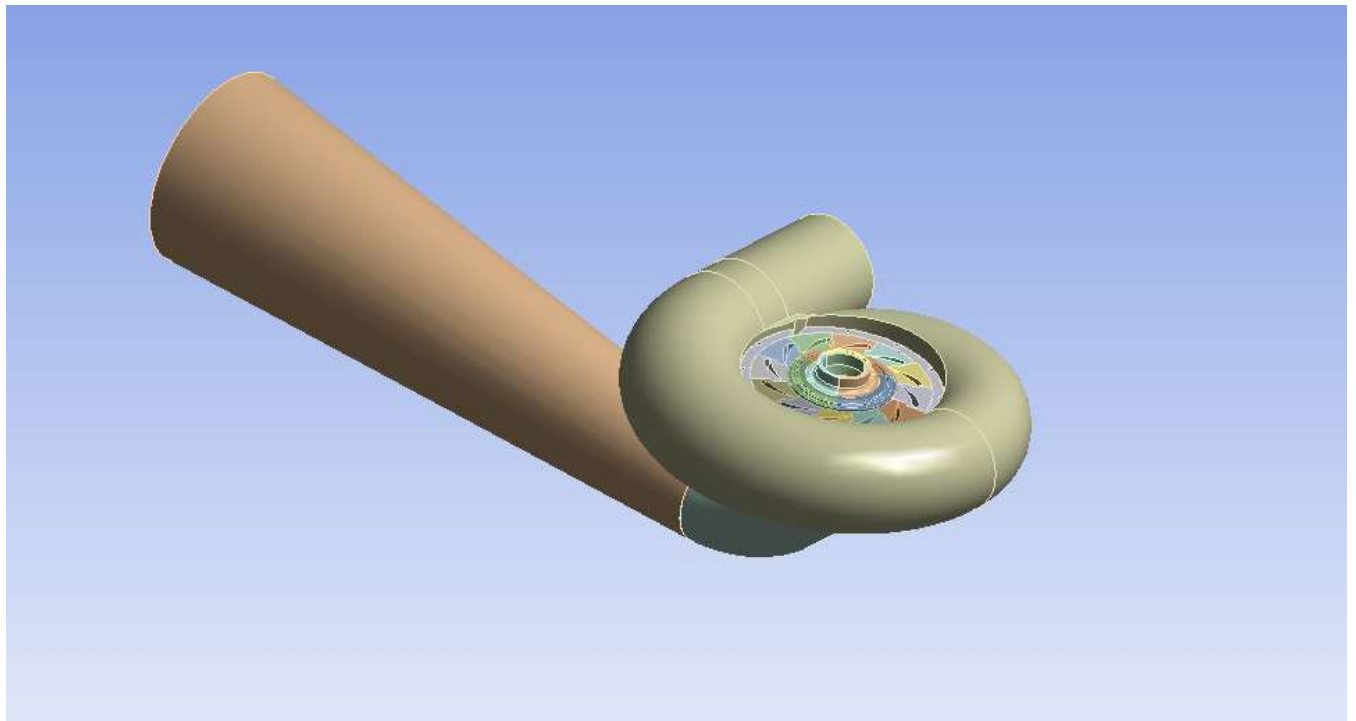# CFD Final Project Report

COLORADO SCHOOL OF MINES

Authors: Anastasia Candelaria, Amneh Jaber, Basanta Rijal, Nikhil Tiwari, and Richard Aipperspach.

COLORADOSCHOOL**OF**MINES
EARTH • ENERGY • ENVIRONMENT

Contents

Figures

Tables

Abstract

At the Colorado School of Mines in Golden, Colorado an introductory course in Computation Fluid Dynamics (CFD) is taught where students learn the fundamental of CFD. Within this course students must deliver a team design project of a practical fluids problem of interest. Using CFD techniques learned from class as well as independent research, students are tasked with creating a CFD model, validating this model, and analyzing the practical fluids problem of interest.

In this class project, a CFD analysis was performed on a 500 kW Francis turbine to study the effects of a variation in mass flow and pressure. The geometry and validation data used for this study was obtained from the corresponding author of a previous academic study. In the first stage of the study pre-processing was done which included both the meshing and set-up of the model. The mesh included creation of the mesh, mesh generation, and validating mesh independence. The set-up process included selecting the method of solver, turbo set-up, solver set-up, and selecting the material properties. The second stage of the project post-processing was done which included model validation, CFD solutions analysis, and an analysis of the results.

The results determined that by increasing the mass flow at the inlet, a higher velocity was generated inside the turbine, but at a price of non-uniformity in the spiral casing. When the mass flow inlet was decreased, a smaller velocity and more uniformity in the spiral casing was produced but at a price of adding more vorticies and spiraling inside the draft tube, where cavitation increased. The next part of the study determined the effects of pressure on the turbine flow. The outlet pressure was changed to simulate a change in different exit conditions and overall pressure head. When the the outlet pressure was increased, it was observed that it created more vortices and swirling inside the draft tube. Overall, the turbine geometry that was provided was optimized for the specific inlet mass flow of 1460 kg/s and outlet pressure of 6.8 kPa, given to us in the study. If we were to increase the mass flow slightly, the turbine would operate efficiently but any large changes in the inlet or outlet conditions caused the flow in the turbine to become chaotic and reduce the overall turbine efficiency.

Introduction

Today hydropower produces more than 1,211 GW of electricity worldwide, which is approximately 20% of total electricity produced [1]. Hydropower is one of the most reliable, efficient, and clean energy sources available today. A critical component of that hydropower generation is the Francis turbine, which converts the energy contained in the water into mechanical rotary power to drive an electrical generator.

The Francis turbine is the workhorse of the hydropower industry and was developed by an American engineer named James Bichens Francis around 1855 [2]. The Francis type turbine is a reaction type turbine that consists of the main components; the spiral case, stay vanes, guide vanes, runner, and the draft tube as shown in Figure 1. The flow of water through a Francis turbine enters the turbine in a radial direction through the spiral casing and changes to the axial direction when making contact with the runner blades. To reach optimal efficiency the water should flow smoothly through the turbine. Flow rate is generally the limiting factor for a given pressure head as there is an increase in the pressure head, the Francis turbine requires an increased flow and conversely as head decreases, lower flow is required [3] .

Possibly the most significant advance in turbomachinery engineeing in the late twentieth and early twenty-first century has been the development and use of a wide array of CFD methods [4]. The goal of a CFD is to develop a comprehensive numberical system to simulate the detailed flow field in the turbomachine of interest.
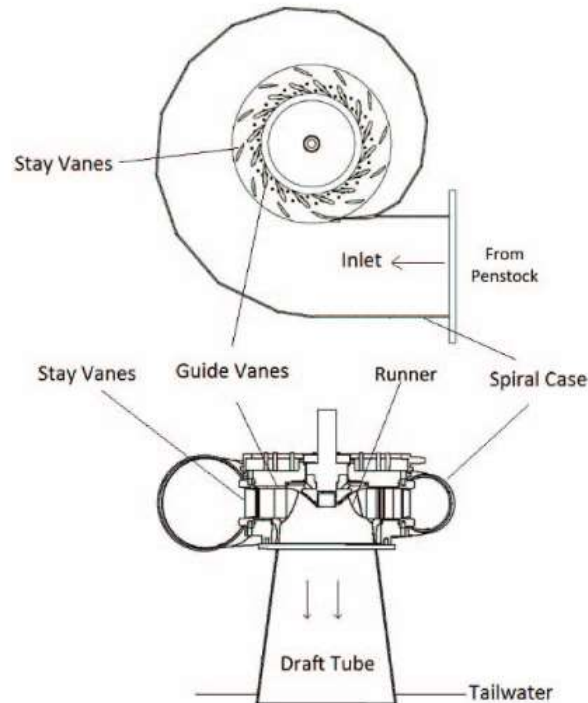

Figure 1: Main components of a Francis turbine [5]

Problem description

The Francis turbine is the most widely used hydro power turbine in the world with the highest efficiency rate. Francis turbines are normally designed with the aim to maximize hydraulic efficiency, minimum size, and to avoid cavitation [1]. Cavitation is the formation of empty space or cavities in liquid in areas of low pressure around turbine blades. To avoid cavitation it is critical to maintain water flow smoothly throughout the turbine. This can be challenging when the water level in the reservoirs are subject to large fluctuations resulting in subsequent changes in head pressures to the inlet of the Francis turbine. The Bureau of Reclamation recommends to recuce the chances of cavitation to avoid exceeding your rated head pressure by 125% or going lower that 65% of your rated head pressure as shown in Table 1 and *Figure 2*.

This study has three main objectives. One, is to ensure that the CFD mesh generated is independent and that the solution is truly converged and accurate. Two, is to validate the CFD model against the provided academic data to ensure the model's accuracy and reliability. The third and final objective is to examine the effects of a variation in design mass flow and pressure on the fluid flow throughout the turbine. This is critical to maintain a high efficiency and avoid cavitation.

Table 1: Determination of min and max head pressure ranges

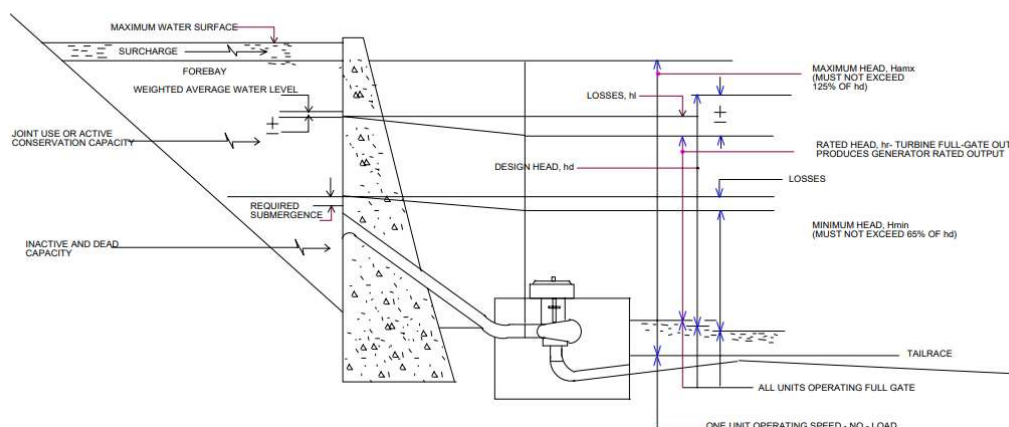| Type of turbine	Maximum head (percent)	Minimum head (percent)
Francis	125	65
Propeller – fixed blade turbine	110	90
Propeller – Adjustable blade turbine	125	65

