FACULTY OF MECHANICAL ENGINEERING

UiTM Terengganu, Campus Bukit Besi

Program : Diploma in Mechanical Engineering

Course : MECHANICAL ENGINEERING DESIGN

Course Code : MEC 332

Group : EM110 5E

Lecturer : MD ZAMIN BIN MD JUSOH

FINAL YEAR PROJECT

CFD ANALYSIS

No.	Name	Student ID	Signature
1.	MUHAMMAD SYAKIR BIN MOHAMAD SAIDI	2018640158	
2.	MUHAMMAD HISYAMUDDIN AQIL BIN HASMADI	2018269384	
3.	AMER DANIEL BIN ADAM ARIF	2018637368	
4	MUHAMMAD AKMAL BIN YUSOFF	2018284528	
5	CHE WAN MUHAMMAD AIZAT BIN CHE WAN ROZUWAN	2018444878	

Received by:

Date of Submission: 28 January 2021

_____

Lecturer / Instructor

Table of Content

1.0 Introduction

1.1 Overview of the Project

Our project is about doing the analysis of the Computational Fluid Dynamic on Payang Fiber Boat hull. Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and data structures to solve and analyze problems that involve fluid flows are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. Computational Fluid Dynamics or CFD as it is popularly known, is used to generate flow simulations with the help of computers. CFD involves the solution of the governing laws of fluid dynamics numerically. The complex set of partial differential equations are solved on in geometrical domain divided into small volumes, commonly known as a mesh (or grid).

Today, several of the CFD tools play an important role in the ship hull form design. In this project CFD has been used to determine the pattern of the flow and drag force when the coordinates of the boat were changed which represents a very important task in the principal and final design stages. The design and the position of the hull has and important rule to see the pattern of fluid flow and its drag force. ANSY CFX software were used in this project. This software is capable to estimate and predict some characteristics of the flow pattern around the ship hull form which are not possible to be obtained by model tests. However, in order to use the computational method, CFD, as a useful tool for hull design, the CFD results must be credible and accurate enough, and can forecast the changes in the flow pattern characteristics accompanied by the hull's coordinate variations.

Lastly, there are three stage to make the simulation in ANSY.it started with the geometry, meshing and solver.

Payang Fibre Boat

Perahu payang' or simply payang is a traditional Malay open fishing boat that famous in East Cost Malaysia. They usually found in Terengganu, Kelantan, Pahang and even Johor coast. Perahu payang is the biggest and popular boat that has been used by the fisherman in East Cost Malaysia. Perahu Payang has a crew of 15 to 20 men when fishing. They will caries a net called 'Pukat Payang'. The bow and the stern are built up fantastically giving a most striking appearance. The ends, the keel and the bottom planks are built of Chengai wood, the top plank usually Bening Serayah wood. The upper strakes of Malay boats are usually made of light woods as Serayah or Medang

Modelling

As all around the world need to deal with this pandemic, the usually schedule to make the measurement of the Payang boat with guidance by our lecturer and supervisor our lecturer and supervisor at the museum was cancelled. But we still got the model and design of the Payang boat with helped of Sir Azhari which is our lecturer in Mechanical Engineering Design (MEC332). Some of adjustment need to be done on the boat because we only need the solid hull for this CFD project. So the Rhinoceros and SolidWork were used to make this adjustment.

Simulation

The ANSY CFX were used to make the analysis and generate the simulation of Payang Fibre Boat. Ansys CFX software is a high-performance, general-purpose fluid dynamics program that engineers have applied to solve wide-ranging fluid flow problems. CFX is an advanced solver technology, it was the key to achieving reliable and accurate solutions quickly and robustly.