CFD Analysis of Flow Past a Submerged Vane

Parth Chennur
Department of Mechanical Engineering, Sinhgad College of Engineering, Pune, India
Saloni Suryawanshi
Department of Mechanical Engineering, Sinhgad College of Engineering, Pune, India
Siddhi Suryawanshi
Department of Mechanical Engineering, Sinhgad College of Engineering, Pune, India
Durvesh Vikhankar
Department of Mechanical Engineering, Sinhgad College of Engineering, Pune, India
T. L. Gilbile
Department of Mechanical Engineering, Sinhgad College of Engineering, Pune, India
Sunil Chandel
Department of Mechanical Engineering, Defence Institute of Advanced Technology, Pune, India

Abstract

A submerged vane functions as a hydraulic structure that generates vortices, inducing secondary circulation in the fluid by creating a pressure differential across its surfaces. The performance of the vane depends on its dimensions and configuration. The analysis was conducted in a straight open channel with an approximate discharge of 0.22 m^3/s using computational fluid dynamics (CFD). The finite volume method (FVM) and the numerical tool ANSYS FLUENT were employed to solve the three-dimensional Reynolds-averaged Navier-Stokes equations with the Reynolds stress model. As the flow progresses downstream from the vane, two minor counter-rotating vortices develop near the channel sides, while a single major vortex forms near the vane's centerline. Further downstream, the minor vortices migrate toward the suction side, and the major vortex shifts toward the pressure side. Circulation strength is generally negative upstream and positive downstream, reaching its maximum near the vane. The transverse bed shear stress also varies, exhibiting negative values on the suction side and positive values on the pressure side at the vane's location, before transitioning to a bell-shaped curve with positive values downstream.

Index Terms

CFD, FLUENT, FVM, Submerged Vane