Figure 2: Head min and max ranges

Geometry

Turbine Geometry

The team utilized an optimized geometry provided by Dr. Sara Aida Rodriguez Pulecio, the assistant professor of the Research Group of Fatique and Surfaces at the University of Valle, Columbia. Dr. Rodriguez along with Leonel Teran and Francisco Larrahondo used this same geometry in their academic publication, *Performance improvement of a 500-kW Francis turbine based CFD*. The geometry was developed using a 3D scanning process that had a dimensional accuracy of 0.38mm [5]. The optimized geometry has the guide vane openings at 68.57%, this geometry was proven as the geometry that optimized the turbines overall efficiency and was validated by comparing the CFD results to experimental results. With the provided geometry our team has successfully uploaded the .x_t file to ANSYS Workbench 19.1 as seen in Figure 2: Francis Turbine geometry.
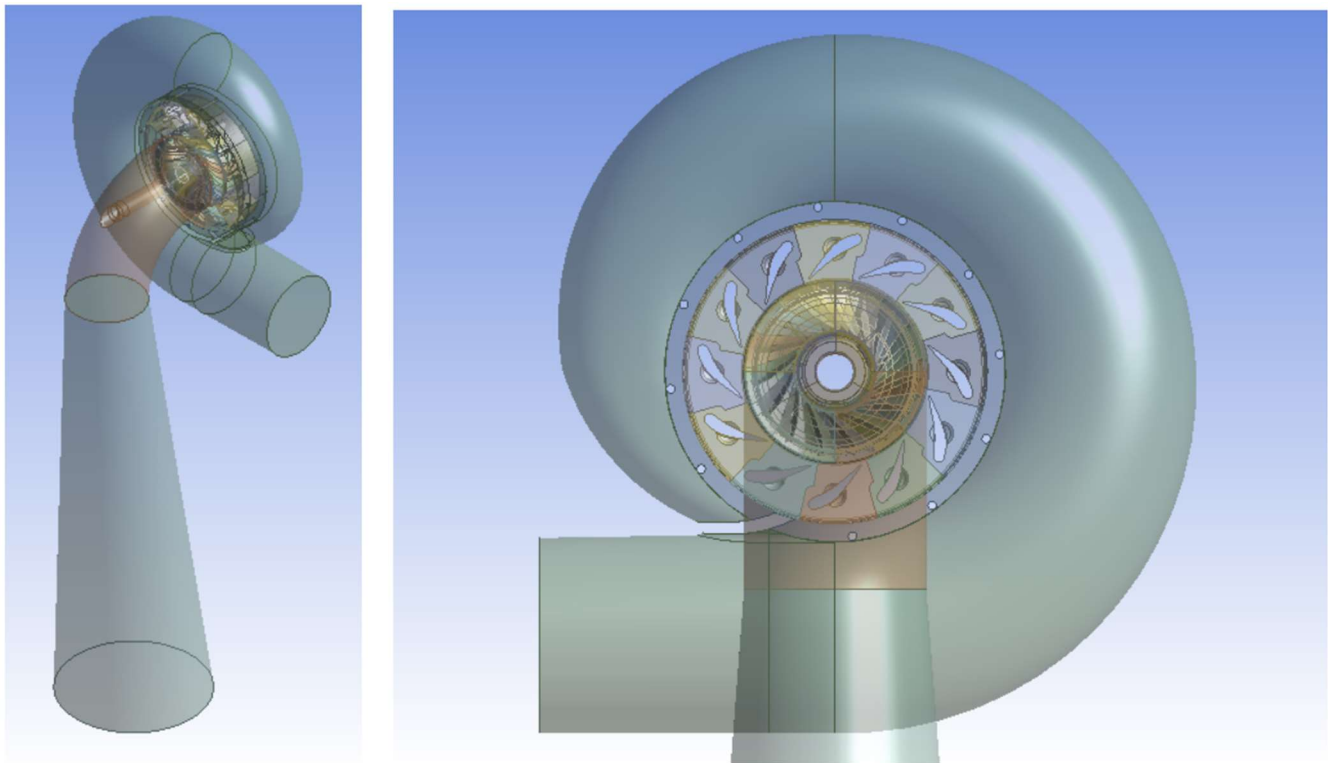
Figure 3_Turbine Geometry

Interfaces & Named Selections

The next step to improve our geometry required us to create name selections for the different components and interfaces within our system. We divided our geometry into three separate components. There are two stationary components and one rotational component. Stationary 1 consists of the spiral casing and the guide vanes. Stationary 2 consists of the draft tube. The Rotational component consists of the turbine hub,

runner blades and all the rotational fluid. Then we defined the inlet and outlet faces. These components are shown in Figure 4.

Figure 4_Turbine Components

After defining the three components we then defined the interfaces within these components so our simulation would allow the correct fluid flow throughout the turbine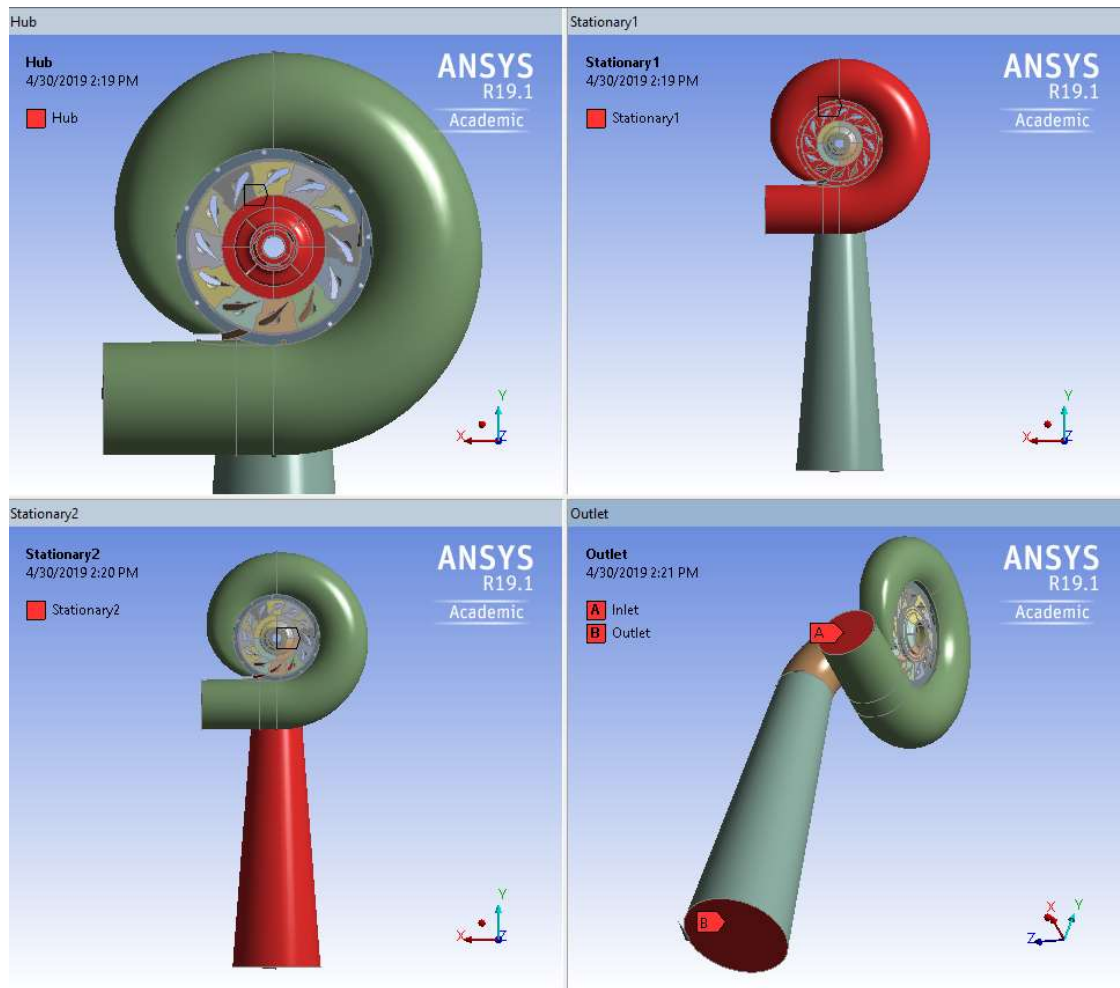. There are twenty total interfaces in the model. Fourteen of the interfaces are in the Rotational component. These interfaces allow communication and flow through each separate parts in the model, since the Rotational component is made up of multiple parts instead of one complete piece. The remaining six interfaces are between the other components of the model. There are two interfaces that allow communication and fluid flow between the spiral casing and the guide vanes, another interface is between the guide vanes and the Rotational component as well as an interface that allows flow from one guide vane blade to another guide vane blade. The remaining two interfaces are located between the back of the hub and Stationary 2 (start of the draft tube), and the other interface is located in Stationary 2, where fluid flow moves from the curved part of

the draft tube to the straight part of the draft tube. These interfaces are very important in our simulation and are shown in Figure 5.
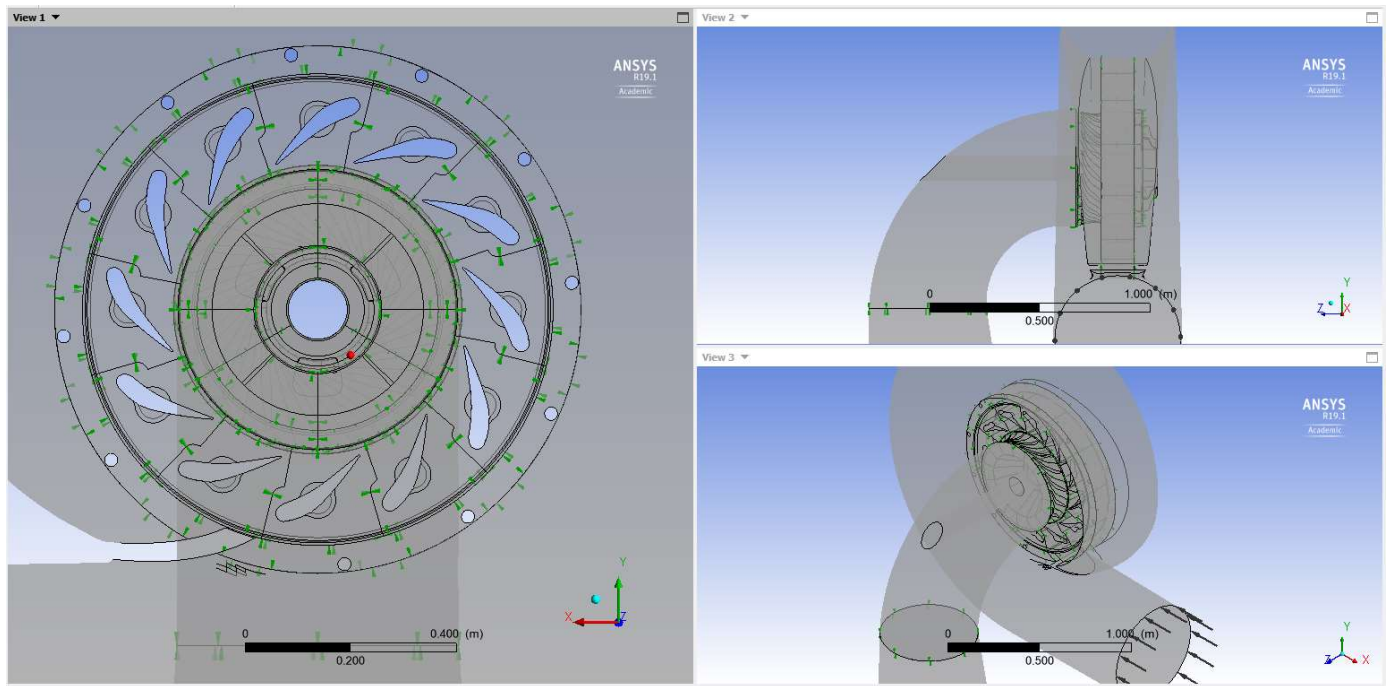

