

ABSTRACT

This report aims to describe the flow behavior and calculation of flow parameters in an Elbow draft tube at elevated head of Francis turbine using the software ANSYS CFX Mesh and OpenFOAM. It aims to find & study the Simulated results of Draft tube of a Francis Turbine Design for a specified parameter. The Draft tube is a Elbow with varying cross section, It consists of an extended elbow type tube. Generally, used when turbine has to be placed close to the tail-race. The conventional method to predict the performance is physical testing of turbine model which is time consuming and costly. It helps to cut down the cost of excavation and the exit diameter should be as large as possible to recover kinetic energy at the outlet of runner.

Keywords: draft tube of Francis Turbine, CFD, OpenFOAM.

PROBLEM STATEMENT

For a Incompressible laminar flow for Newtonian fluids, Analyze the Water Flow behavior and calculate of flow parameters in the Draft tube (Figure 1) of a Francis turbine. Use a suitable turbulence model.

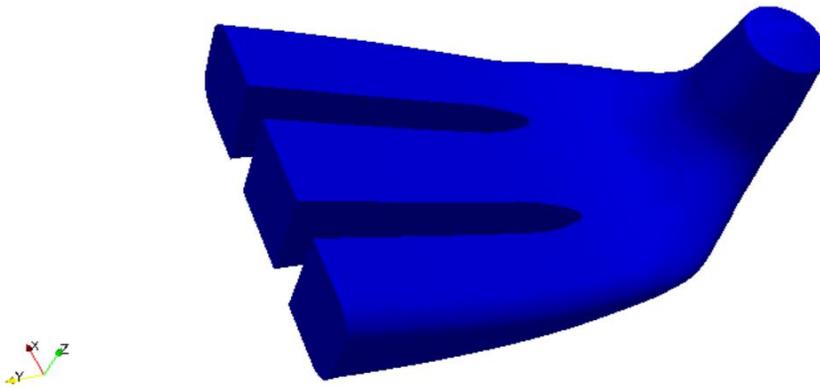


Figure 1

Following are the initial conditions that you will need to solve the problem:

Velocity(inlet) = 5.5 m/s;

Pressure(Outlet), $P = 1$ atm;

Density = 1000 kg/m³;

$\nu = 8.9e-4$;

Include plots and pictures in the report wherever necessary