UNDERGRADUATE THESIS

SCALE EFFECTS IN FREE-SURFACE FLOW MODELING WITH TURBULENT REGIME VIA 3D LARGE EDDY SIMULATION (LES) TECHNIQUE

RAVI ANTHONY TARTANDYO
NPM : 6101901037

ADVISOR: Dr.-Ing. Bobby Minola Ginting

PARAHYANGAN CATHOLIC UNIVERSITY
FACULTY OF ENGINEERING
DEPARTMENT OF CIVIL ENGINEERING
BACHELOR PROGRAM

(Accredited by SK BAN-PT Number: 11370/SK/BAN-PT/AK-ISK/S/X/2021)
BANDUNG
JANUARY 2023

SKRIPSI

EFEK SKALA PADA PERMODELAN ALIRAN PERMUKAAN BEBAS DENGAN REZIM TURBULEN MENGGUNAKAN TEKNIK LARGE EDDY SIMULATION (LES) 3D

RAVI ANTHONY TARTANDYO
NPM : 6101901037

PEMBIMBING: Dr.-Ing. Bobby Minola Ginting

UNIVERSITAS KATOLIK PARAHYANGAN
FAKULTAS TEKNIK
PROGRAM STUDI SARJANA TEKNIK SIPIL
(Terakreditasi Berdasarkan SK BAN-PT Nomor: 11370/SK/BAN-PT/AK-ISK/S/X/2021)
BANDUNG
JANUARI 2023

UNDERGRADUATE THESIS

SCALE EFFECTS IN FREE-SURFACE FLOW MODELING WITH TURBULENT REGIME VIA 3D LARGE EDDY SIMULATION (LES) TECHNIQUE

RAVI ANTHONY TARTANDYO
NPM : 6101901037

BANDUNG, 12 JANUARY 2023
ADVISOR:

Dr.-Ing. Bobby Minola Ginting

PARAHYANGAN CATHOLIC UNIVERSITY
FACULTY OF ENGINEERING
DEPARTMENT OF CIVIL ENGINEERING
BACHELOR PROGRAM
(Accredited by SK BAN-PT Number: 11370/SK/BAN-PT/AK-ISK/S/X/2021)
BANDUNG
JANUARY 2023

UNDERGRADUATE THESIS

SCALE EFFECTS IN FREE-SURFACE FLOW MODELING WITH TURBULENT REGIME VIA 3D LARGE EDDY SIMULATION (LES) TECHNIQUE

NAME: RAVI ANTHONY TARTANDYO
NPM: 6101901037

ADVISOR:	**Dr.-Ing. Bobby Minola Ginting**	....
EXAMINER 1:	**Yiniarti E. Kumala, Ir., Dipl.HE.**	....
EXAMINER 2:	**Bambang Adi Riyanto, Ir., M.Eng.**	....

PARAHYANGAN CATHOLIC UNIVERSITY
FACULTY OF ENGINEERING
DEPARTMENT OF CIVIL ENGINEERING
BACHELOR PROGRAM

(Accredited by SK BAN-PT Number: 11370/SK/BAN-PT/AK-ISK/S/X/2021)
BANDUNG
JANUARY 2023

SURAT PERNYATAAN

Yang bertandatangan di bawah ini, saya dengan data diri sebagai berikut:

Nama : Ravi Anthony Tartandyo

NPM : 6101901037

Program Studi : Teknik Sipil

Fakultas Teknik, Universitas Katolik Parahyangan

Menyatakan bahwa skripsi / ~~tesis~~ /~~-disertasi~~*) dengan judul:

Scale Effects in Free-Surface Flow Modeling with Turbulent Regime Via 3D Large Eddy Simulation (LES) Technique

adalah benar-benar karya saya sendiri di bawah bimbingan dosen pembimbing. Saya tidak melakukan penjiplakan atau pengutipan dengan cara-cara yang tidak sesuai dengan etika keilmuan yang berlaku dalam masyarakat keilmuan. Apabila di kemudian hari ditemukan adanya pelanggaran terhadap etika keilmuan dalam karya saya, atau jika ada tuntutan formal atau non formal dari pihak lain berkaitan dengan keaslian karya saya ini, saya siap menanggung segala resiko, akibat, dan/atau sanksi yang dijatuhkan kepada saya, termasuk pembatalan gelar akademik yang saya peroleh dari Universitas Katolik Parahyangan.