Figure 5_Turbine Interfaces

Pre-Processing

Meshing

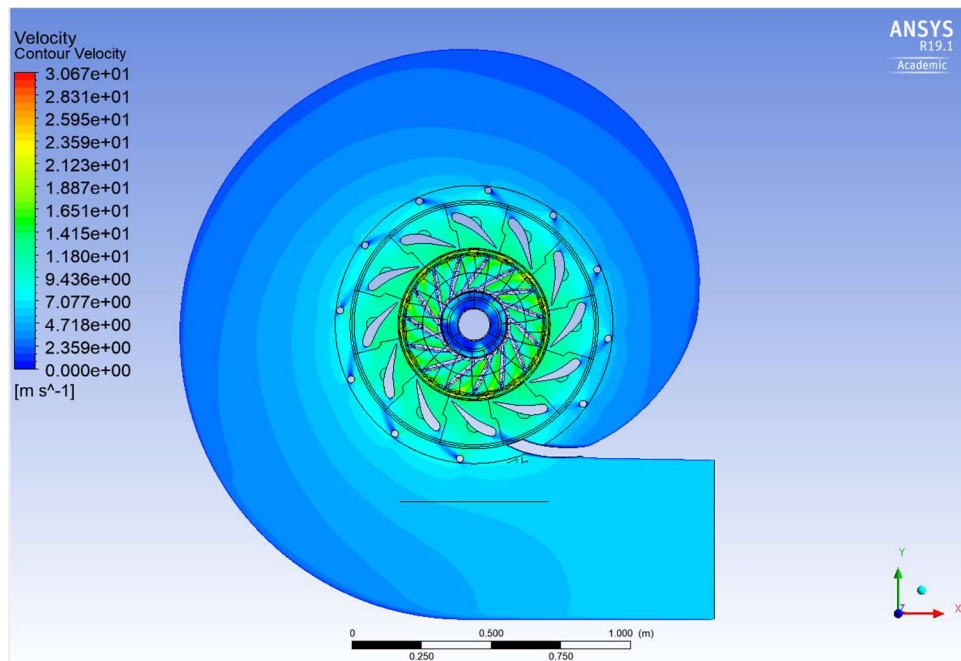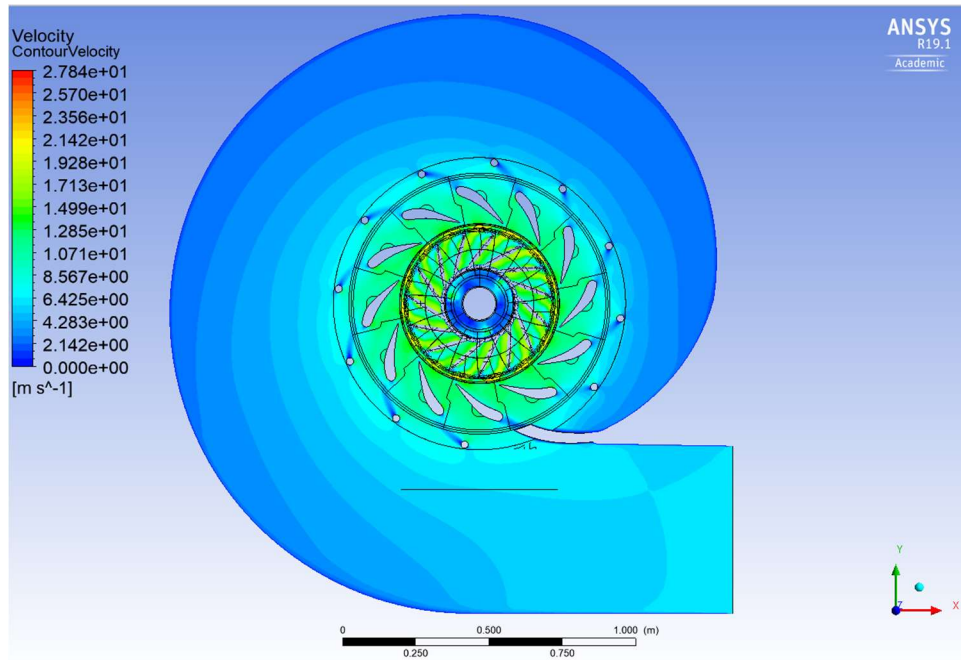### Creation of Mesh

Lorem.

Mesh Generation

Lorem.

Mesh Independence

Original:

1.5mm Mesh:

Dynamic Mesh:

Setup

Description of Methods

To create a 3-D numerical model of the Francis Turbine, ANSYS Simulation CFX Software was used. A steady-state operation in ANSYS CFX was used to obtain comparable results for the optimized geometry provided to us by Dr. Rodriguez. The study that we reproduced validated that a steady-state, turbulent model in ANSYS does indeed produce appropriate results for this simulation. For this project the turbulent model in ANSYS CFX solver used a numerical Finite Volume Method utilizing the turbulent Shear Stress Transport (SST) model. By using the mass inlet and pressure outlet boundary conditions we were able to accurately reproduce results and used this model to explore the effects of different boundary conditions on this geometry. Below are details regarding the Turbo Mode and Solver setup conditions.

Turbo Mode Setup

The next step in the CFX Analysis was the setup. We used the Turbo tool and selected the axial turbine machine type with the previous geometry to perform a steady state analysis. We had to use engineering judgement to select the correct boundary conditions, interfaces and solver settings to run this model. The axial turbine machine type was selected around a coordinate frame 0 and rotation around the Z-axis. We

decided to use a Steady-State analysis based on the conclusions in the research paper that determined that the Steady-State results are valid for this model [5]. This model used the fluid as water and the fluid flow in the turbine was turbulent, therefore we selected the Shear Stress Transport (SST) turbulence mode for the setup to match the selection in the previous study. The heat transfer is neglected in this model, so the energy mode is was turned off. The boundary conditions for this problem are defined in the published study, and it used a mass flow inlet, a pressure outlet and wall conditions for the flow around the rest of the turbine geometry for a modified geometry with the runner vanes that have a 68.57% vane opening [1]. These boundary conditions are adequate to run the ANSYS CFX solver (Inlet Mass Flow: 1430[kg/s] & Outlet Gauge Pressure: 6.8 [kPa]). The location of the inlet and outlet boundary conditions is depicted in Figure 6. For the rotational component of the setup we set the revolutions per minute (rpm) at -1000 rpm. The rpm is negative based on the coordinate system, and was calculated to be 1000 using the turbine equation as shown in Equation 1. The velocity used was the average fluid velocity inside the runner for a turbine of this size and the frequency correction is based on the area where the study was conducted, (Columbia: 60 Hz). This frequency is needed to correct the amount of rpm the generator can handle in this experiment for this location. After selecting these values for the Rotational component, the number of passages that interact with this rotational component were defined. There are 15 passages between the Rotational Hub and the Stationary Components, and these interface types are Frozen-Rotor to match the study [5]. This concludes the setup for the Turbo Mode, for more details on the setup see Table 2.
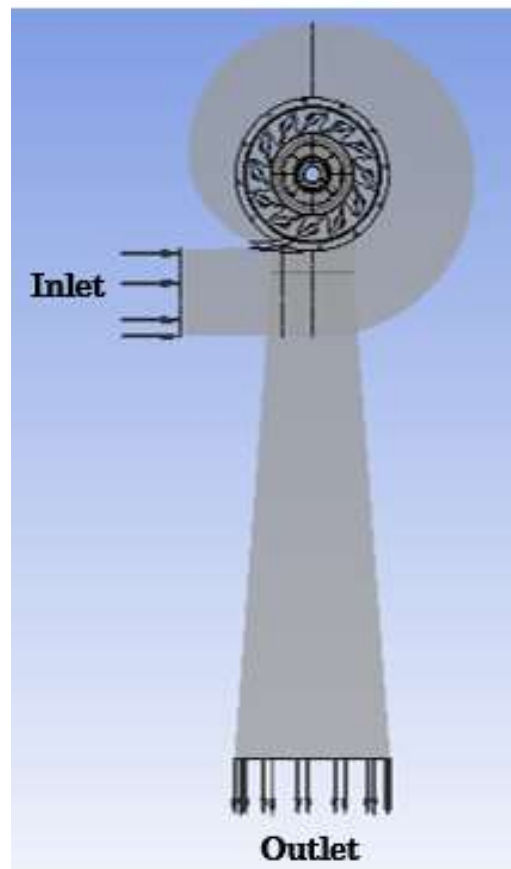

Figure 6: Inlet and outlet boundaries

$$f = 60Hz \qquad D = 0.507m \quad V = 35m/s$$

$$rpm = \frac{60 \; x \; V}{\pi \; x \; D} = 1318$$

$$Z_P = \frac{f \; x \; 50}{rpm} = 2.27 \sim 3$$

$$rpm_{adj} = \frac{f \; x \; 50}{Z_P} = 1000$$

Equation 1: RPM calculation

Table 2: Turbo Mode Setup Table

	Francis Turbine (Fluid CFX Analysis) - Turbo Tool			
Setup	**Basic Settings**	**Machine Type**	Axial Turbine	
		Coordinate Frame	Coord 0	
		Rotation Axis	Axis Z	
		Analysis Type	Steady State	
	Component Definition	**Type**	Rotating	Stationary
		Components	Hub	Stationary1 + Stationary 2 (Stay vanes & spiral case + Draft tube)
		RPM Value	-1000 (rev/min)	NA
		Passages/Alignment	15/15/15	NA
	Physics Definition	**Fluid**	Water	
		Reference pressure	0 (atm)	
		Heat Transfer	None	
		Turbulence	SST (Shear Stress Transport)	
		Inflow/Outflow Boundary Templates	Mass Flow inlet & P-static Outlet	
		Interface Default Type	Frozen Rotor	
	Interface Definition	20 Interfaces (Between each individual part of the geometry)		
	Boundary Definition	**Added Boundaries**	Inlet	Outlet
		Location	Inlet of Spiral Case	Outlet of Draft Tube
		Total Pressure	1430 (kg/s)	6.8 (kPa) (Gauge)
		Flow Direction	Normal to Boundary	NA
	Final Operations	General Mode		
	Finish			

Solver Setup

After selecting the correct setup for the Turbo Mode, we selected the correct setup for the solver. We chose to use High Resolution for the numerical scheme and turbulence solver. High Resolution settings solves the simulation using a second order method. This method is prefered over the other option of a first order Upwind as this setting will give us more accurate results and less error. For the convergence settings we set the max iterations to 500. After running a test simulation we discovered that the residuals converge around 150 iterations. Therefore, this selection of 500 iterations is acccurate for our simulation and different case values. For this project, our residuals consist of velocity, continuity, momentum and turbulence. These residual values need to be set at 1e-3 for convergence in the solver setup. Fore more information regarding the solver setup see Table 3.

