

A REPORT ON PERFORMANCE OF RADIAL IMPELLER



SUBMITTED BY:
DHRUBA ARYAL
ROLL No.:072BME616

SUBMITTED TO:
HARI DURA
DEPARTMENT OF MECHANICAL ENGINEERING
PULCHOWK CAMPUS

MARCH 9, 2019

Contents

1	Introduction	1
1.1	Plug Fan	1
1.2	Historical Background	1
1.3	Applications	3
2	Methodology	4
2.1	1D Design	5
2.2	3D Design	7
2.3	CFD Simulation	8
2.3.1	Geometry	8
2.3.2	Mesh	11
2.3.3	Physics Setup	13
2.3.4	Solver	13
3	Results	14
3.1	Contour Plot	14
3.2	Result Verification	17
4	Conclusions and Remarks	17
5	References	18

1 Introduction

1.1 Plug Fan

Centrifugal compressors are a sub-class of dynamic axisymmetric work-absorbing turbo-machinery. They achieve a pressure rise by adding kinetic energy/velocity to a continuous flow of fluid through the rotor or impeller. This kinetic energy is then converted to an increase in potential energy/static pressure by slowing the flow through a diffuser. There are three main types of centrifugal compressors on the basis of design of the blades of the impeller namely radial centrifugal compressor, forward facing centrifugal compressor and backward facing centrifugal compressor.

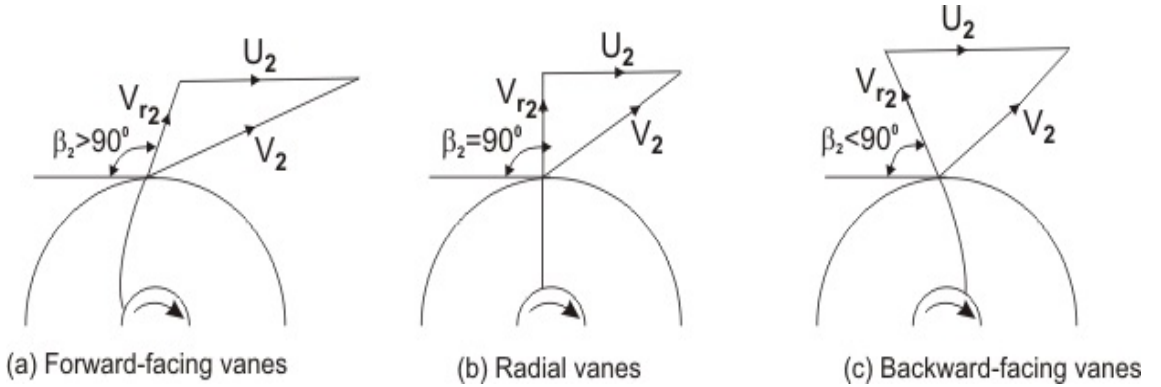


Figure 1: Velocity triangle of different blades

Practically, the bigger centrifugal compressors give maximum efficiency of about 78 percent while the smaller compressors give efficiency of about 74 percent which is mainly due to scale effect.

1.2 Historical Background

The first industrial application of centrifugal or radial compressors was in conjunction with early gas turbine work done around the turn of the 20th century. Some of the earliest work was done by Elling, who developed the first gas turbine to generate positive power in 1903. Such compressors have also been used by the process industry since the early 1900s. One of the earliest applications was as blast furnace blowers for steel mills. Improvement in centrifugal compressors have not been achieved through large discoveries. Rather, improvements have been achieved through understanding and applying incremental pieces of knowledge discovered by many individuals. A table illustrating achievements in this field is shown below.

Table 1: Historical Background

Before 1689	Early Turbo-machines	Pumps, blowers, fans
1689	Denis Papin	Origin of the centrifugal compressor
1754	Leonhard Euler	Euler's "Pump and Turbine" equation
1791	John Barber	First gas turbine patent
1899	Dr. A.C.E. Rateau	First practical centrifugal compressor
1927	Aurel Boleslav Stodola	Formalized "slip factor"
1928	Adolf Busemann	Derived "slip factor"
1937	Frank Whittle and Hans von Ohain, independently	First gas turbine using centrifugal compressor
After 1970	Modern turbo-machines	3D-CFD, rocket turbo-pumps, heart assist pumps, turbocharged fuel cells

In the early days of process centrifugal compressor development, design choices were restricted in large part by the manufacturing methods available at that time. Original Equipment Manufacturer's (OEM's) had to create designs that could be fabricated with the limited number of methods available. These included machining (i.e., 3-axis milling), joining (i.e., welding, riveting) and castings. Such methods are capable of creating fairly simple 2-D shapes. While these are adequate for a wide range of compressor applications, they proved to be inadequate for high flow and/or high Mach number machines. Castings were the method of choice for most OEMs because of the cost advantages attained when multiple copies of the same component were required and because the optimized performance was not a critical consideration at that time.