Bandung, 3 Januari 2023

Ravi Anthony Tartandyo

6101901037

SCALE EFFECTS IN FREE-SURFACE FLOW MODELING WITH TURBULENT REGIME VIA 3D LARGE EDDY SIMULATION (LES) TECHNIQUE

Ravi Anthony Tartandyo
NPM: 6101901037

Advisor: Dr.-Ing. Bobby Minola Ginting

PARAHYANGAN CATHOLIC UNIVERSITY
FACULTY OF ENGINEERING
DEPARTMENT OF CIVIL ENGINEERING
BACHELOR PROGRAM

(Accredited by SK BAN-PT Number: 11370/SK/BAN-PT/AK-ISK/S/X/2021)
BANDUNG
JANUARY 2023

ABSTRACT

This research was conducted using numerical simulation to investigate scale effect in hydraulic modeling, which can lead to differences between the results of the experiment and the real-world system, thus significantly affect the accuracy and reliability of physical modeling experiments. For this, OpenFOAM was utilized to investigate the scale effect, for which flow around submerged conical island as the model. Large Eddy Simulation (LES) was chosen due to its ability to solve and capture instantaneous movement in turbulent flow. The numerical simulations were conducted in two stages. The first phase was to verify whether the numerical model was in agreement with the experimental data. The second phase was to simulate upscaled model which was scaled using Froude similarity, in order to investigate scale effect, especially in the recirculating flow zone. Our results demonstrated that OpenFOAM was able to replicate the experimental data with a relatively low margin of error. Furthermore, the result showed that the scale effect appeared in the recirculating zone, in the form of either difference in velocity magnitude or difference in vortices period. However, for the nonrecirculating zone, the scale effects were not significant. This evidence proved our hypothesis that the scale effects due to the Froude similarity is quite significant when the recirculating turbulent flow occurs.

Keywords: Froude similarity, Hydraulic modeling, Large Eddy Simulation (LES), OpenFOAM, Re-circulating flow, Scale effect.

EFEK SKALA PADA PERMODELAN ALIRAN PERMUKAAN BEBAS DENGAN REZIM TURBULEN MENGGUNAKAN TEKNIK *LARGE EDDY SIMULATION* (LES) 3D

Ravi Anthony Tartandyo
NPM: 6101901037

Pembimbing: Dr.-Ing. Bobby Minola Ginting

UNIVERSITAS KATOLIK PARAHYANGAN
FAKULTAS TEKNIK PROGRAM STUDI TEKNIK SIPIL
(Terakreditasi Berdasarkan SK BAN-PT Nomor: 11370/SK/BAN-PT/AK-ISK/S/X/2021)
BANDUNG
JANUARI 2023

ABSTRAK

Riset ini dilakukan dengan menggunakan simulasi numerik untuk menyelidiki efek skala pada permodelam hidraulika. Efek skala pada permodelan fisik hidraulika dapat mengakibatkan perbedaan pada hasil eksperimen dengan sistem dunia nyata, sehingga dapat memiliki implikasi signifikan terhadap tingkat akurasi dan keandalan pada eksperimen permodelan fisik. OpenFOAM digunakan untuk menyelidiki efek skala dengan menggunakan aliran melewati pulau berbentuk kerucut yang terendam sebagai modelnya. *Large Eddy Simulation (LES)* dipilih karena kemampuannya untuk menghitung dan menangkap pergerakan seketika di aliran turbulen. Simulasi numerik dilakukan dalam dua tahap. Tahap pertama digunakan untuk memverifikasi apakah model numerik dapat mensimulasikan eksperimen. Tahap kedua digunakan untuk menyelidiki efek skala, terutama pada area resirkulasi menggunakan model yang dibesarkan menggunakan kesamaan Froude. Hasil kami menunjukan bahwa OpenFOAM dapat mendapatkan hasil yang sama dengan data eksperimen dengan margin kesalahan rendah. Selain itu, hasil menunjukan bahwa efek skala muncuk di area aliran resirkualasi dalam bentuk perbedaan kecepatan maupun perbedaan periode pusaran. Meskipun demikian, untuk area aliran non resirkulasi efek skalanya tidak signifikan. Hasil ini memberikan bukti terhadap hipotesis kami yang mana efek skala karena kesamaan Froude cukup signifikan ketika aliran turbulen resirkulasi terjadi.