Table 3: Solver Setup Table

Francis Turbine (Fluid CFX Analysis) - Simulation Settings					
Flow Analysis	Analysis Type	Steady State			
	Hub	Rotational Component: Checked			
	Draft Tube/Spiral Casing	Stationary Components: Inlet and Outlet Checked			
Interface	Interfaces Checked				
Solver	Solution Units		[kg], [m] , [s] , [K] , [rad] , [s]		
	Solver Control	Basic Settings	Advection Scheme	High Resolution	
			Turbulence Numeric	High Resolution	
			Convergence Control	Min Iteration	1
				Max Iterations	500
			Fluid Timescale Control	Timescale Control	Auto Timescale
				Length Scale Option	Conservative
				Time Scale Factor	1
			Convergence Criteria	Residual Type	RMS
				Residual Target	1.00E-36
		Equation Class Settings	Continuity / Momentum / Turbulence Eddy Dissipation		
		Advanced Options	Global Dynamic Model Control		
Materials	Standard Water (No Changes in Basic Settings or Material Properties)				

Post Processing

Post processing is the procedure of comparing and analyzing our results. The first step of post processing was to validate that our model was indeed, an accurate representation of an authentic Francis Turbine. In order for us to validate our model we compared our simulated results with the experimental results from the "Performance Improvement of a 500-kW Francis Turbine based on CFD" study. This study created an optimized geometry and validated their simulated results against experimental results. If we are able to validate our simulated results against these results, then we can show that our model has been validated. Once the model has been validated, the model then can be used to compare and analysis how mass flow and pressure effect the flow inside a Francis Turbine.

Model Validation

Using the above mentioned boundary conditions and setup we were able to validate our model against the experimental/simulated results from the study. Using the inlet mass flow and output pressure as boundary conditions we produced results with error that ranged from 0-7.22%. This error is for the accuracy at the inlet and outlet of the turbine. We believe that this small error is sufficient for validating our results. There will always be small errors in reproducing another model, but this error is small enough to conclude that the boundary conditions for our model are accurate. The experimental results that were given only showed the inlet conditions of the turbine and the study used these to obtain correct values for the outlet conditions. Our model accurately captures the inlet conditions and the simulated outlet conditions that were provided to us for validation. See Table 4 below for more details on the percent error for the boundary conditions.

Table 4: Validation percent error

	Experimental Inlet Velocity (m/s)	Experimental Inlet Pressure Total - (kPa)	Outlet Velocity (m/s)	Outlet Pressure Gauge - (kPa)
Study Results	5.6	407.95	2.51818	6.85
Validated Results (Propped at Max.)	5.755	437.415	2.61262	6.85
Error (%)	2.77	7.22	3.75	0.00

Aside from the boundary conditions, one of the main goals of our validation was to accurately capture the correct flow at the guide vanes and inside the Rotational component. The study shows that there is a maximum of around 18 m/s near the ends of the guide vane blades and a minumun of around 2 m/s the closer you get to the 'connection' walls where the spiral casing is connected to the stay vane component. To show this we created an XY contour plot of the guide vanes as shown in Figure 7. Our contour plot matches the values of the study where the max velocity is 18m/s at the end of the guide vane blades and a minumun of 2m/s near the connection points. Overall, our model is a proper representation of the Francis Turbine for this optimized geometry.
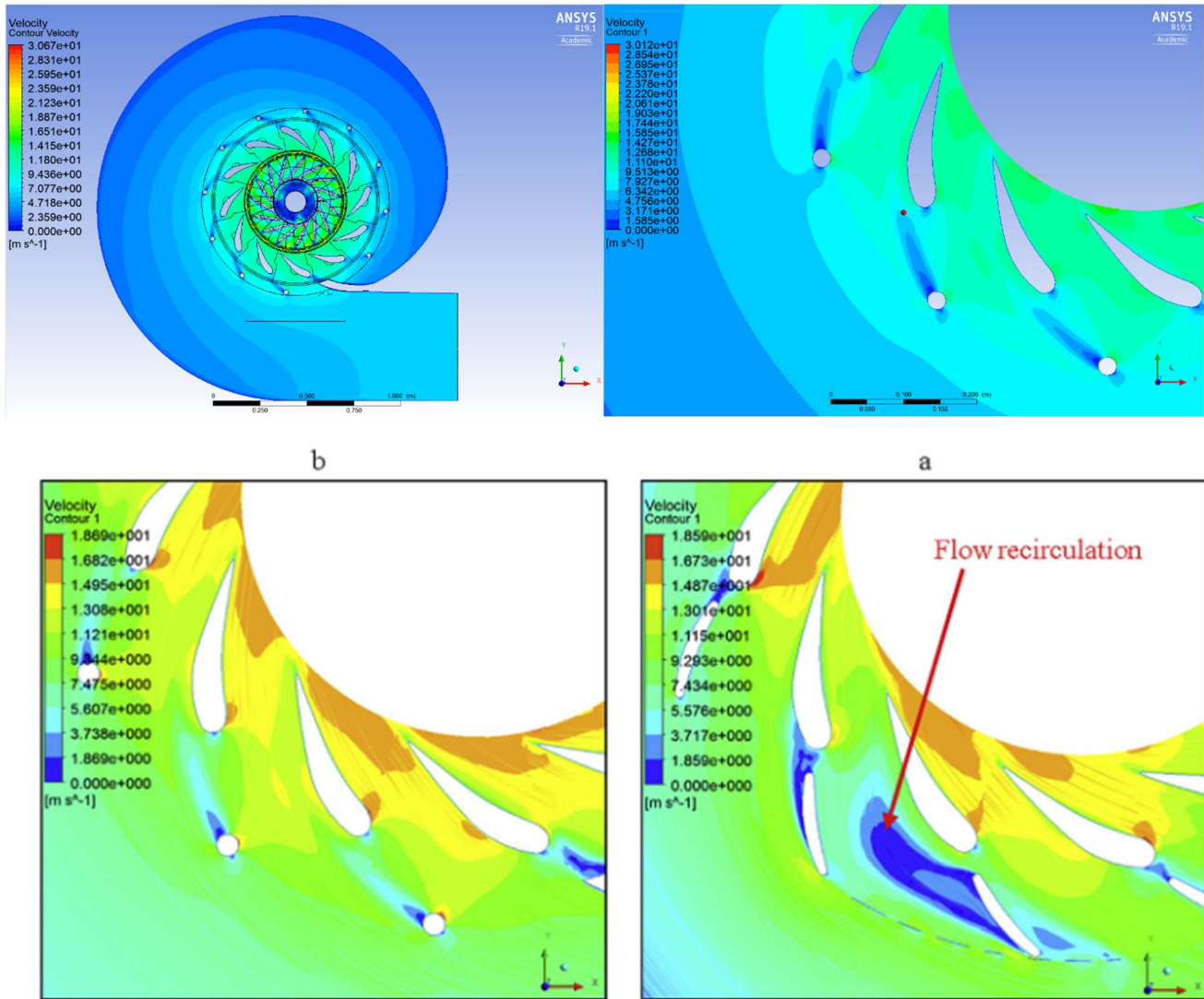
Figure 7: Contour plots of guide vanes

