Water Flow Simulation in a Elbow Draft Tube

Arpit Jain, IVth year, B.Tech., Mechanical Engineering
Maulana Azad National Institute of Technology, Bhopal (India)

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.

Keywords: Elbow Draft tube, OpenFOAM, icoFoam

I. INTRODUCTION

We are using an Elbow Draft Tube with horizontal splitters. The hydraulic reaction turbines are provided with draft tube to recover the part of kinetic energy coming out of runner. The energy recovery depends on the design of draft tube. The Draft tube is a Elbow with varying cross section, It consists of an extended elbow type tube. 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
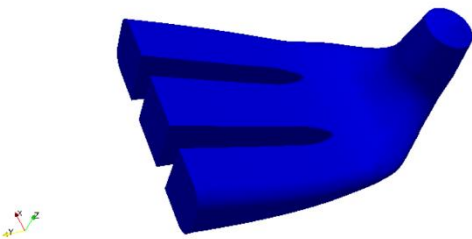

Figure 1: Draft tube

A. *Geometry*

The geometry of the Draft tube was created using the software ANSYS CFX V5. The elbow draft tube has three parts namely cone, elbow diffuser. The length of the leg of Draft Tube is 1.405 m(horizontal). Height of the Draft Tube from the cone to the base is 1.010 m. There are 3 splitters. Diameter of the cone is 0.365 m as shown in figure 2.
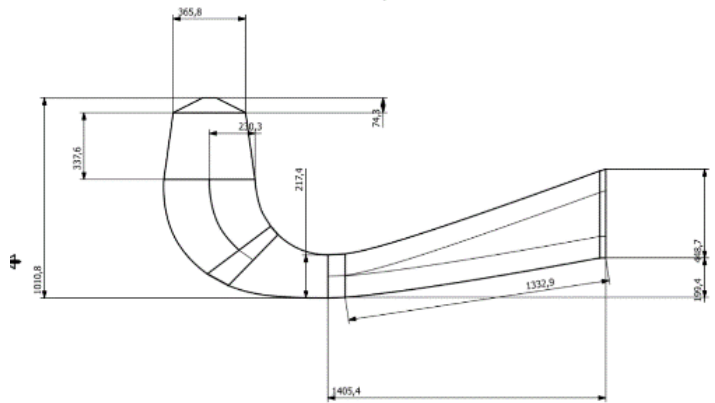

FIGURE 2: Geometry of the Draft Tube

B. *Meshing*

The meshing for this simulation was done using the ANSYS CFX v5 Mesh, The 3D mesh was done here using CFX mesh. For this certain geometry we require a coarse mesh for optimum results, so we have created a simple Tetrahedron mesh.
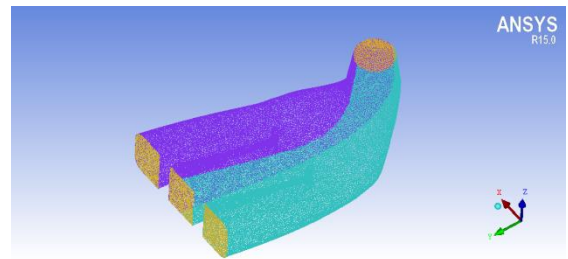

FIGURE 3: isometric view of 3D mesh

II. ANALYSIS

The CFD analysis of the water flow in an Elbow Draft tube was done using the icoFoam solver in OpenFOAM (v-5.0) software.

A. *Boundary Conditions*

Water enters the Draft tube after exiting the runner with a velocity of 5.5 m/s . The pressure at outlet was kept fixed at . The rest of the surfaces of the domain were defined as wall.

III. PROCEDURE

A. *ANSYS ICEM CFD software*

ANSYS ICEM CFD is a popular software package which provides advanced geometry /mesh generation as well as mesh diagnostics and repair functions useful for in-depth analysis. This was used to make the Geometry and Mesh.

The Mesh was imported to OpenFOAM through *fluent3DmeshToFoam* command from ".msh" format file. The casefiles were prepared, the solution was initialized for desired Velocity/ Pressure value.

The material is set to water. The numerical and solver settings are set to default values. The solver used here is ICOFOAM solver.

