FLOW ANALYSIS ON A DUCTED PROPELLER BY USING CFD (COMPUTATIONAL FLUID DYNAMICS) APPROACH

Name : Widayatno
NRP : 4107100027
Institution : Department of Naval Architecture and Shipbuilding Engineering
Faculty of Marine Technology
Sepuluh Nopember Institute of Technology
Supervisor : Prof. Ir. I Ketut Aria Pria Utama, M.Sc., Ph.D.

ABSTRACT

The usage of ducted propeller can influence the thrust generated on a ship, where the presence of a duct with a cross-shaped foil surrounding the propeller can improve propeller efficiency and generate greater thrust than propeller without duct. The aims of this study are to determine the flow characteristics of the ducted propeller and generated the thrust. This research uses Ansys-CFX software with CFD method (Computational Fluid Dynamics) to analyze the value of the thrust on ducted propeller. From the simulation results of both models, propeller thrust can be increased by about 17% in the ducted propeller when compared to the propeller without ducted, and this is in good agreement with published data which showed an increase of about 15%.

Keywords: Ducted Propeller, Thrust and CFD