Results

Validated Case

Following the confirmation of our model validation, we were able to analyze our results for these boundary conditions. Figures 8-11 depict the contours, graphs and streamline plots for this simulation. As you can see the velocity profile in the spiral casing has very little eddy vortices as the flow enters the hub or rotating component, Figure 8. The streamlines show the fluid exiting the hub, entering the draft tube with small vortices, swirling, and exiting the draft tube with a uniform flow, Figure 9. This simulated flow is the desired flow profile for the optimized turbine geometry. The optimized pressure and velocity flow throughout the draft tube shows a smooth profile with no vorticities and swirling, Figure 10-11.
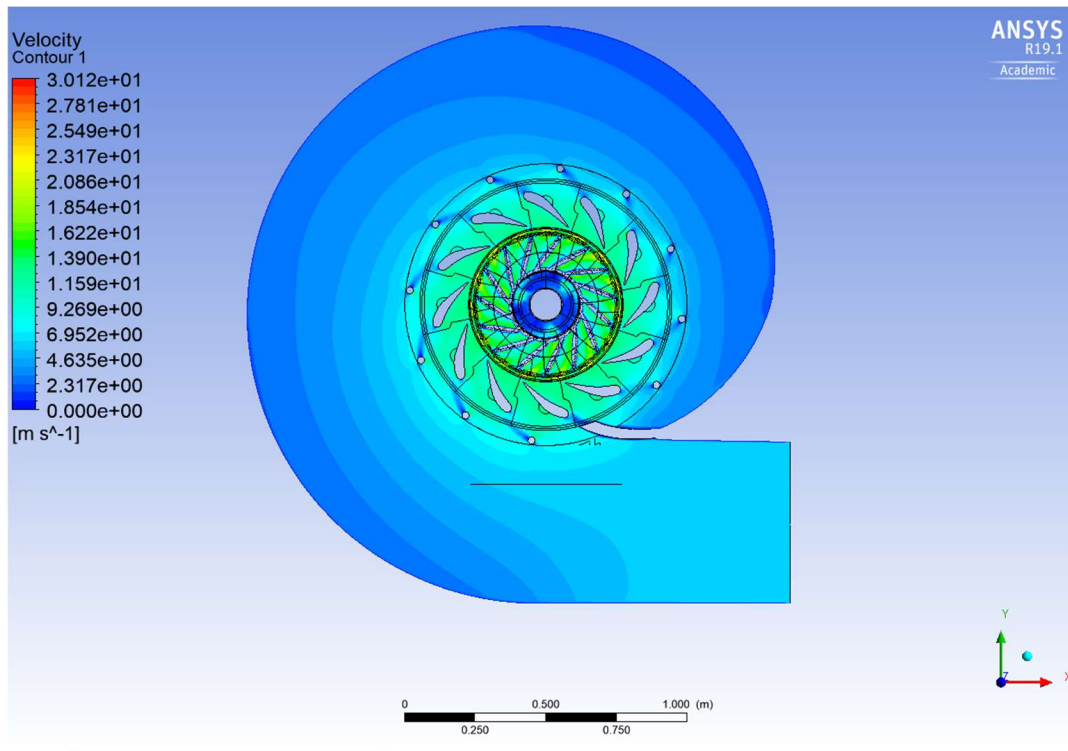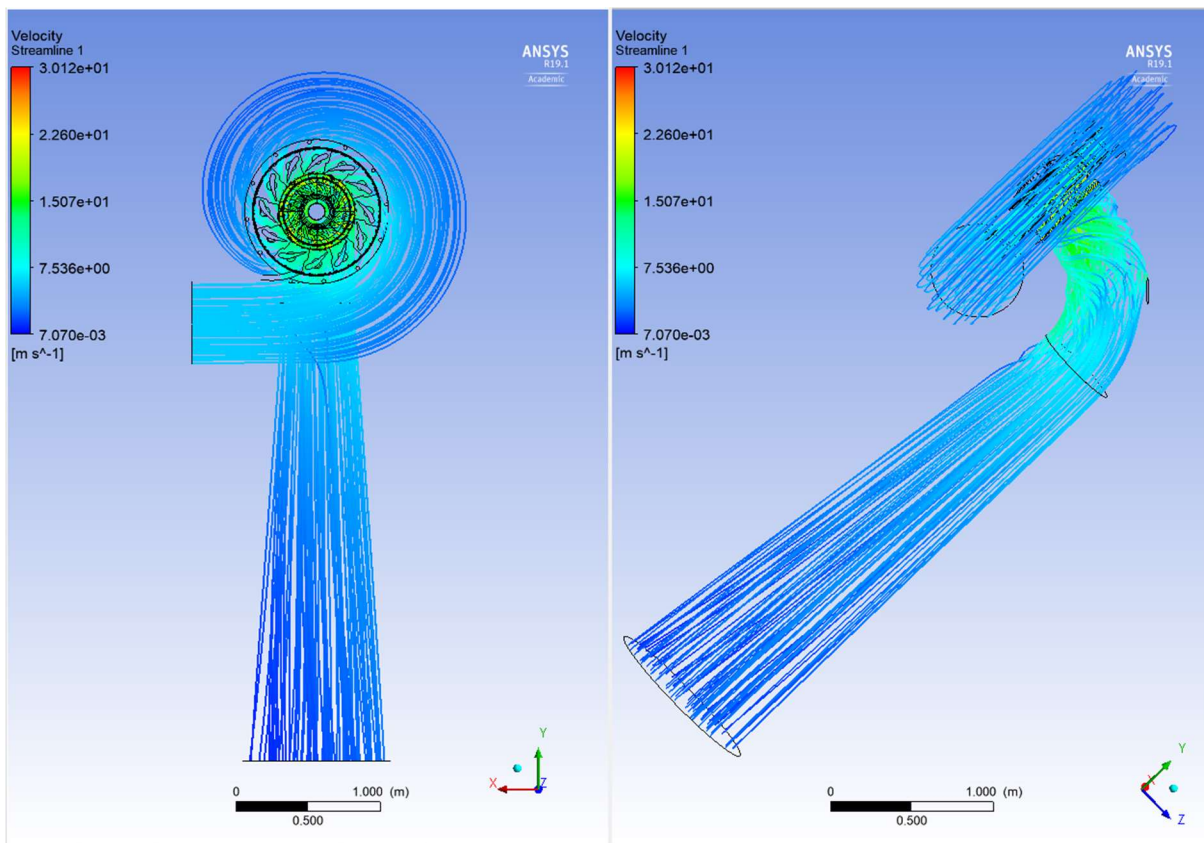
Figure 8: Velocity contour 1
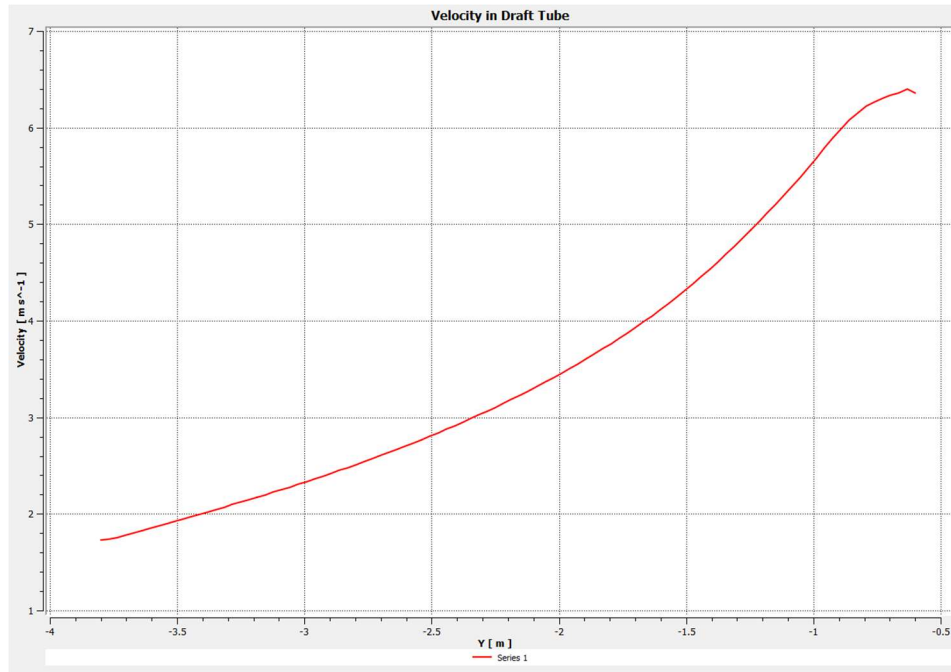

Figure 9: Streamline plot
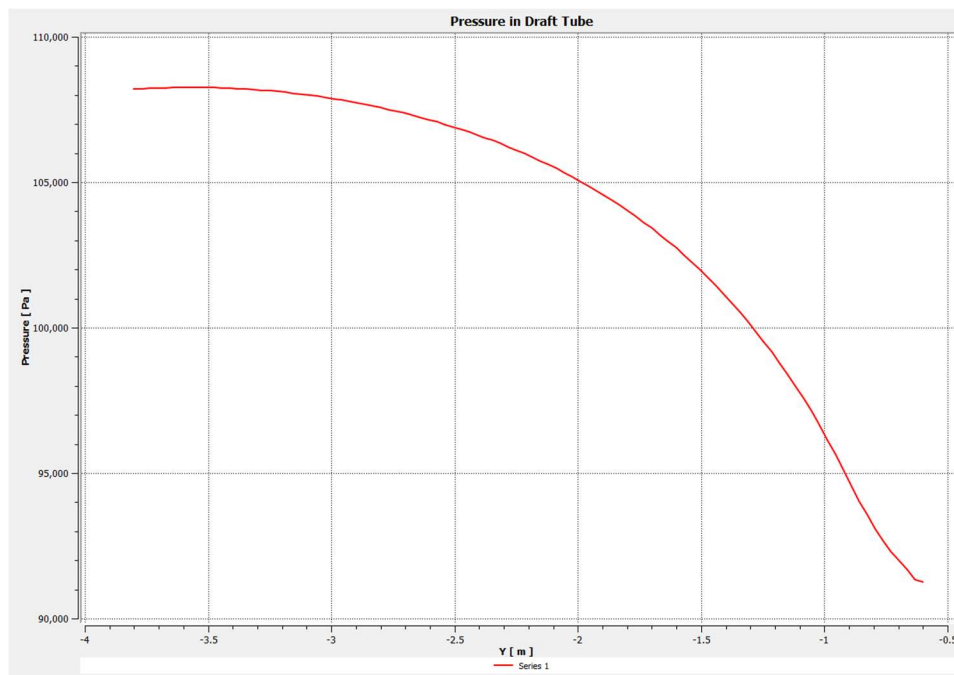
Figure 10: Velocity plot in draft tube

Figure 11: Pressure plot in draft tube

Mass Inlet Parametric Study

Now that our model is validated, we were able to run a parametric study on how different mass flow inlet values effect the flow throughout the turbine. We conducted two simulations. The first used the mass flow inlet of 2000 kg/s and the second of 1000 kg/s. This change in mass flow changes the inlet pressure from

the original 437 kPa to 517 kPa and 252 kPa respectively. Figures 12-15 depict the contours, streamlines and graphs of each study. Overall, if we increase the mass flow then the velocity inside the turbine also increases, Figure 12 and Figure14. The flow throughout the spiral casing becomes less uniform as well, since the turbine geometry was optimized for the original inlet and outlet conditions, Figure 12 and Figure 13. We can also see that when we change the inlet mass flow the outlet pressures also change. As we increase the mass flow the outlet pressure decreases and it increases as we decrease the mass flow, Figure 15.
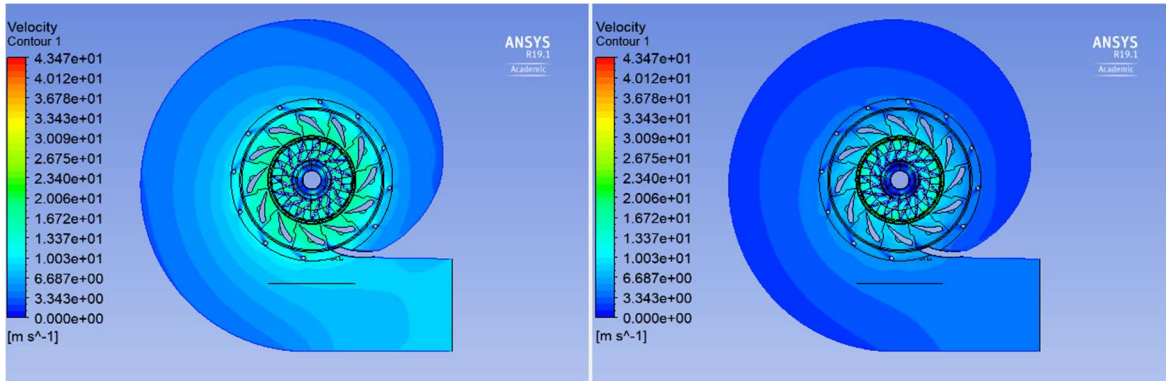

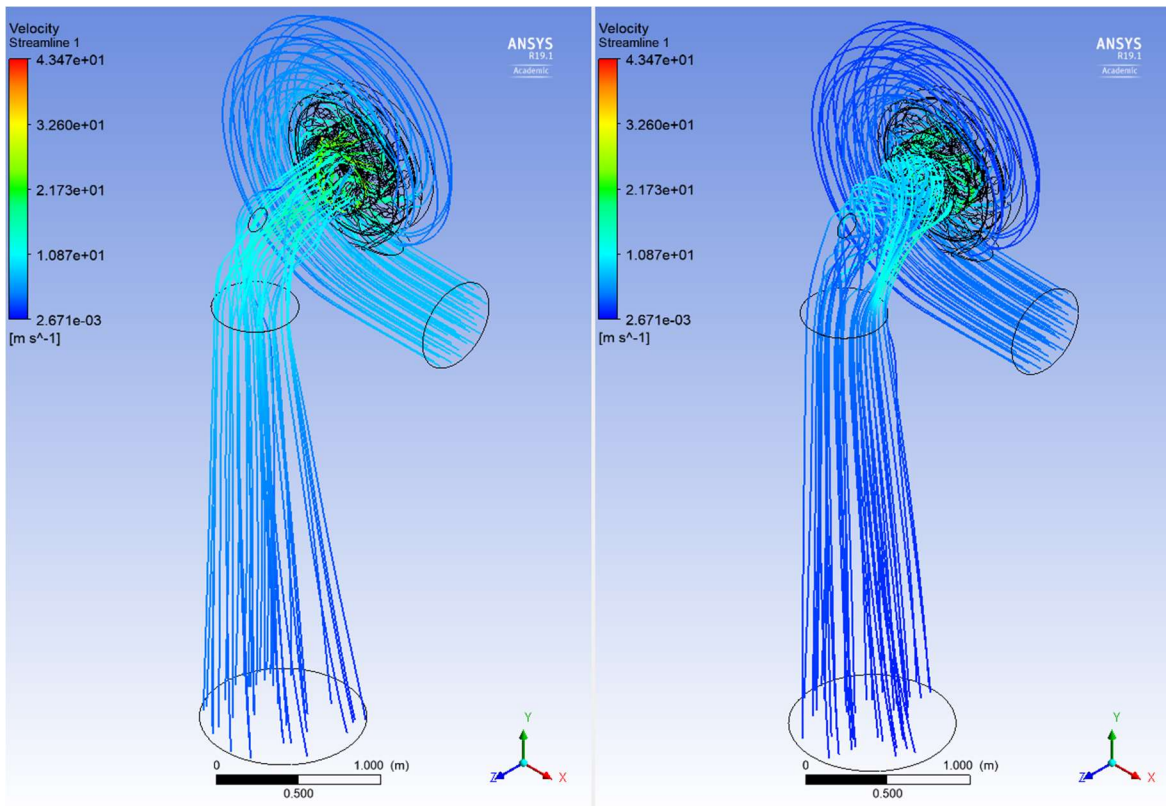*Figure 12: Velocity contours at 2000 and 1000 kg/s*

