

CFD Simulation of Conical Draft Tube

ABSTRACT

This report aims to describe the flow behavior and calculation of flow parameters in an Conical Diffuser or Divergent draft tube at elevated head of turbine using the software ANSYS CFX Mesh and OpenFOAM. It aims to find and study the Simulated results of Draft tube of a Turbine Design for a specified parameter. The Draft tube is a Conical Diffuser or Divergent with varying cross section. It consists of a conical diffuser with half angle generally less than equal to 10° to prevent flow separation. 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

Problem Statement

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

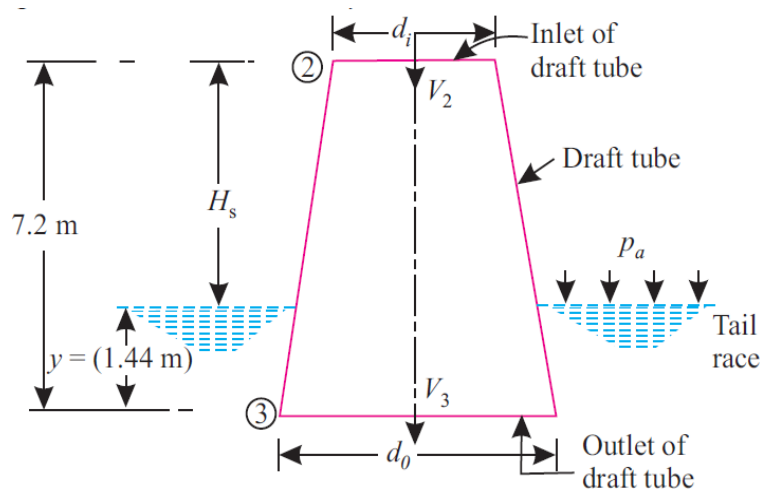


Figure:1

The Conical Draft Tube with following parameters was analyzed in this project.

1. Inlet Diameter= 1.2 m
2. Outlet Diameter= 1.8 m
3. Draft Tube Length=7.2 m
4. Length/Diameter= $L/D=6$
5. Divergence angle= 4.76°

References:

Text book of fluid mechanics and hydraulic machines by R.K Rajput.