Kata Kunci: Aliran resirkulasi, Efek skala, Kesamaan Froude, *Large Eddy Simulation (LES)*, OpenFOAM, Permodelan hidraulik.

PREFACE

I would like to start off by expressing my gratitude to Parahyangan Catholic University for all the resources I had access to. Also, I would like to thank all the members involved in the evaluation of my report, such as my advisor, reviewers and examiners. I highly appreciate the time investment they put forth into helping me improve my thesis.

More particularly, I would like to thank my advisor: Dr.-Ing. Bobby Minola Ginting. He helped introduce me to turbulence modeling as well as bolstered my motivation for advanced CFD studies. Moreover, the exciting scientific discussions pertaining to CFD were fun and essential to this work's fruition and my career development. Thank you for offering me this amazing opportunity to work on something I really enjoy. Also, I cannot but mention my special thanks to James Zulfan S.T., M.Sc. for his helps and inputs in my study.

Additionally, I would like to thank Prof. Robertus Wahyudi Triweko, Ph.D, Doddi Yudianto, Ph.D., Bambang Adi Riyanto, Ir., M.Eng., Salahudin Gozali, Ph.D., Yiniarti E. Kumala, Ir., Dipl.HE., Albert Wicaksono, Ph.D., Stephen Sanjaya, S.T., M.Sc., and Finna Fitriana, S.T., M.S as the examiners, and for all their support throughout the recent years of my academic life.

I would like also to thank my colleagues from Water Resource Engineering, who have helped me greatly with their inputs, particularly Andrea Polisar, Janssen Alandrio Hendry, Nicholas Audwin Agathon, and Jose Cristobal. On a personal level, I cannot but mention my friends (particularly Danas Atmaja, Juan Nugraha, Raymond Deprista, Thalia S Azzahra, Divanza Audi D, Atharayhan A, Henry Baswara, Renaldi C Gomel, Matthew Hernando, Yohanes Yovin) both near and far, that helped me along my academic journey and made it a little more fun and much more bearable.

Last, but definitely not least, I would like to express my eternal gratitude to my wonderful parents and my family. They were/are always there for me, despite the distance. You perpetually advocate me to become a better person and consistently place very high emphasis on my education, while making sure I have everything I need to achieve it; you believed in my success, even when I, myself, did not. I cannot find words to express how indebted I am to your unceasing support. In my humble opinion, while I did my part and worked diligently, yet none of the outstanding opportunities could have ever been possible for me if it wasn't – and still isn't – for them. Also, I thank God for all the delightful people I have met, and still meet, along my journey.

The computations were performed on resources provided by the Water Resource Engineering Laboratory at Parahyangan Catholic University.

Bandung, 3 January 2023

Ravi Anthony Tartandyo
6101901037

TABLE OF CONTENT

LIST OF NOTATIONS

c	:	Courant Number or CFL Number
Ca	:	Cauchy Number
E	:	Bulk Modulus of Elasticity (Pa)
Eu	:	Euler Number
Fr	:	Froude Number
g	:	Gravitational Acceleration (m/s^2)
L	:	Hydraulic Length (m)
L_m	:	Model Hydraulic Length (m)
L_p	:	Prototype Hydraulic Length (m)
p	:	Pressure (Pa)
Re	:	Reynolds Number
t	:	Time (s)
ρ	:	Density (kg/m^3)
U	:	Velocity (m/s)
V	:	Velocity (m/s)
V_m	:	Model Velocity (m/s)
V_p	:	Prototype Velocity (m/s)
u_η	:	Kolmogorov Velocity Scale
τ^R	:	Reynolds Stress Tensor (m^2 . s^2)
τ^{SGS}	:	Sub-grid Scale Stress Tensor (m^2 . s^2)
τ_η	:	Kolmogorov Time Scale
η	:	Kolmogorov Length Scale
λ	:	Scale
Δt	:	Timestep (s)
Δx	:	Maximum Element Size (m)
ε	:	Average Rate of Dissipation (m^2/s^2)
μ	:	Dynamic Viscosity (Pa . s)
ν	:	Kinematic Viscosity (m^2/s)