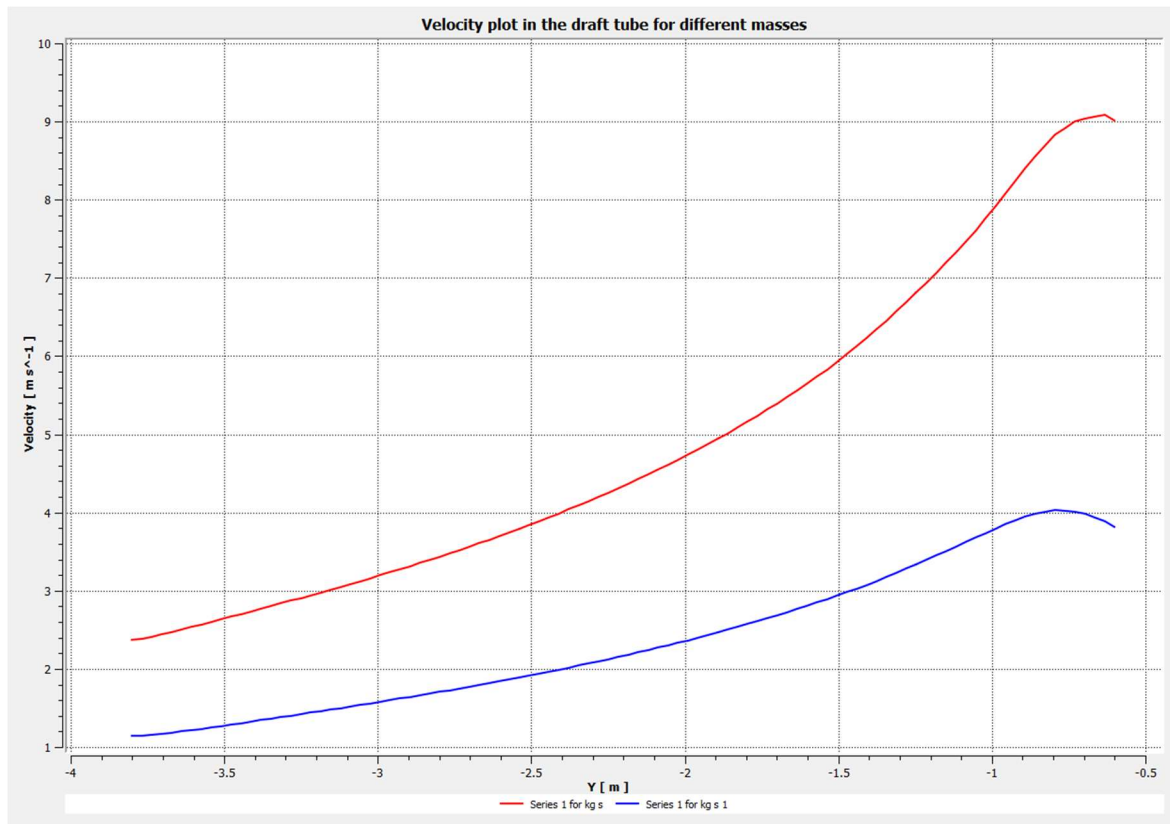*Figure 13: Velocity streamline plots at 2000 and 1000 kg/s*

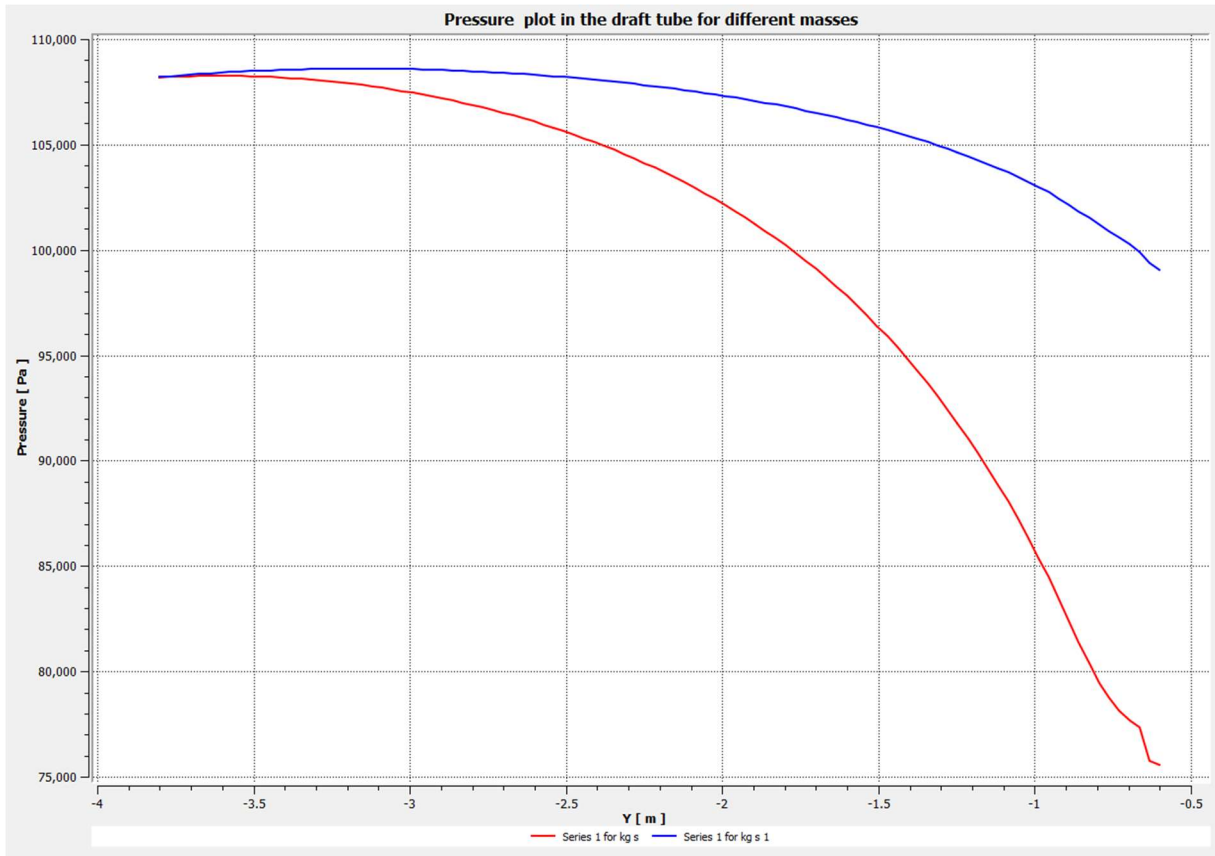*Figure 14: Draft tube velocity plots at 2000 and 1000 kg/s*

Figure 15: Draft tube pressure plots at 2000 and 1000 kg/s

Outlet Pressure Parametric Study

Now that our model is validated, we were able to run a parametric study on how different pressure outlet values effect the flow throughout the turbine. We conducted two simulations. The first used the pressure outlet of 1.62 atm and the second pressure outlet of 1.31 atm. Figures 16-19 depict the contours, streamlines and graphs of each study. As we can see from the results below, if we increase the pressure, it causes the internal flow to become more turbulent and develop more eddies and swirl inside the draft tube, Figure 17. The velocity and pressures in the draft tube becomes more chaotic and does not have a uniform profile, Figures 17-19.
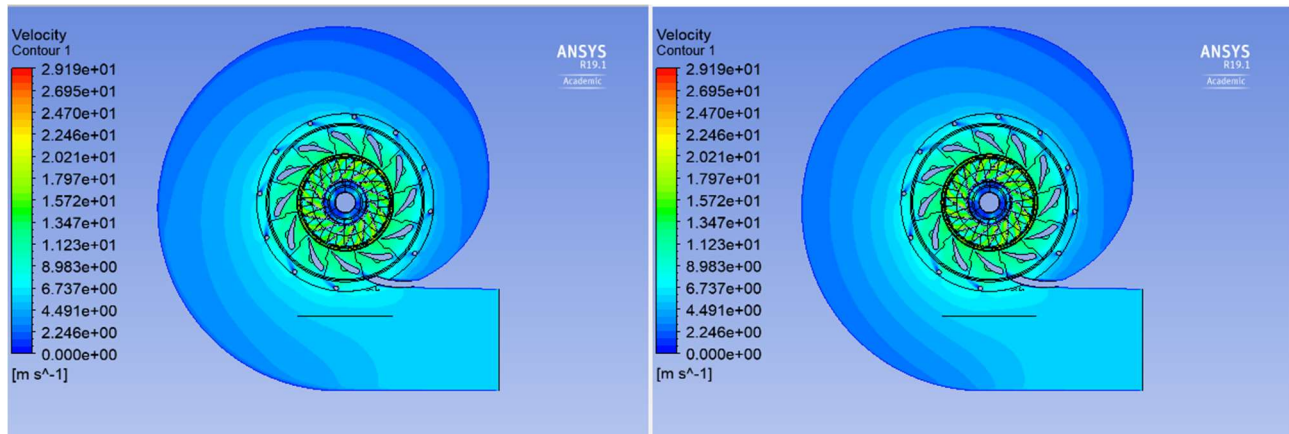
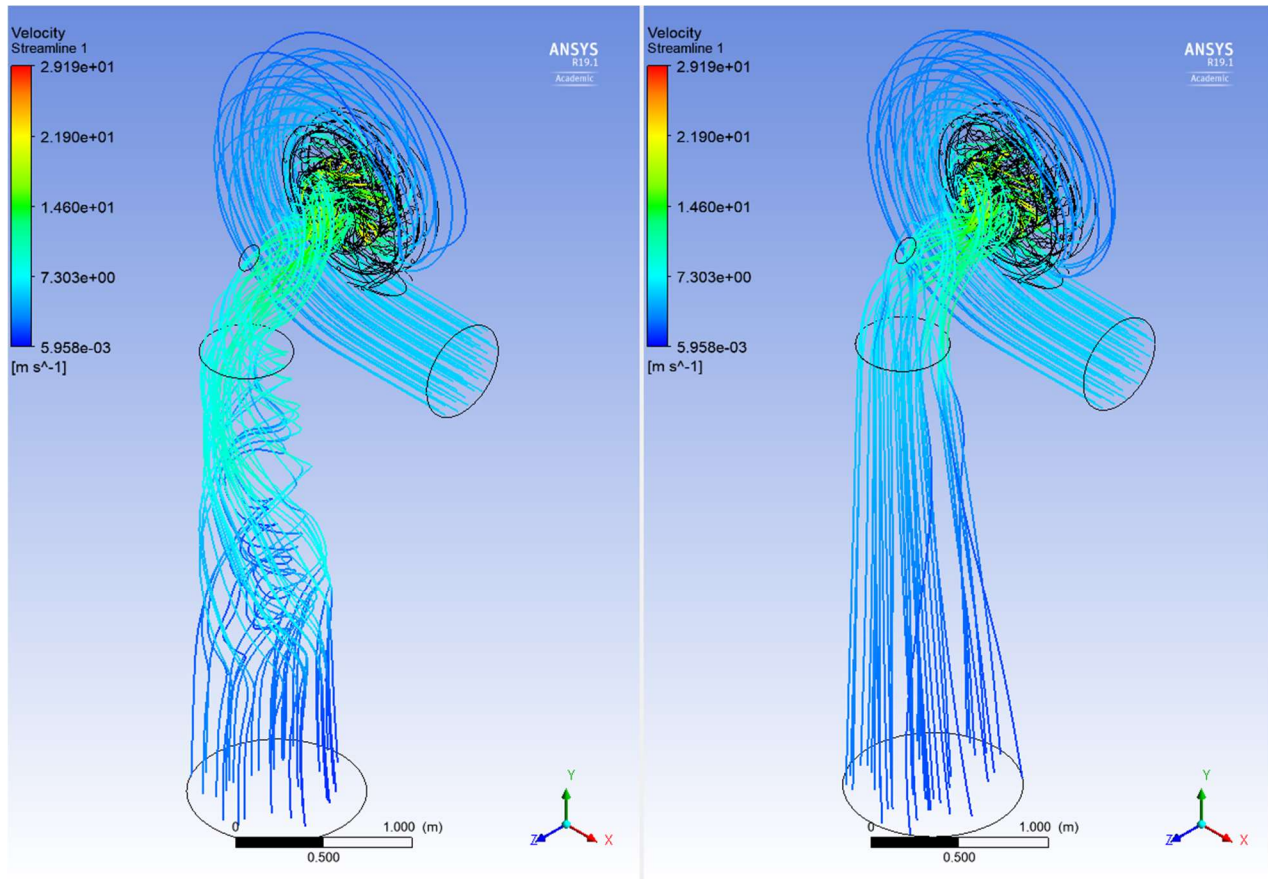*Figure 16: Velocity contours at 1.62 and 1.31atm*