As the demand for higher performance increased, OEMs were forced to develop methods to manufacture 3-D blading. Early approaches included castings or 3-piece fabrication. In the former, the 3-D blade shapes could be created via casting of complex patterns, providing the leading edge angles necessary to achieve reasonable incidence angles. In the 3-piece fabrication, the "three pieces" were the shroud, the hub, blades. In the most primitive 3-D designs, the blade shapes were sections of cones, cylinders or toruses. These could be readily formed via rolling or stamping. However, these shapes, while providing improved incidence, did not provide adequate control over the area distribution through the impeller passage. For this, the blades were formed via die-pressing or other methods of forming with the blades compared against a check block to ensure the proper shape. However, the blades often deviated from the desired shape because of "spring-back". The blades were then welded or riveted to both the cover and disk to create impeller. More recently, OEMs have begun to single-piece machine covered impeller from a single forging, thus eliminating the need to weld or join cover to the blading. The individual impeller passages are machined by plunge milling from the inside and

outside diameter of the impeller. The inner and outer cuts are then tied together near the center of the individual flow passage.

The companies leading in design and manufacture of centrifugal compressors are Venco, AirPro, Greenheck, Plymovent, NEV, Fandis, Kemper, Nestro, Fumex, Chau Fan, Howden, Corac, Geovent, Ecofit and ETRI, Trotec, Atlas Copco, etc.

1.3 Applications

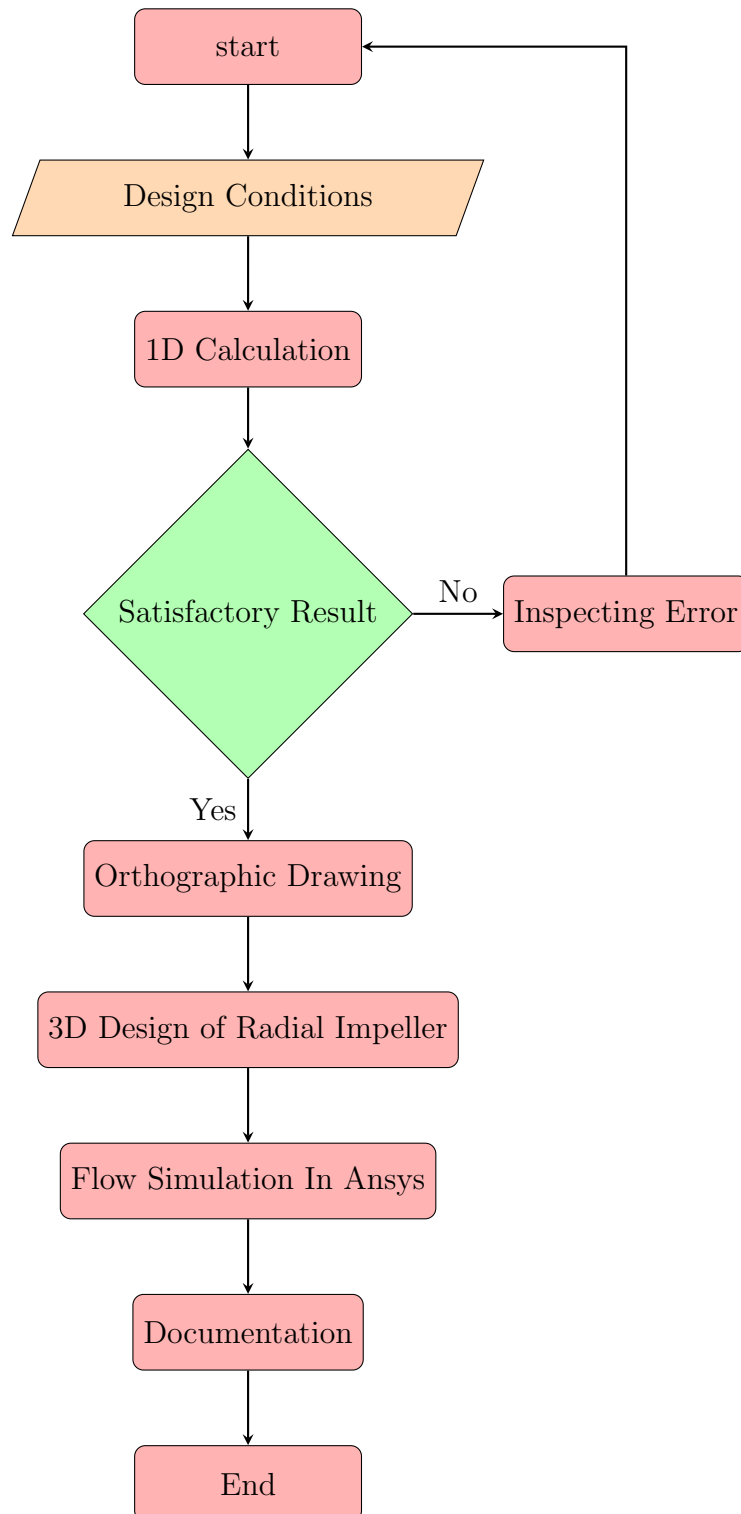
The centrifugal compressors are used in various fields of engineering applications. They are used in gas turbines of small aircraft engines and auxiliary power units to provide compressed gas. They are also used in automotive engine and diesel engine turbochargers and superchargers. These compressors are also used in pipeline compressors of natural gas to move the gas from the production site to the consumer. Likewise, they are also used in air-conditioning and refrigeration. They are also used to supply compressed air for all types of pneumatic tools in industry and manufacturing.