B. *ICOFOAM Solver*

Incompressible, laminar flow of icoFoam is transient solver for Newtonian fluids. icoFoam solves the incompressible laminar Navier-Stokes equations using the PISO algorithm. The code is inherently transient, requiring an initial condition (such as zero velocity) and boundary conditions. The icoFoam code can take mesh non-orthogonality into account with successive non-orthogonality iterations. The number of PISO corrections and non-orthogonality corrections are controlled through user input. The icoFoam solver has an advantage of being able to solve for higher courant numbers (> 1).

C. *RUNTIME SETTING*

The runtime setting were altered from the *ControlDict* file. The deltaT was se to 1e-5 for the interval of 0 to 1, this was done as the geometry is complex this was done to adjust the Courant number. The results of the transient solver settled after around 3000 iterations.

D. *POST PROCESSING*

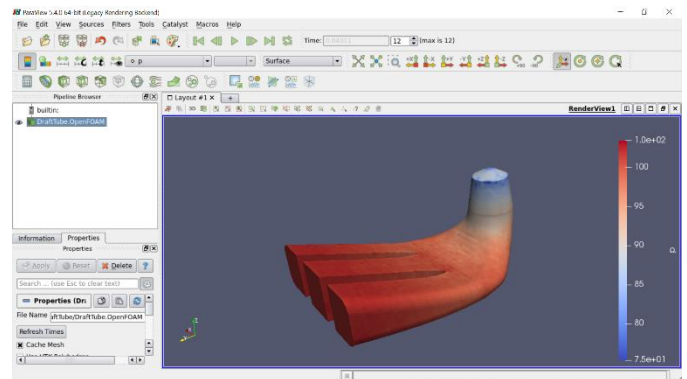The iterations are done and the project is opened in ParaView to infer the results.

Figure 4: ParaView Interface

The Residuals Graph was plotted using GNUPLOT application on Linux terminal.
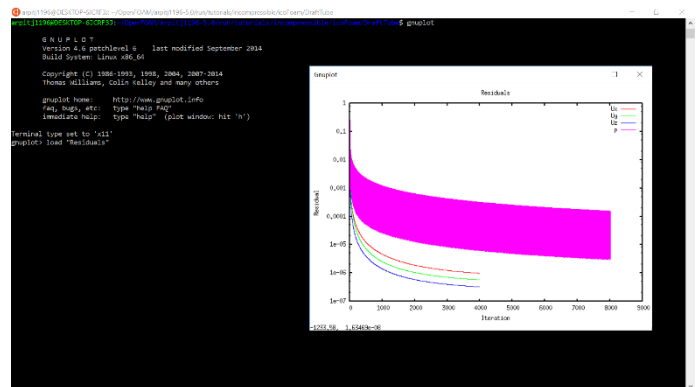

Figure 5: Gnuplot Interface

IV. RESULTS AND DISCUSSION

A. *CONTOURS AND RESIDUALS*

The Velocity magnitude & pressure values were key variables. We obtained the required flow parameters against the simulation time is as follows:
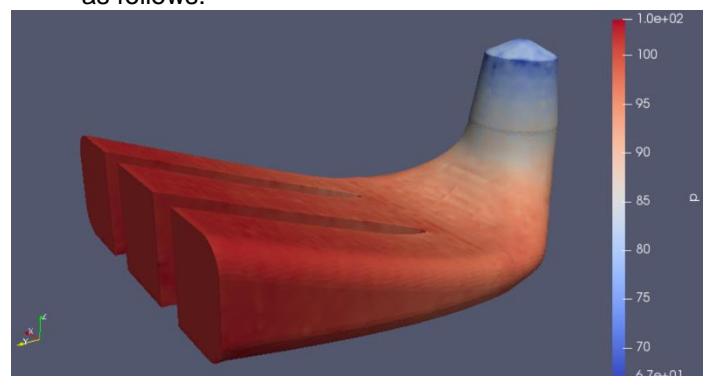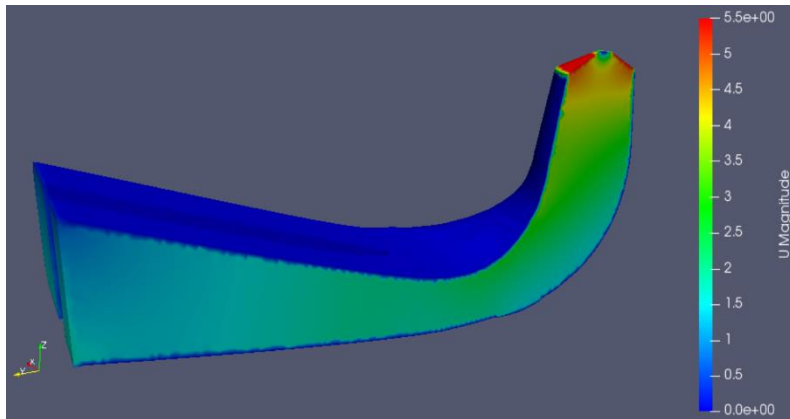

