

**UNIVERSITI TEKNOLOGI MARA**

**ANALYSIS OF WAVE GENERATED  
AT THE BOW OF CATAMARAN-  
HULLED BOAT FOR JET SKI  
ATTACHMENT USING ANSYS  
FLUENT**

**MUHAMMAD HARITH IKHWAN BIN SOUFEI**

Dissertation submitted in partial fulfillment  
of the requirements for the degree of  
**Diploma in Mechanical Engineering**

**College of Engineering**

**December 2023**

## ABSTRACT

This project will develop the combination of the jet ski at the front of the boat. The problem statement for this project is to stimulate and analyse the fluid flow created by the combination jet ski at the front which is catamaran hull. However, this design is improved from the current boat which is wave boat that the attachment of jet ski at the back of the boat, but for this project the attachment of the jet ski will be at the front of the boat because it will be easy for the driver to manoeuvring the jet ski and will unblock the view of the driver. This design also will improve from the monohull design into the catamaran hull design. This project will study the interaction of the fluid with the attachment of the jet ski of the model. The problem for this project to find the interaction of the fluid with the combination of the boat in term of resistance, contour velocity by using an Ansys Fluent Software. There are some objectives for doing this project which are to design the combination jet ski at the front of boat using the computer aided Design (CAD). Other than that, to simulate and analyse the fluid flow created by the combination jet ski at the front, which is catamaran hull using Fluid Simulation Software, Ansys Fluent. The result for this project that gets from the Computational Fluid Dynamic, Ansys Fluent which are achieve for the calculation of resistance of the model, the velocity contour of the model and the vector distribution of the model. From this project, can conclude that the higher the speed, the higher the resistance of the interaction between model and fluid. For the contour velocity that get from the result are when speed is higher the contour will provide red colour of contour while when the speed in intermediate the colour of contour is green and yellow colour. From this project also show the movement of the fluid from the inlet through the outlet. In conclusion, Computational Fluid Dynamic (CFD) can help to analyse and stimulate the resistance, velocity contour distribution and vector distribution between the fluid and the hull. Computational Fluid Dynamic (CFD) has become a valuable tool for engineering in the design and analysis of marine system.

## **ACKNOWLEDGEMENT**

Firstly, and foremost, I would like to express wish to thank to Almighty, Allah SWT for giving me the opportunity to embark on my diploma and allowing me to complete this long and challenging journey successfully. Next, I would like to express my appreciation and thanks go to my supervisor, Ts. Mohamad Zamin Mohamad Jusoh for guiding me to complete this Final Year Project Report and help me to analyse using ANSYS Fluent, Work Bench.

Finally, this dissertation is dedicated to my parent for the vision and determination to educate me. This piece of achievement is dedicated to both of you. Alhamdulillah's.

# TABLE OF CONTENTS

	<b>Page</b>
<b>CONFIRMATION BY SUPERVISOR</b>	<b>ii</b>
<b>AUTHOR'S DECLARATION</b>	<b>iii</b>
<b>ABSTRACT</b>	<b>iv</b>
<b>ACKNOWLEDGEMENT</b>	<b>v</b>
<b>TABLE OF CONTENTS</b>	<b>vi</b>
<b>LIST OF TABLES</b>	<b>viii</b>
<b>LIST OF FIGURES</b>	<b>ix</b>
<b>LIST OF ABBREVIATIONS</b>	<b>xi</b>
<b>CHAPTER ONE : INTRODUCTION</b>	<b>1</b>
1.1 Introduction	1
1.2 Background of Study	1
1.3 Problem Statement	3
1.4 Objectives	3
1.5 Scope of Work	4
1.6 Significance of Study	5
<b>CHAPTER TWO : LITERATURE REVIEW</b>	<b>6</b>
2.1 Jet Ski Attachment.	6
2.2 Computational Fluid Dynamics (CFD).	8
2.2.1 Resistance.	8
2.2.2 Wave pattern.	13
2.3 Modify monohull into catamaran hull.	13
2.3.1 Ship analysis.	14
2.3.2 Resistance Component Analysis.	16
2.4 Ship resistance using Computational Fluid Dynamic (CFD)	17
<b>CHAPTER THREE : METHODOLOGY</b>	<b>18</b>
3.1 Introduction	18

3.2	Information For Method Selection	19
3.3	Modelling	20
3.4	Boundary Creation	22
3.5	Meshing	24
3.6	Fluent	26
	3.6.1 Setup For Calculation	26
	3.6.2 Contours	28
3.7	Scaled Residual Graph	28
3.8	Result Obtain	29
3.9	Flow Chart.	31
3.10	Project Planning (Gantt Chart).	32
 <b>CHAPTER FOUR : RESULTS AND DISCUSSION</b>		<b>33</b>
4.1	Introduction	33
4.2	Computational Fluid Dynamic (CFD) Resistant Result	33
4.3	Velocity Distribution on the hull	36
4.4	Vector Distribution on the hull	38
 <b>CHAPTER FIVE : CONCLUSION AND RECOMMENDATIONS</b>		<b>41</b>
5.1	Conclusions	41
5.2	Recommendations	42
 <b>APPENDICES</b>		<b>46</b>