There are various advantages of using centrifugal compressors. These compressors are low weight, easy to design and manufacture. Similarly, they are suitable for continuous compressed air supply. Likewise, they are oil free in nature, have fewer rubbing parts, are relatively energy efficient, require low maintenance and are more reliable. They generate a higher pressure ratio per stage as compared to axial compressors. They also do not require special foundation.

The financial value of the compressors differ according to the type of compressors and their specifications. The price ranges from Rs.10,000 to even 2-3 lakhs depending upon the compressors capability, design, material used in manufacturing and other accessories provided with it.

2 Methodology

The flow chart showing the design process from 1D to 3D simulation is show below.



As seen from the flowchart, the initial design was an iterative process. After getting satisfactory result from the 1D calculation, the orthographic drawing and 3D

design were performed. After the design of the radial impeller, various geometries of fluid domain were made according to the impeller's design. These geometries were then set up and meshed in Ansys. After setting up the solver control, the whole system was simulated in CFX.

The initial design conditions for the design of the radial impeller are tabulated below.

Table 2: Design Parameters

Student Roll No: 616	Blade Thickness= 0.003m
Radius of Eye=0.316m	$\beta_2 = 90$
Radius of Outlet= 0.532m	No of Blades= 8
Thickness of Channel= 0.1m	Hub and Shroud Thickness= 0.03m

2.1 1D Design

The Euler pump and turbine equations are the most fundamental equations in the field of turbo-machinery. These equations govern the power, efficiencies and other factors that contribute to the design of turbo-machines. With the help of these equations the head developed by a pump and the head utilized by a turbine can be easily determined. As the name suggests these equations were formulated by Leonhard Euler in the eighteenth century. These equations can be derived from the moment of momentum equation when applied for a pump or a turbine. The Euler's equation is given as:

$$\Delta h = \Delta(u \cdot C_\theta) = u_1 C_{\theta 1} - u_2 C_{\theta 2}$$

Here, Δh = total change in enthalpy

u = blade speed

C_θ = component of absolute velocity in θ direction

For turbine, the value of h is positive while for pump its value is negative.

The graphs showing Power vs Flow rate and delta Pressure vs Flow rate as calculated from Excel are shown below.

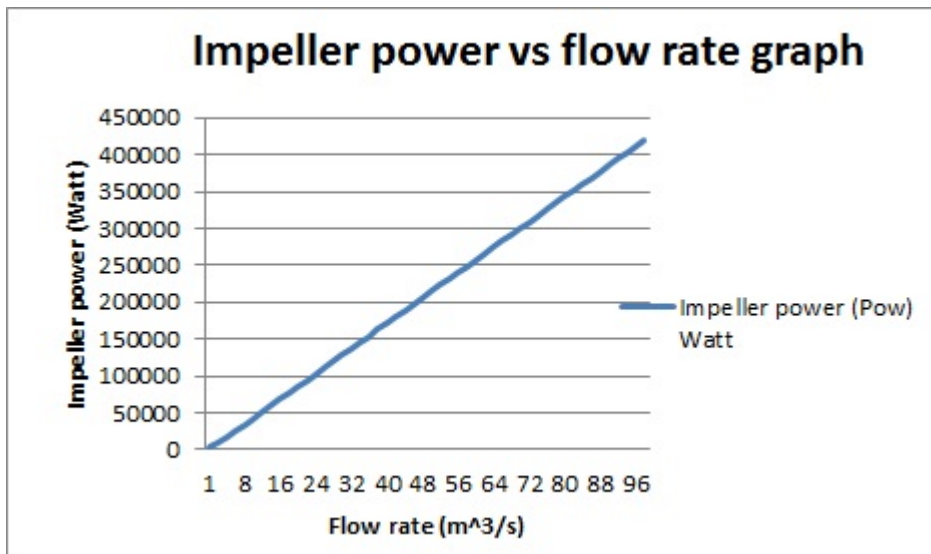


Figure 2: Impeller power vs Flow rate graph

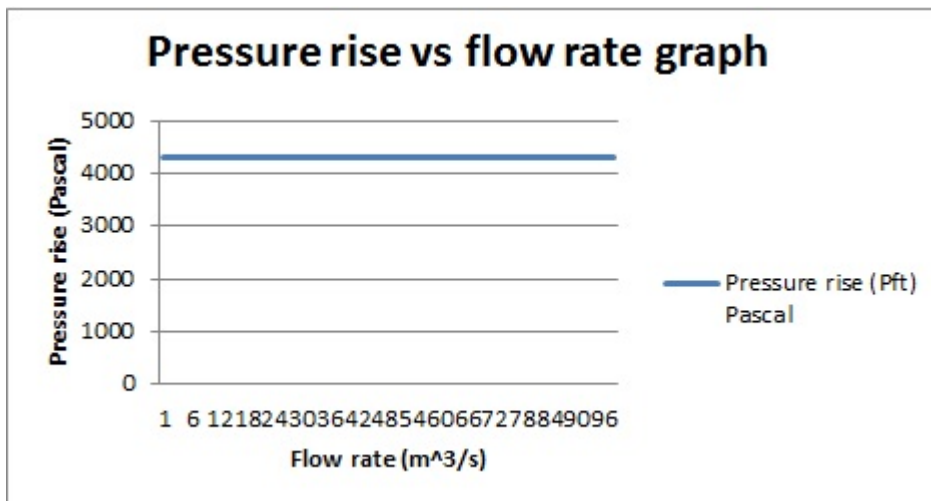


Figure 3: Pressure Rise vs Flow rate graph

2.2 3D Design

The figure showing the orthographic view and isometric view of the 3D-Design of radial impeller is shown below.

