



UNIVERSITI TEKNOLOGI MARA CAWANGAN BUKIT
BESI

MEC299

**Simulation of flow behaviour of open channel flow
through a linear straight contraction using computational
approach**

MUHAMMAD AMIR ASHRAF BIN MOHD HAZIMAN

2020846144

SUPERVISOR:

Ts. Mohamad Zamin Mohamad Jusoh

ABSTRACT

This project investigates open channel flow behaviour through the linear straight contraction. On the test portion, several angles of wall gaps are installed. This will result in a varied water flow for each wall gap. This method we can use in the weirs to slow down the flow of water so that we can minimize the velocity of movement for the fluid. A weir is a small dam that spans a body of water like a river and can control flood problems. This objective is to determine the shape and velocity of the water flow. The application used in this experiment are CAD and CFD software. The material of the test section is made from transparent acrylic cover. The expected result is that a larger wall gap will reduce water flow velocity. The angle of the wall gap will influence the water flow rate. The larger the angle used, the water flow rate will decrease. At the same time, the resulting water patterns may also be different. In conclusion, what is the relationship between the wall gap angle and the resulting water flow velocity?

TABLE OF CONTENT

ABSTRACT	4
CHAPTER 1 INTRODUCTION	6
1.1 Background study	6
1.2 Problem statement	7
1.3 Objective	7
1.4 Scope of work	7
1.5 Signafacant study	8
CHAPTER 2	9
LITERATURE REVIEW	9
2.1 Open channel	9
2.2 Flow rate	10
2.3 Perforated Weir	12
2.4 Study on the leakage flow	14
CHAPTER 3	16
METHODOLOGY	16
3.1 Methododlogy	16
3.2 Flow chart	20
3.3 Gant chart	21
CHAPTER 4	22
4.1 Expected result	22
REFERENCES	23

Chapter 1 Introduction

1.1 Background study

Akers and Bokhove's recent research on several steady states in open channel flow through a contraction inspired their work. The authors understand that when the incoming flow is supercritical, the flow pattern is two-dimensional (2D), yet they prefer to utilize the one-dimensional model (1D). (Defina & Viero, 2010) However, Open channel expansions for subcritical flow are encountered in constructing hydraulic structures such as aqueducts, siphons, and barrages. When subjected to the positive pressure gradient associated with flow, the flow tends to split in these structures and slow down, resulting in significant energy loss. If the divergence angle surpasses a certain value, the flow tends to split from its diverging sidewalls and generate chaotic eddies.

This method can lead to unfavorable flow energy losses and local and downstream sidewall erosion. In many hydraulics engineering applications, the issue of flow energy losses in channel expansions is vital. Najafi [1]. The dimensions of the flow zones grow as the flow depth decrease, and the Froude number rises to make the energy losses during the transition. In their tests, Abbott and Kline [2] observed asymmetric flow patterns on an expansion transition.

They also discovered that the flow pattern is unaffected by Reynolds numbers or turbulence intensities. Nashta and Grade [3] provide the findings of analytical and experimental investigations of subcritical flow channels with a rapid expansion. Swamee and Basak [4] present a method for designing expansions for subcritical flow that connect a rectangular channel part with a trapezoidal pipe. (Muhaisen, 2016) Other than that, With turbulent eddy motion, flow separation, and other factors, the problem of turbulent flow in an open channel expansion is extremely complex. Even under highly simplified terms, using the analytical approach to obtain solutions to problem is difficult. Although an experimental approach to solving the problem is possible, experiments are exceedingly costly to conduct. The CFD modeling approach is used in this work. Using the finite-volume approach and the RNG k-turbulence model, a numerical simulation was created. Following calibration, the influence of varying inlet discharges on the created separation zones at the transition corners and the created secondary currents were investigated. Thus, Open capillary channels are structures that contain a liquid and have one or more free surfaces (gas-liquid interfaces) where capillary forces rather than gravitational forces dominate the flow characteristics.

They're employed in various applications in space liquid management for positioning and moving liquids, including heat pipes, capillary pumped loops, and liquid handling systems for wastewaters, condensates, and propellants (Rollins et al., 1985; Rosendahl et al., 2010). Or example, in spacecraft propellant management systems, open capillary channels are employed to acquire the liquid propellant and preserve a continuous, bubble-free path between a fuel bulk and an outlet to the thruster (Bronowicki et al., 2015). Free surfaces in open capillary channels balance the pressure difference between the flow of the liquid in the channel and the ambient gas by changing their curvature by the Young-Laplace equation. For most applications, the two-phase flow generated by gas ingestion is undesirable (Rosendahl et al., 2010). It is critical to understand the effects of open channel geometry parameters and liquid properties on the flow rate to prevent gas ingestion and achieve maximum performance. (Wu et al., 2019).

1.2 Problem statement

This work has some problems. Different barriers to water flow will affect fluid movement. This problem is usually related to coastal areas because of the frequent occurrence of soil erosion due to large waves. This reduces the crowds to breathing fresh air on the beach. Therefore, we use a weir that serves as a flow controller.

However, the study of this project was carried out by placing the wall slits at different angles. The resulting flow shape after the gap in place will also be observed.

1.3 Objective

- To determine the velocity of a fluid with different angles of wall gap.
- To analyze the shape of the water flow with different wall gaps.

1.4 Scope of work

In the experiment, water will be used to find the flow of water in the gap. Water that has an average room temperature is typically around 20°C, and a density of 997 kg/m³ will be used. Other than that, in the experiment, CAD and CFD software will be used. (CAD) Computer-aided design (CAD) is a design and technical documentation technology that automates the manual drafting process (CFD). Computational Fluid Dynamics (CFD) software generates computational fluid dynamics simulations that engineers and analysts use to predict the behavior of liquids and gases. The different gaps will be used in the experiment. The gaps (30°, 60°, 90°) will be applied. The experiment requires a lot of water, so the flow rate can be clearly shown. The used of foam will be used to build the project.