Figure 6: Pressure Contour

Figure 7: Velocity Contour

B. *INFERENCE FROM RESULTS*

- The pressure increase from inlet to outlet, and the velocity decreases as the water flows along the Draft Tube.

- The horizontal splitters do their work right of distributing and directing water in a smoother flow.

- The results show that our model is true to the theory of a Draft Tube which tube to recover the part of kinetic energy coming out of runner.

C. *IMPROVEMENTS POSSIBLE*

- We have used fixed value inlet velocity, which is not exactly what we find at a Francis turbine runner outlet. So further introducing a swirl to the inlet flow & specifying the mass flow rate at inlet would give a much accurate flow simulation.

- The Timestep i.e. deltaT can be reduced to 1e-3 or 1e-2 for faster results with decent courant number.
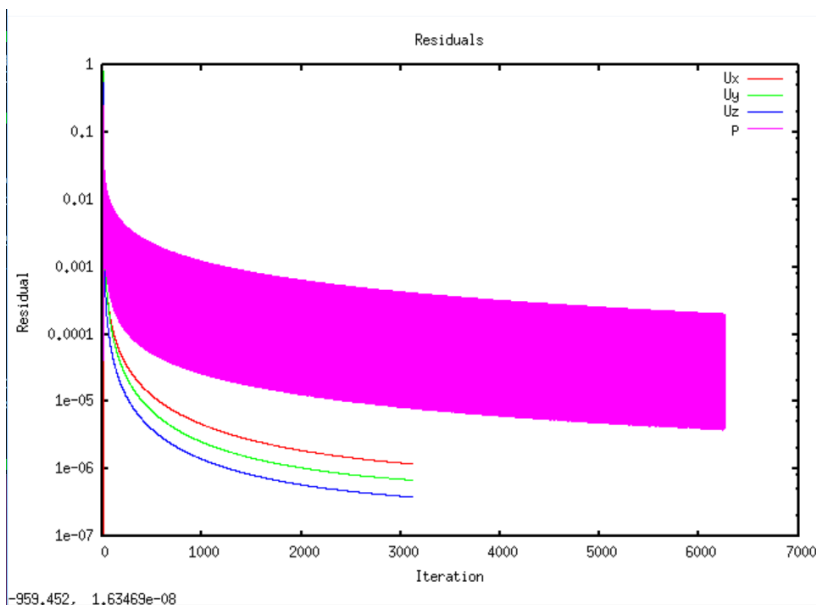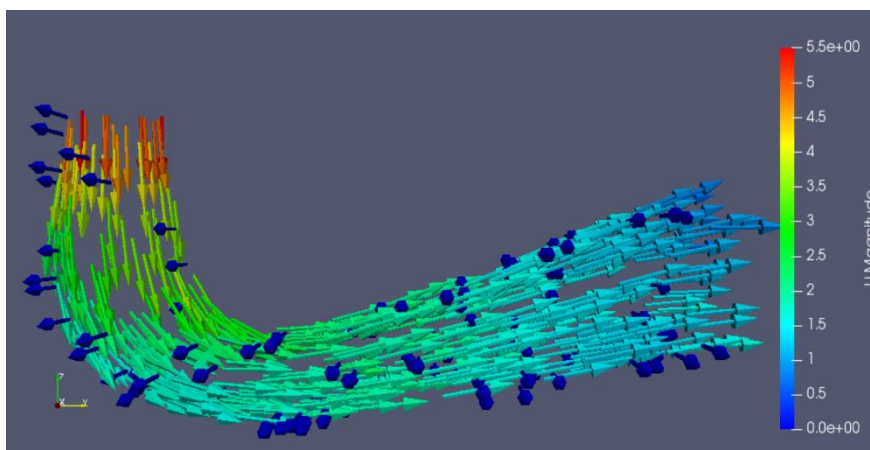

Figure 8: Residuals Graph

Figure 9: Velocity Vector Glyphs (side view)