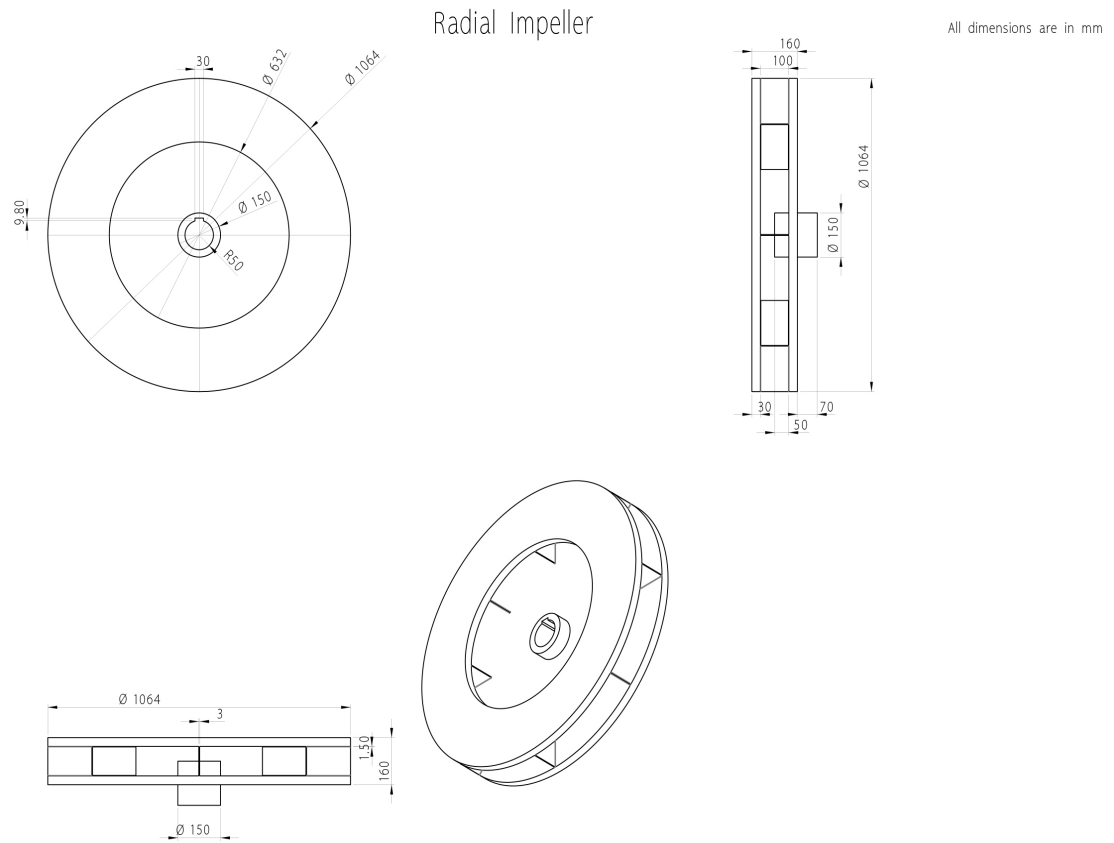


Figure 4: Orthographic and Isometric drawing of radial impeller

2.3 CFD Simulation

2.3.1 Geometry

The three separate fluid domain for representing inlet fluid, fluid inside the rotor and the outlet fluid were designed in Solidworks. The inlet and outlet fluid domains were simple to design without having to modify the domain to fit any solid structure like blade. But, while designing the rotor fluid domain, the volume occupied by the blade had to be removed as fluid did not occupy that space. Removing of that space occupied by the fluid in Solidworks would have been a difficult approach. So, rotor fluid domain including blade space and rotor tool were designed separately. The blade space was then removed in Ansys using Boolean feature.

The orthographic view of rotor tool and fluid domains are given below.