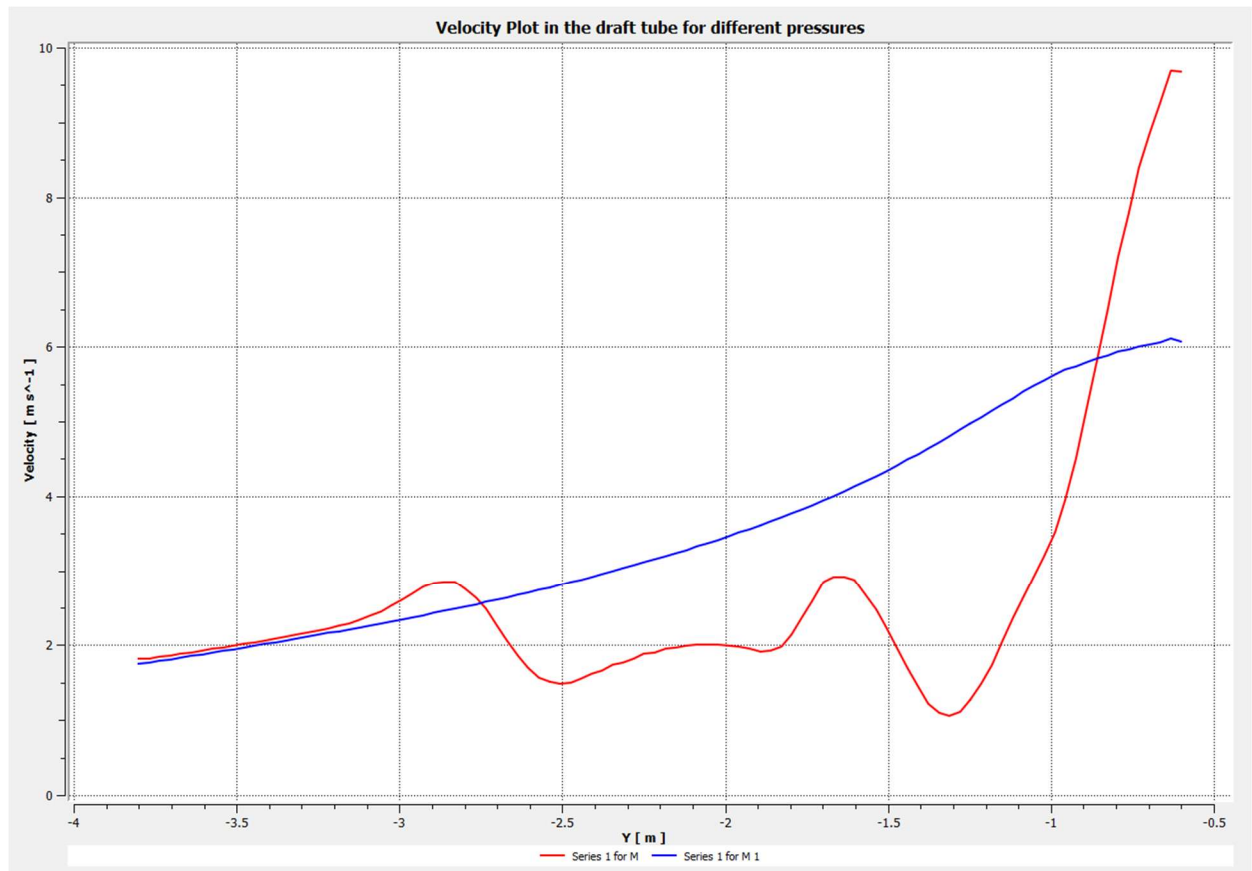*Figure 17: Velocity streamlines at 1.62 and 1.31 atm*

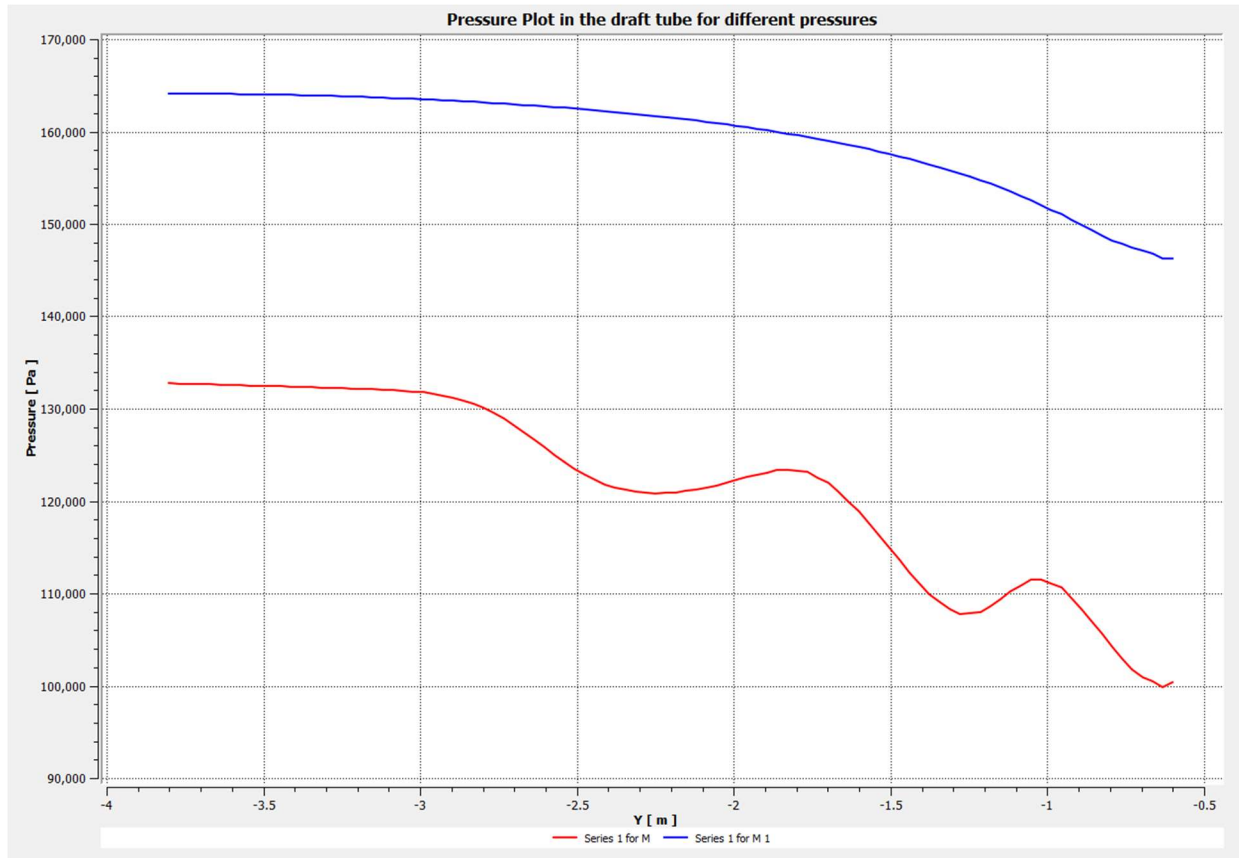*Figure 18: Draft tube velocity plots at 1.62 and 1.31 atm*

Figure 19: Draft tube pressure plots at 1.62 and 1.31 atm

CFD Solution Analysis

Computational solution analysis is an important part in CFD. The primary concern about the computational solution in the CFD are whether the solution can be guarnteed to approach the exact solution of the partial differential equation and, if so, under what circumstances [book]. Discritization of partial differential equation gives rise to system of algebraic equations consisting of finite quantities that depends on time (Δt) and space (Δx, Δy, Δz). But still we cannot directly establish the convergence. So, the following criteria are crucial to perform a numerical algorithm.

- Consistency
- Stability
- Convergence (residuals, relaxation factors)
- Accuracy (error analysis)
- Efficiency

Consistency

Consistency deals with the extent to which the finite difference equations approximate the partial differential equation. It is an important property and concerns the discretization of the partial differential equations where the approximation performed should diminish or become exact if the finite quantities, such as the time step (Δt) and

mesh spacing (Δx, Δy, Δz) tend to zero [book] i.e. the discretization of a partial differential equation should become exact as the mesh size tend to zero meaning that the truncation error should vanish. A numerical approximation is said to be consistent with the PDE if the exact solution to the PDE satisfies the algebraic equation obtained after discretization, at least up to first order in the discretization parameters[A].