LIST OF FIGURES

LIST OF TABLES

LIST OF APPENDICES

CHAPTER 1
INTRODUCTION

1.1 Background

Free-surface flow or open-channel flow can be described as a gravity-driven fluid flow, of which the free-surface is adjacent to the air in the atmosphere (river, drainage system, etc.). According to the ratio of inertial forces to viscous ones within a fluid, known as Reynolds number, there are three types of flow, namely laminar, transitional, and turbulent flows. Using the Reynolds number, instability usually starts to occur when $Re = 10^5$, and fully developed turbulent flow will start to occur when $Re = 5 \times 10^6$ (Incropera & DeWitt, 1981).

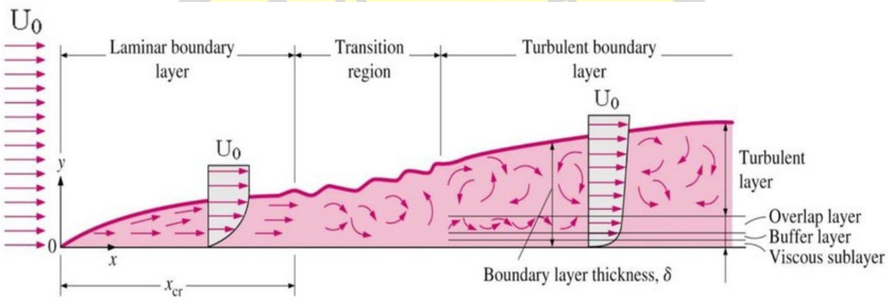

Figure 1.1 Critical Reynolds Number (Frank M White, 2010)

Each type of free-surface flows has a different behavior, which can be inspected based on the movement of flow particle: laminar flow has a relatively straight particle movement, whereas turbulent flow has a random, rotating one. Generally speaking, exogeneous perturbations that enter the boundary layer and are filtered, eventually turn into unstable waves. Variables which determine the path to turbulence are the coherent flow structures arising, the 'critical' or 'transitional' Reynolds number, the skin friction and the heat transfer to/from the wall.

In hydraulics, the characteristics of free-surface flow can be observed by two approaches: physical and numerical modeling. Physical modeling relates to a laboratory model that replicates a real-life prototype and is used as an instrument to find an optimal solution in engineering and economic terms regarding the problems of hydraulic engineering (Novak, 1984). The result differences between a laboratory model and its prototype may occur due to model, scale, and/or measurement effect (Heller, 2011). Scale effect will not occur if a physical model is entirely identical to its real-life prototype in terms of three similarity criteria, namely geometric, kinematic, and dynamic (Yalin, 1971; Kobus, 1980; Novak, 1984; Hughes, 1993; Martin and Pohl, 2000; Heller, 2007; Heller, 2011). The geometric similarity involves a similarity in shape indicating that the length of the prototype must be scaled with a certain factor (λ); hence, the model will have a length that λ times shorter. The kinematic similarity deals with the ratio between a model and its prototype in terms of the properties of motion, such as velocity, time, acceleration, and discharge, in addition to the geometric similarity. The dynamic similarity states that the force ratio in the system of both model and prototype must remain the same, in addition to the kinematic and geometric similarities.

Numerical model is a representation of a physical system through a combination of mathematical equations that rely on computers to find approximate solutions to the underlying physical problem. In many cases, numerical modeling is very useful to deepen the understanding of this physical problem, especially in hydraulic engineering, with its result visualization as several properties of fluid which may be difficult to be obtained in physical modeling, such as turbulent kinetic energy, vorticity and particle movement. Numerical models can approximate the solutions to many physical problems defined by several mathematical equations such as the Navier-Stokes equation in spatial (1-D, 2-D, and 3-D) and temporal dimensions. Several numerical models are commonly used for free-surface problems such as HECRAS (1-D), SRH-2D and NUFSAW2D (2-D), as well as Ansys Fluent and OpenFOAM (3-D). It appears that any fluid flow is in reality of 3D problems; however, these problems are often too difficult to calculate, and thus simplifications are needed. This can be achieved by neglecting the changes of flow in one or two directions (e.g., 1-D or 2-D model), thereby reducing the complexity as well and computational time.