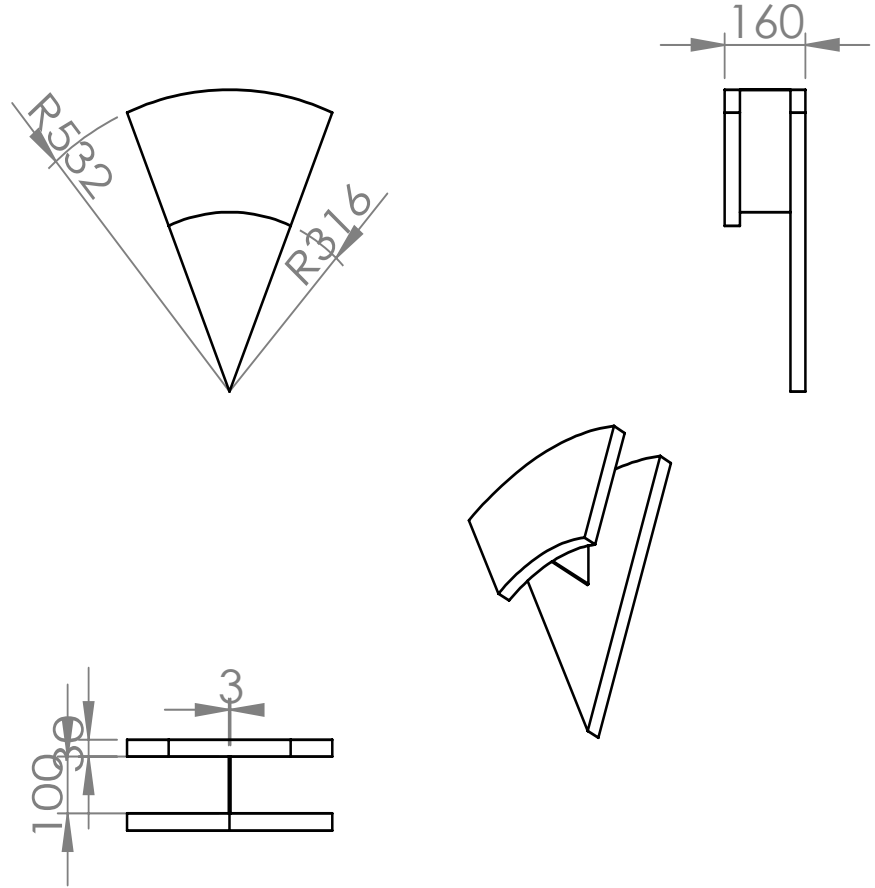


Figure 5: Rotor tool

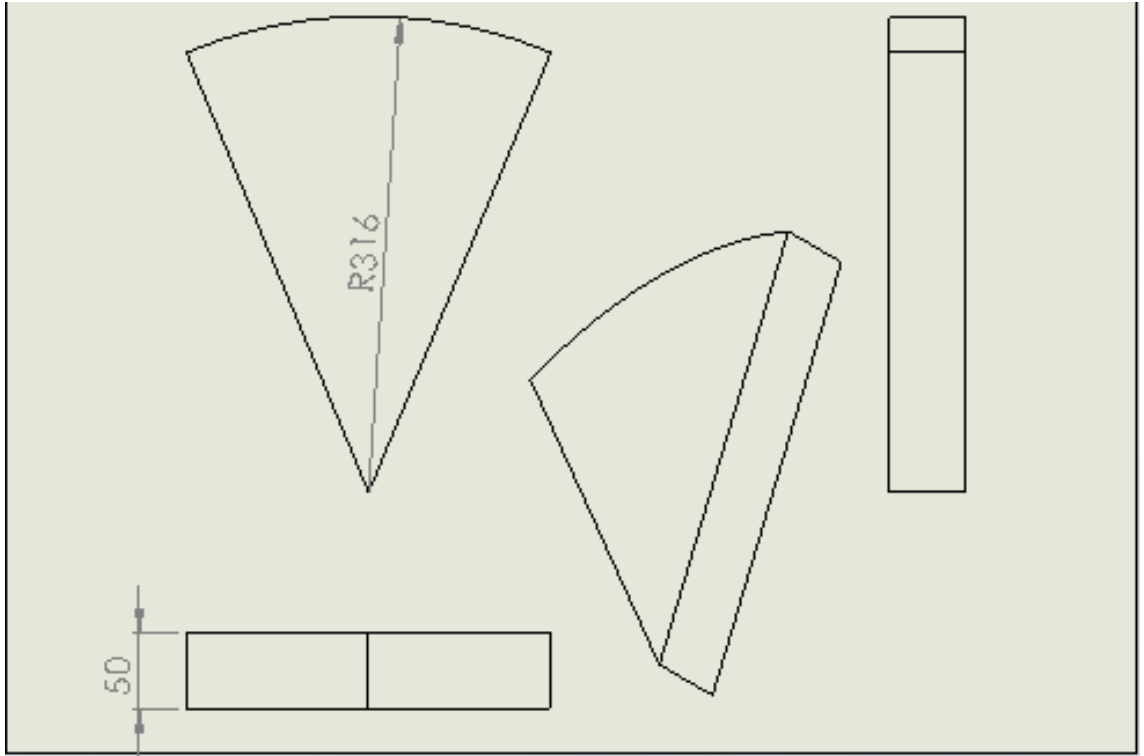


Figure 6: Inlet fluid domain

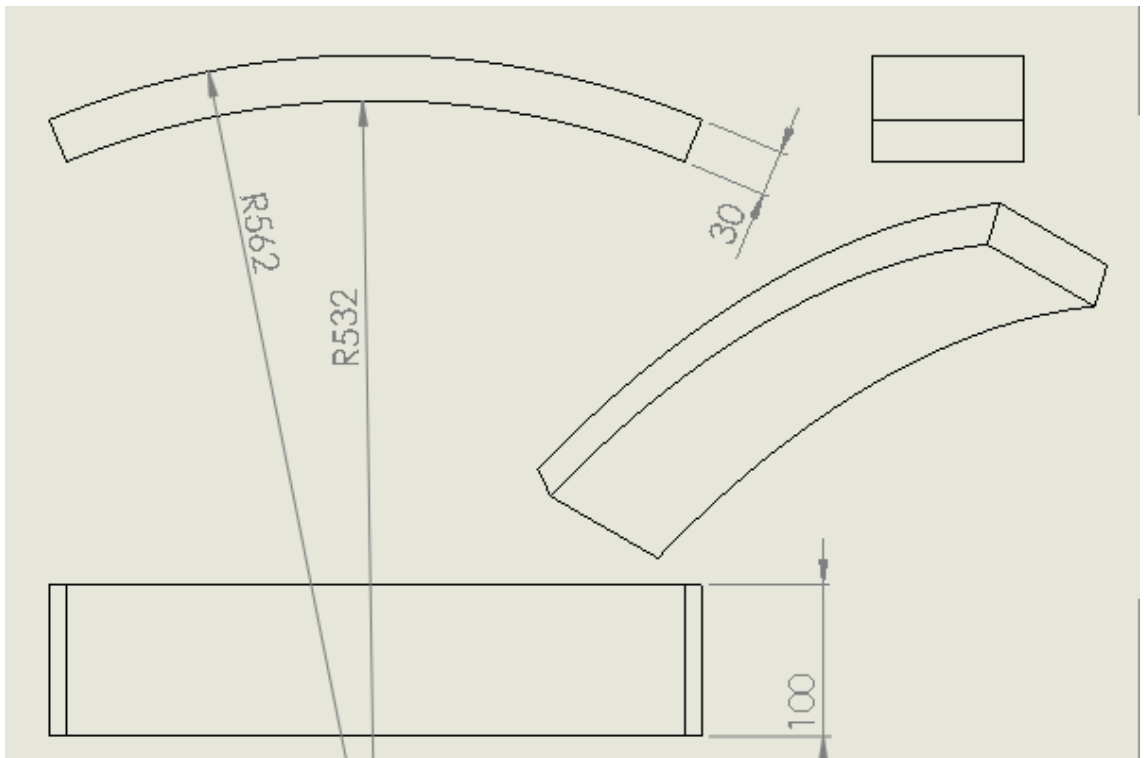


Figure 7: Outlet fluid domain

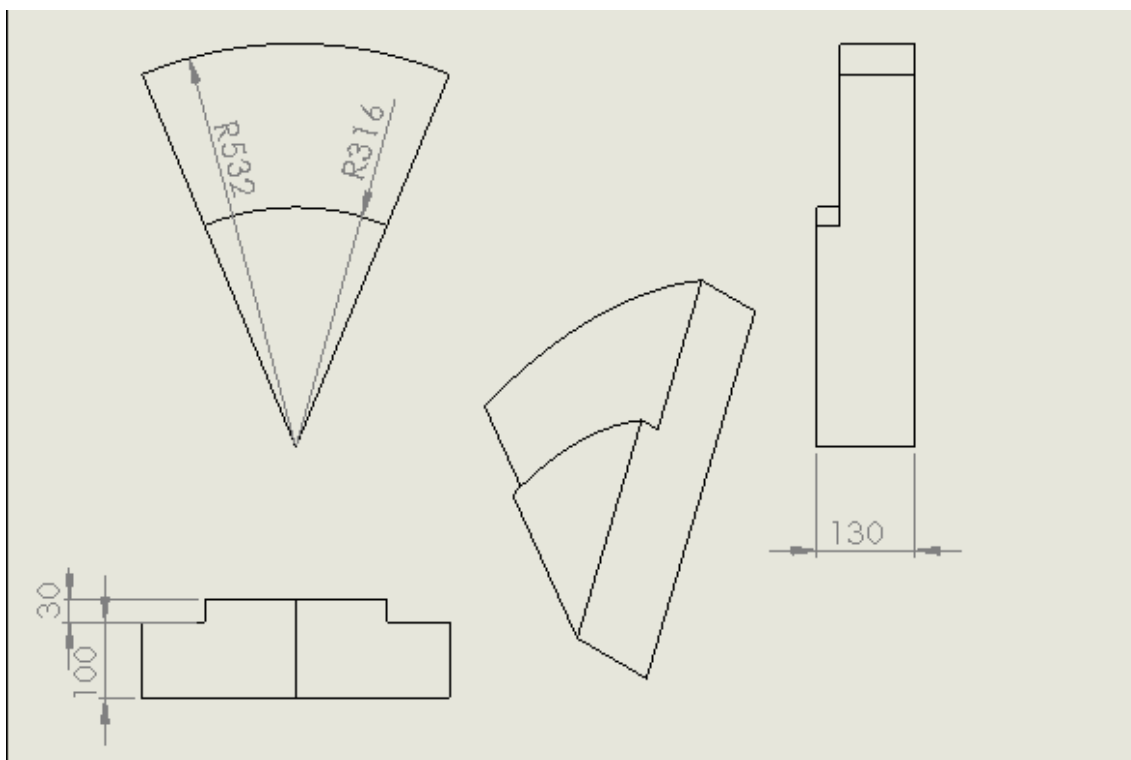


Figure 8: Rotor fluid domain

2.3.2 Mesh

The type of mesh used in all of the geometries i.e. the fluid domains was fine mesh. Since using coarse and medium mesh would have resulted in less convincing result, fine mesh was used. The poor performance capacity of the computer as well as non-converging of residual value discouraged any kind of mesh refinement.