Stability

In addition to consistency, another property that also governs the numerical solution method is stability. It deals with the growth or decay of the errors (truncation, round-off etc.) during computation [book]. An approximation is said to be stable if the error decays as we proceed from one step to the next step during computation. Stability also helps us to ensure that the solution does not diverges in context of iterative method. In general stability means that errors at any stage of the computation are not amplified but are attenuated as the computation progresses [A].

Convergence (residuals, relaxation factors)

Convergence of a numerical process can be defined as the solution of the system of algebraic equations approaching the true solution of the partial differential equations having the same initial and boundary conditions as the refined grid system (*grid convergence*) [book]. In other words, convergence means that the numerical solution converges to the exact solution as the discretization parameters tend to zero. According to the Lax's equivalence theorem consistency and stability are the necessary and sufficient condition for the convergence to be satisfied i.e. [book] Our residuals were able to converge successfully at 1e-3 and produce sufficient results, see Figures 20-21 for the convergence plots.
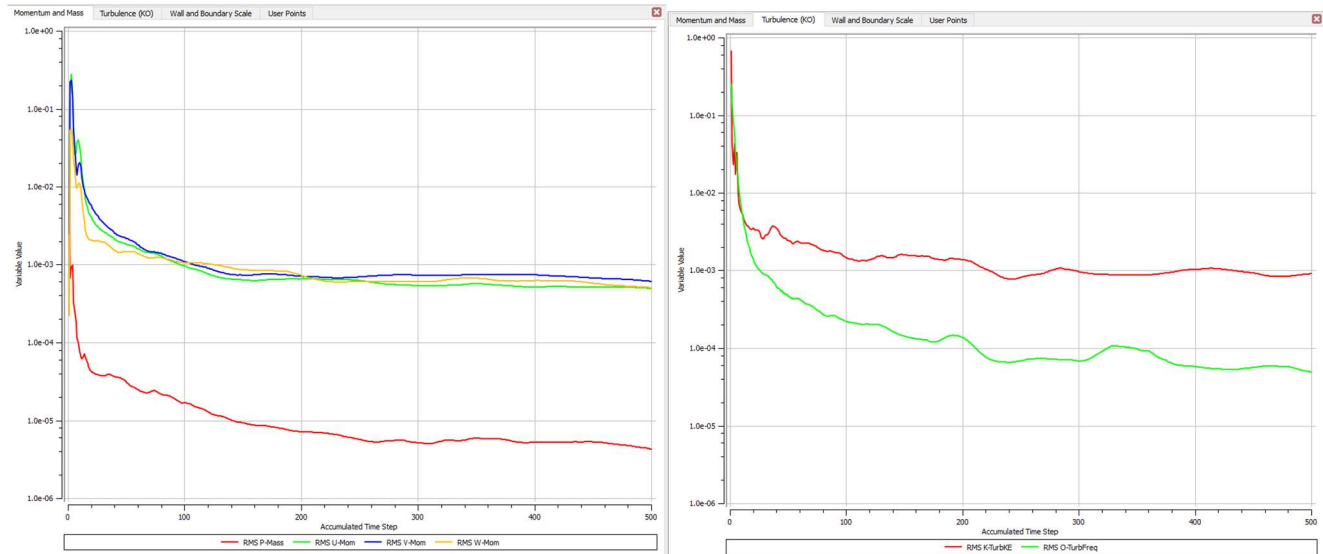
Consistency + Stability = Convergence
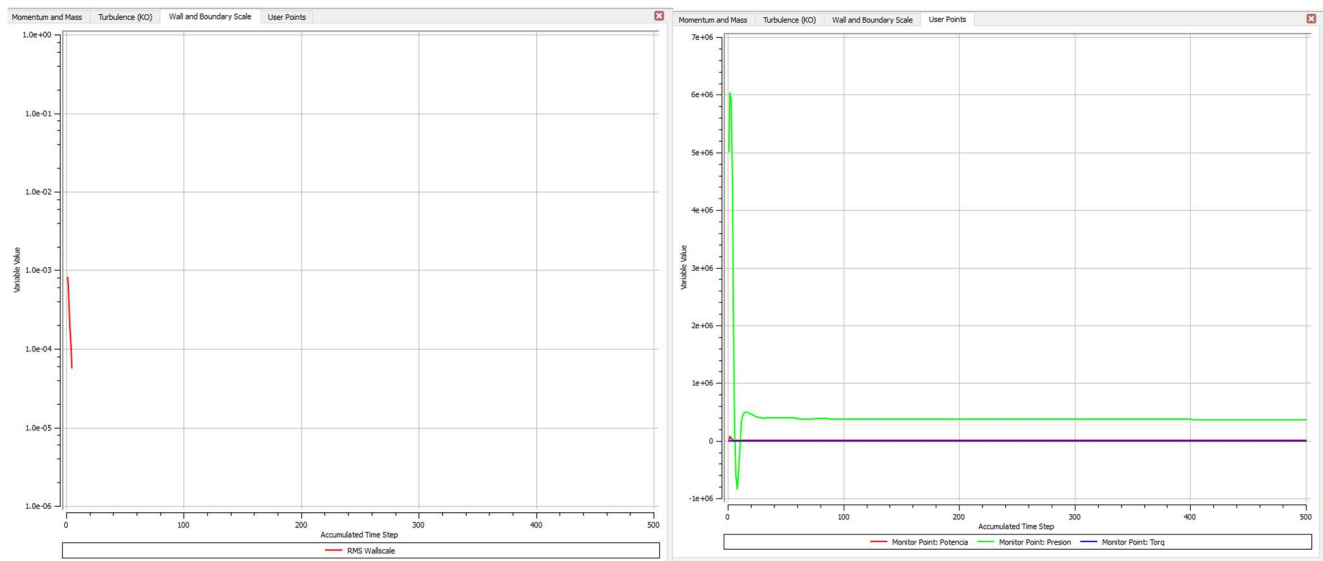

Figure 20: Residual convergence plots

Figure 21: Iterations vs value plots

Accuracy (error analysis)

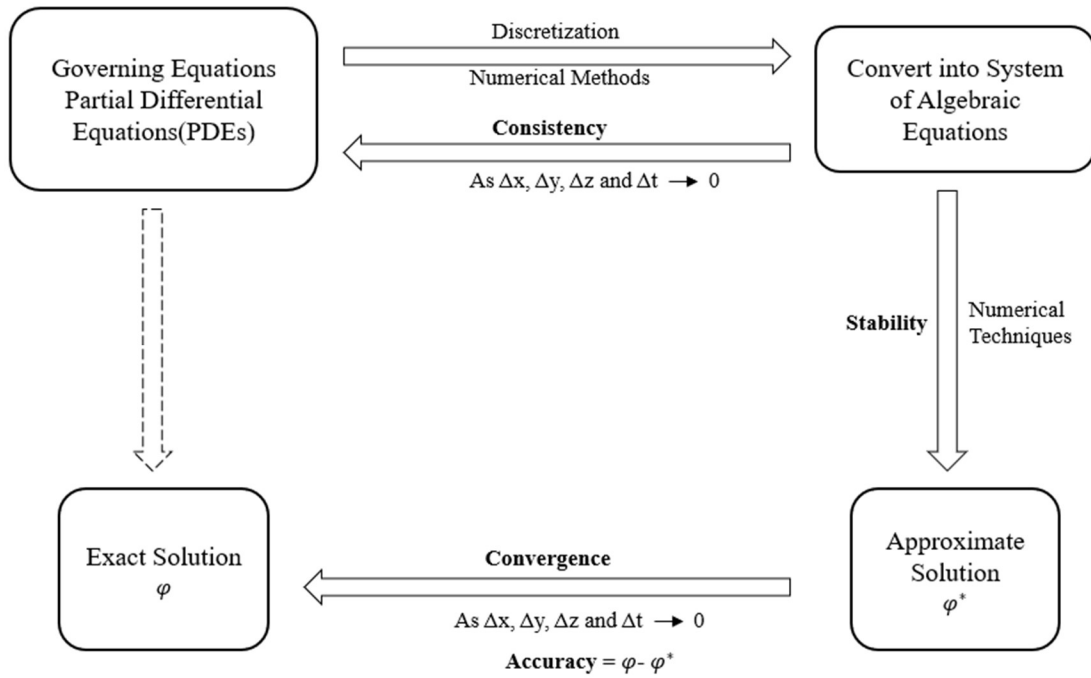Lorem.

Efficiency

Lorem.

Figure 22: Convergence flow diagram

Conclusion

This study had three main objectives. One, is to ensure that the CFD mesh generated is independent and that the solution is truly converged and accurate. Two, is to validate the CFD model against the provided academic data to ensure the model's accuracy and reliability. The third and final objective is to examine the effects of a variation in design mass flow and pressure on the fluid flow throughout the turbine. This is critical to maintain a high efficiency and avoid cavitation. We were able to successfully generate a grid independent model and validate it against previous experimental and simulated results. Using this model we explored the different effects of mass flow and pressure. From our results we determined that by increasing the mass flow at the inlet we were able to generate a higher velocity inside the turbine, but at a price of non-uniformity in the spiral casing. When we decreased the mass flow inlet, we produced a smaller velocity and more uniformity in the spiral casing but at a price of adding more vorticies and spiraling inside the draft tube, where cavitation increased. The next part of our study was to determine the effects of pressure on the turbine flow. We changed the outlet pressure to simulate a change in different exit conditions and overall pressure head. When we increaed the outlet pressure, we observed that it creates more vortices and swirling inside the draft tube. Overall, the turbine geometry that we are given was optimized for the specific inlet mass flow of 1460 kg/s and outlet pressure of 6.8 kPa, given to us in the study. If we were to increase the mass flow slightly, then the turbine would operate efficiently but any large changes in the inlet or outlet conditions cause the flow in the turbine to become chaotic and reduce the overall turbine efficiency.

References

[1] O. G. D. G. T. Biraj Singh Thaps, "Flow measurement around guide vanes of Francis Turbine: A PIV approach," *Renewable Energy,* vol. 126, p. 177, 2018.

[2] P. Breeze, Hydropower, London: Academic Press, 2018.

[3] C. M. Cutler J. Cleveland, Handbook of energy, Volume II, Elsevier Inc., 2014.

[4] P. M. G. Terry Wright, Fluid Machinery Applicatin, Selection, and Design, Boca Raton: CRC Press, 2009.

[5] Z. A. F. A. E. O. E. A. K. C. S. A. Hasan Akin, "A CFC Aided Hydraulic Turbine Design Methodology Apploed to Francis Turbine," in *4th International Conference of Power engineering, Energy, and Electrical Drives*, Istanbul, Turkey, 2013.

[6] L. A. Teran, F. J. Larrahondo and S. A. Rodriguez, "Performance Improvement of a 500-kW Francis Turbine based on CFD," *Elsevier,* vol. 96, pp. 977-992, 2016.