측정 실험을 통해 본 실험에 사용될 전산 유체 모델 의 유효성을 검증하였다. 수치 모의 결과, 양방향 흐름 내에서 특정 주기 동안의 흐름 총량 차이를 비교하여 가장 좋은 효과를 보이는 구조물의 개수, 길이, 배치, 모양을 결정하였다. 해당 조건들은 최적의 수리 구조물 배치 설계를 제시하였고, 추후의 구조물 설계의 발판을 마련한다.

주요어: 수리 구조물, 테슬라 밸브, 흐름 비대칭, 수치 모델링, 레이놀즈 평균 나비에-스톡스 방정식, 유체 부피 방법, OpenFOAM

학번: 2021-25858