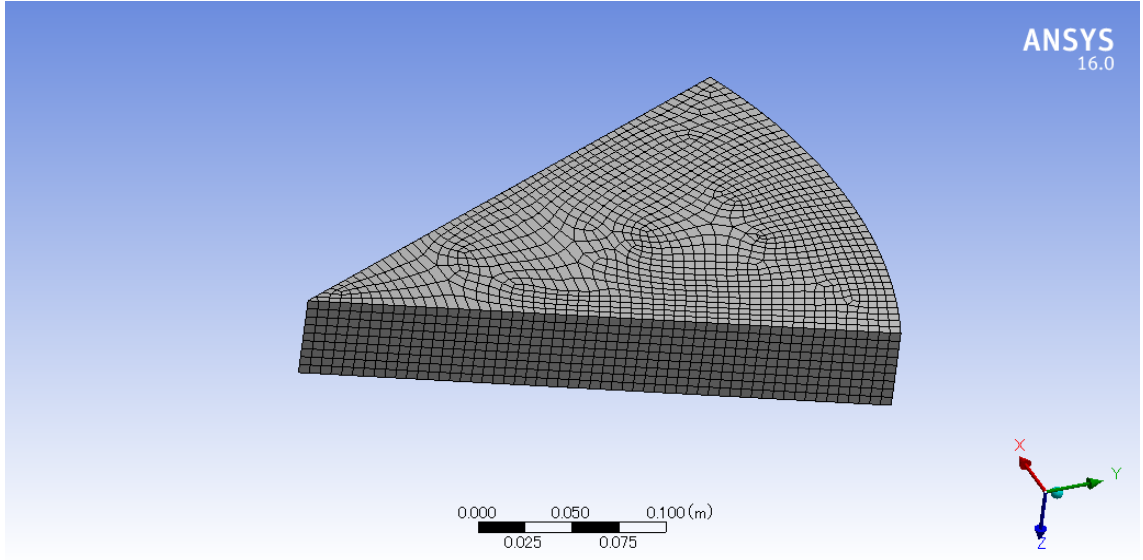


Figure 9: Inlet Fluid Domain's Mesh

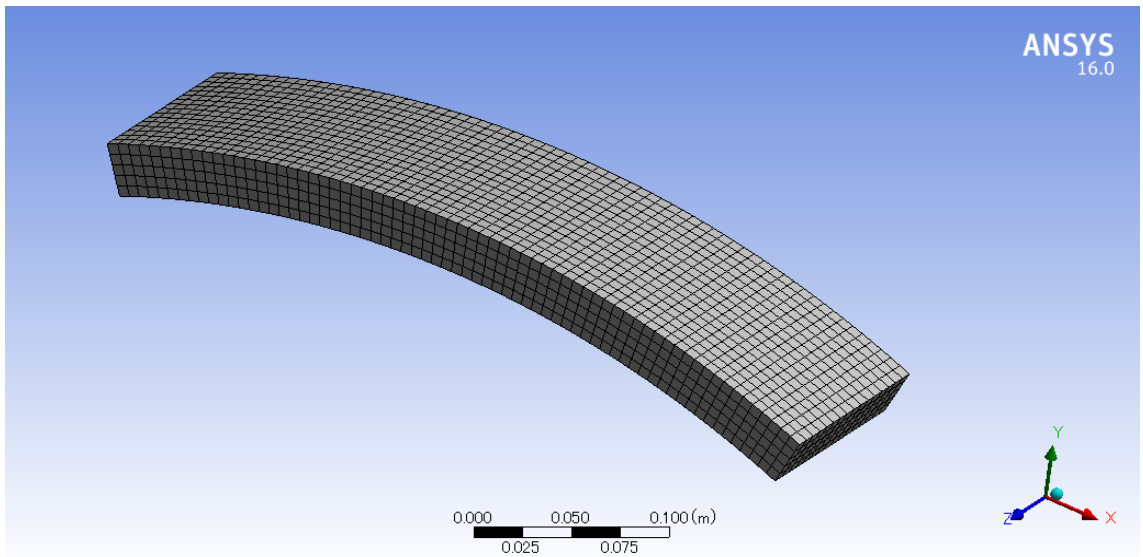


Figure 10: Outlet Fluid Domain's Mesh

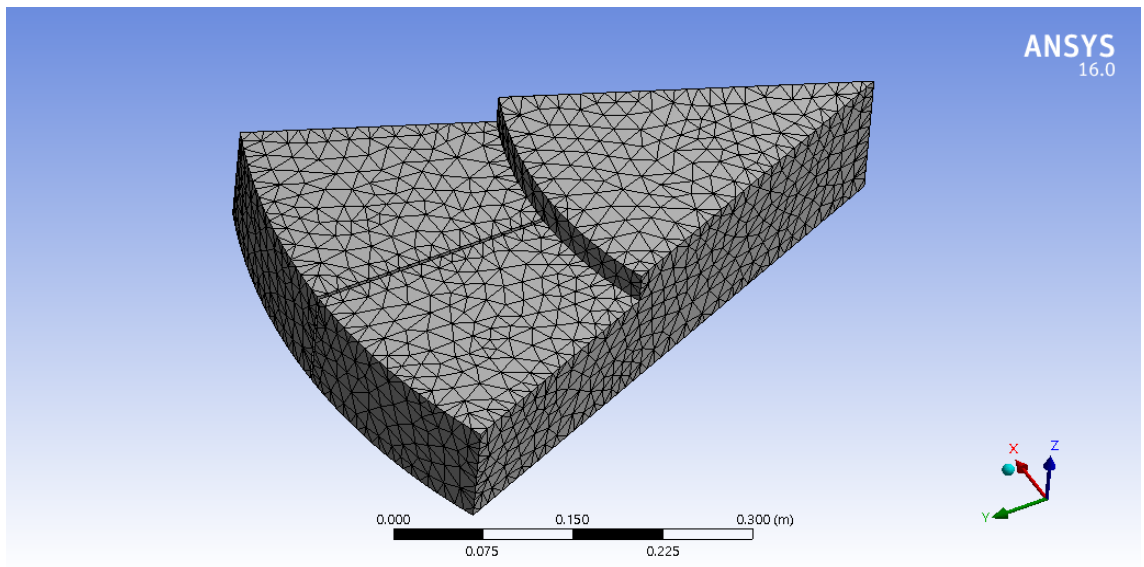


Figure 11: Rotor Fluid Domain's Mesh

2.3.3 Physics Setup

While setting up the whole mesh, various boundary conditions were assumed. The inlet of the fluid domain was made the inlet of the whole system with fluid flow rate of 2m/s and temperature of 300K. Similarly, the outlet of the outlet fluid domain was made the outlet of the whole system with assumption of 0 Pa relative pressure. Two interfaces were distinguished, one between inlet's outlet and rotor's inlet and the other between rotor's outlet and outlet's inlet. The interfaces made were Frozen Rotor type because the Frozen Rotor model treats the flow from one component to the next by changing the frame of reference while maintaining the relative position of the components. The other faces of the fluid domain were respectively categorized as wall and symmetry while creating boundaries.