As previously mentioned, the dynamic similarity ensures the sameness of the force ratio of both model and its prototype. Four scales commonly used to indicate a dynamic similarity are Froude, Reynolds, Weber, and Cauchy numbers. Note that for free-surface flows, the Froude similarity is often used because the inertia and gravity forces are dominant, especially if friction effects are negligible or for highly turbulent phenomena. In reality, only one force ratio can be equal for both model and prototype systems. Hence, the dynamic similarity is impossible to achieve (Kobus, 1980). To deal with this issue, only the most prominent force ratio is thus selected. Consequently, the scale effects arise because the other force ratios are neglected. The implications of scale effect in hydraulic physical modeling can be significant. Scale effect refers to the changes in the behavior of a fluid flow that can occur when the scale of the model is different from the scale of the real-world system being studied. This can be a significant challenge when conducting physical modeling experiments, as it can be difficult to predict how the behavior of the flow will change when the model scale is changed.

One of the main implications of scale effect is that it can limit the accuracy of physical modeling experiments. When the scale of the model is changed, the behavior of the flow can change in unpredictable ways. This can lead to differences between the results of the experiment and the real-world system, and it can make it difficult to accurately predict the behavior of the flow. Another implication of scale effect is that it can make it difficult to compare the results of different physical modeling experiments. If the experiments are conducted at different scales, the results may not be directly comparable, as the behavior of the flow can be influenced by the scale of the model. This can make it challenging to draw conclusions from the experiments, and it can limit the ability to generalize the results to other systems. Overall, scale effect in hydraulic physical modeling can have significant implications for the accuracy and reliability of physical modeling experiments. It is an important factor to consider when conducting such experiments, and it is an area of active research.

Although using the Froude similarity is common for free-surface flows (in turbulent regime), further study to investigate the scale effects is still needed, especially when vortices or recirculating flow occurs. Hypothetically, the scale effects would be very dominant when vortices or recirculating flow appear in physical modeling. In order to prove this hypothesis, we numerically simulated the laboratory experiment done by Lloyd & Stansby (1997) dealing with turbulent recirculating flows. To this regard, the freeware OpenFOAM v2012 will be used.

OpenFOAM is a free, open source CFD (Computational Fluid Dynamics) software developed primarily by OpenCFD Ltd since 2004. It has a large user base across most areas of engineering and science, from both commercial and academic organizations. OpenFOAM has an extensive range of features to solve complex problems from fluid flows involving chemical reactions, turbulence and heat transfer, to acoustics, solid mechanics and electromagnetics. This research will include the investigation of scale effects using Large Eddy Simulation (LES) technique. LES technique is the combination of Direct Numerical Simulation (DNS) and Reynolds-Averaged Navier Stokes (RANS) models. Hence, it will give the best of both methods, the accuracy and the computing time advantage.

1.2 Objective

This thesis aims to study the scale effect in free-surface recirculating flow with turbulent regime by means of numerical modeling (LES model). The objectives of this thesis are:

1. To validate the results of OpenFOAM v2012 with the observed data from Lloyd & Stansby (1997).
2. To investigate the scale effects in free-surface recirculating flow modeling with turbulent regime.

1.3 Scope of Study

This thesis focuses on the turbulent simulation performed using OpenFOAM v2012. The benchmark data used for the research is from Lloyd & Stansby (1997) and it will be used for validating the numerical model. For the spatial discretization, a mesh size of 0.0152 m will be used. For investigating the scale effect, this model will be upscaled by a factor of 3 and 10 (mesh size will also be scaled proportionally), and the Froude similarity will be used. The result of the simulation is the velocity magnitude, which will be compared with the benchmark data.

1.4 Research Methodology

1. Literature Review

   This step is implemented to understand the concepts about the study according to some previous studies.

2. Data Collection

   This step is implemented to collect benchmark velocity data from Llyod & Stansby (1997)

3. Mesh Generation

   This step is carried out using Ansys DesignModeller (student version) to create the geometry and to create the mesh.

