Variable Pitch Propeller Design Tool Development

Christopher Nunes, Michael Neal, Raul Vidal and Dr. Edgar An
College of Engineering and Computer Science
Florida Atlantic University

Introduction

Propellers are used to create thrust to propel a vehicle through a fluid. A propeller rotates by torque being transferred from the drive mechanism to the shaft; it pushes the fluid (in our case, water) backwards which causes the vessel to move forward. As the velocity of the vessel increases the water flowing into the propeller also increases. This inflow of water creates a desired lift effect on the surface of the prop but is also accompanied by a drag force. The net thrust produced (lift force minus drag force) depends on several factors, a few being the geometry of the propeller, total torque transferred to the prop, vehicle design, maximum speed and fluid density (Carlton 2007).

As the prop is rotating, the surface of the blade encounters the fluid at an angle of attack which is responsible for generating lift. The flow angle is determined by the incoming fluid velocity and propeller rpm.

This angle varies with vehicle speed and prop rotation. The pitch is the sum of these two angles, and the blade is designed to maintain a constant thrust along its radius while accounting for these dynamic quantities. This is what gives propellers their characteristic twisted shape. Figure 1.1 shows these angles according to one radial section of the blade. These features are considered to produce a maximum thrust at a designed vehicle and prop speed.

The newest submarine hull has been designed to break the world record for fastest human powered submarine which currently rests at 8.035 knots (14.9km/hr. 9.2mph).

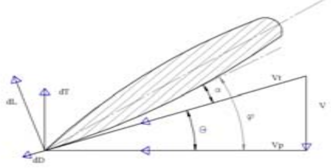

Figure 1.1: Blade Element Angle and Vector Layout (see Table 1.1 for nomenclature)

One of our main propeller design constraints is that the pilots can only maintain maximum speeds for a short period of time. These maximum speed intervals are directly dependent on the driver(s) capacity to pedal.

What this means is that our propeller will not experience a steady torque and as a result, neither the incoming fluid flow nor the submarine speed will be steady. This affects the angle of attack and total thrust that will be produced.

So far, we have described some of the main variables and parameters that needed to be considered while laying down the ground work for designing our prop design tool.

Even though there are many more variables to be considered, we effectively narrowed our design focus to consider the variables that allow us to approximate maximum thrust based off empirical data and theoretical formulation put together by the National Advisory Committee for Aeronautics (NACA). Using this approach we have established a sound method for verifying our final propeller design tool by comparing theoretical results with empirical data.

Design Tool Development

Based off NACA foil wing section theory the thrust and torque of the propeller should be able to be approximated numerically if the NACA foil shape is known. In the effort to develop this tool accurately, we began from a known reference, the one man submarine in the HPS fleet (Talon 1). The propeller's chord lengths and pitch angles were measured along sections of the radius while the foil shape was estimated. We numerically recreated the existing propeller for to verify our design tool. By knowing the chord lengths and pitch angles along radial sections of the propeller, the calculations for thrust and torque are relatively straightforward when the vehicle target speed and prop rpm are identified. In an effort to utilize MATLAB® to gain a fuller understanding of the propeller's characteristics, the formulas were applied to each section at incrementing speeds until a goal velocity was reached. This form of analysis creates outputs such as Figure 2.1. This mesh plot shows how each section of the prop reacts in open water at speeds from 0 to 9 knots

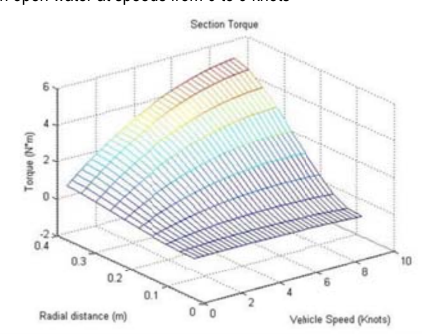

Figure 2.1: Torque calculations done at each radial section over a range of speeds.

This type of plot can show what areas of the prop are responsible for certain effects or behaviors at different speeds. To illustrate this behavior, it can be seen that the torque spikes at the tip of the prop as the velocity increases compared to the rest of the blade. This information can greatly influence design changes to make an efficient prop. This graphical representation will be a helpful learning tool when combined with the MATLAB® GUI described in the next section.

In an effort to accurately simulate the operating conditions of the prop, we considered the nominal wake of the vehicle according to CFD models that ran on SolidWorks® Flow Simulation.

Results analyzed with a cut plot are shown in Figure 2.2. This is an image of the varying velocities in the area that the propeller operates. The probe points show the numerical value at the radial distances tested in the MATLAB® program.

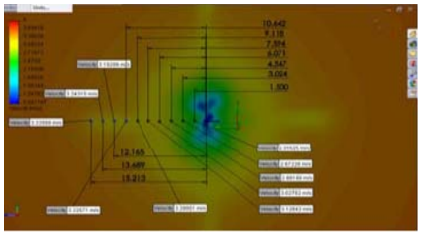

Figure 2.2: Velocity plot of the nominal wake of Talon 1

Instead of using one constant for all radial sections, there are different constants for each radial section which can be plugged in or created with a derived formula. This way the numbers will more accurately represent the conditions experienced by the prop for a more accurate result. Currently, multiple CFD tests are being run on a model of Talon 1 to get a better grasp on how these constant distributions vary with different vehicle speeds. The differences in the results due to this consideration are subtle when looked at graphically, but the change is easier to compare when observing the theoretical vehicle speed where the torque limit is reached, which is represented in Table 2.1.

When considering the velocity gradient caused by the hull, there is an 11% difference in projected speed from open water testing and a 7% difference from a using single constant. This positive and negative change in projected speed shows how dependent the prop is on the inflow velocities. These considerations are not large, but they will be continuously modified with feedback from more tests to fine tune our tool.

Graphical User Interface (GUI)

There were two intentions when the development of the graphical interface was initiated. The main focus of the interface is to streamline the use of MATLAB® script that was developed for the propeller design. When the script was first being developed, it became problematic when the dimensions of the propeller needed to be changed because the users would have to edit the code each time this needed to be done. This caused the goal of applying this script to other propellers to become unrealistic if this approach was not altered. By allowing the user to change the values without editing the code of the design tool this issue was bypassed. The secondary use of the design tool is to allow students who do not have experience in propeller design to have an interactive way to learn the material. This is useful because students may be discouraged when dealing with the large amount of information that is associated with propeller design. By providing visual representations, students will have an experience similar to that of a videogame by seeing immediate changes in figures and outputs due to their efforts.

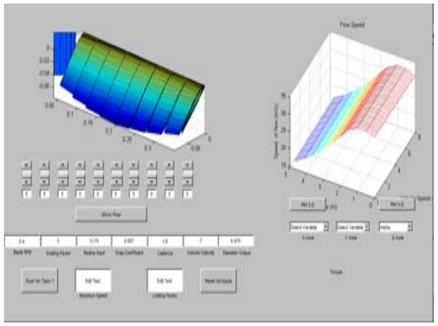

Figure 3.1: Graphical Interface for Design Tool

References

1. Abbot, I.H. and von Doenhoff, A.E. (1959). "Theory of Wing Sections", Dover Publications, New York, NY, USA
2. Carlton, J. (2007). "Marine Propellers and Propulsion", 2nd edition, Butterworth-Heinemann, Burlington, MA, USA