4. OpenFOAM Simulation

   This step is implemented by setting up case in OpenFOAM v2012 in Ubuntu Platform, then the simulation is executed in parallel processing environment.

5. Data Visualization

   This step is implemented by using ParaView and Microsoft Excel to visualize data output from OpenFOAM simulation.

6. Result analysis

   This step is carried out to analyse, compare, and finally conclude the result of the simulation.

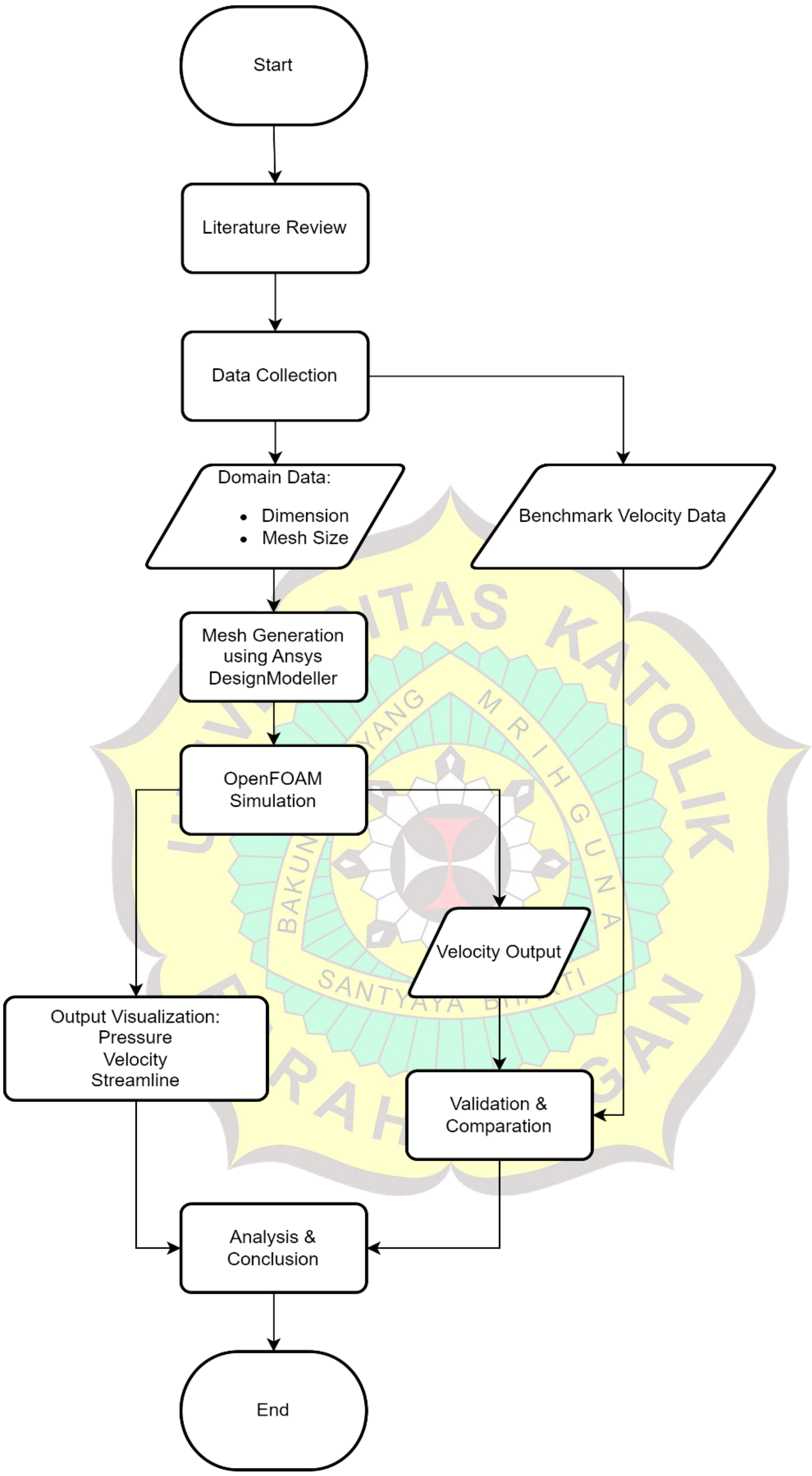

Figure 1.2 Flowchart