As convergence criteria, the maximum number of iterations was limited to 10000 iterations and residual value of $10e-5$. The turbulence model was set up as high resolution.

2.3.4 Solver

Ansys CFX was used for the simulation. CFX is not a Finite Element Method, but is a Finite Volume Method based on Elements with a Cell Vertex Formulation. In CFX, the volume of control is assembled around the nodes. Each element is divided into sub-volumes. The flux through each face is based on the nodal values of element. As in finite element method a Stiff Matrix is assembled for each element, a Flux matrix is assembled for each one.

3 Results

3.1 Contour Plot

The velocity fields and pressure fields are shown in figures below.

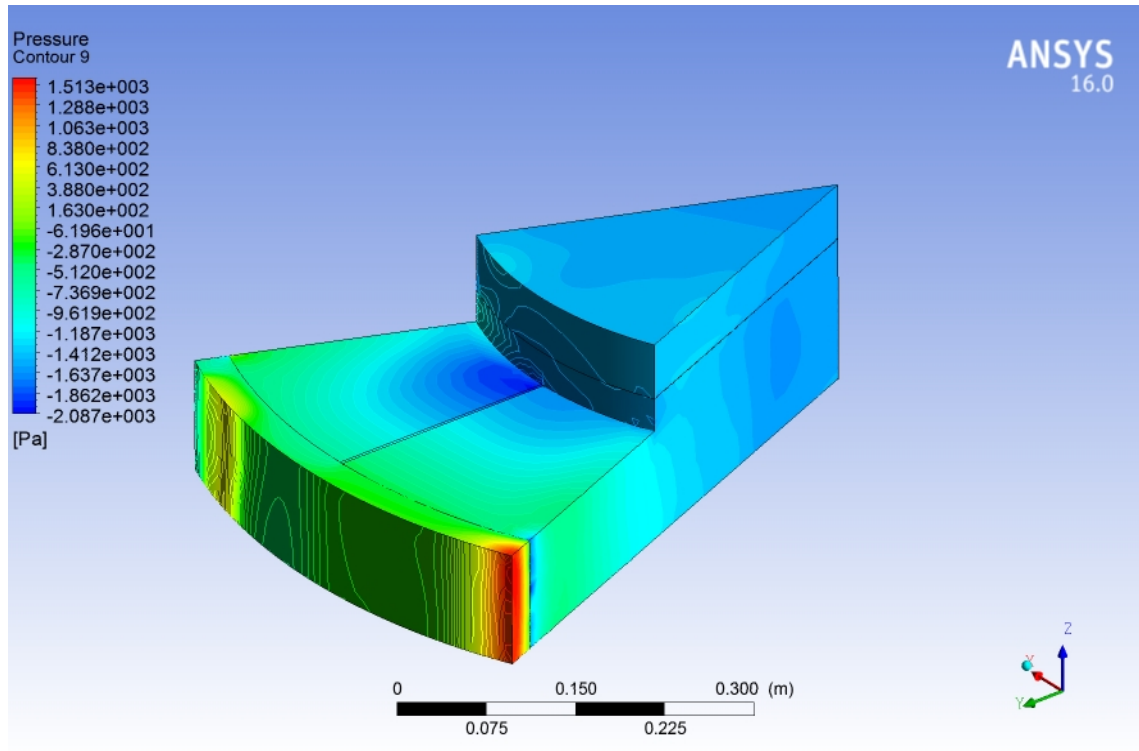


Figure 12: Pressure Contour for $u=2\text{m/s}$

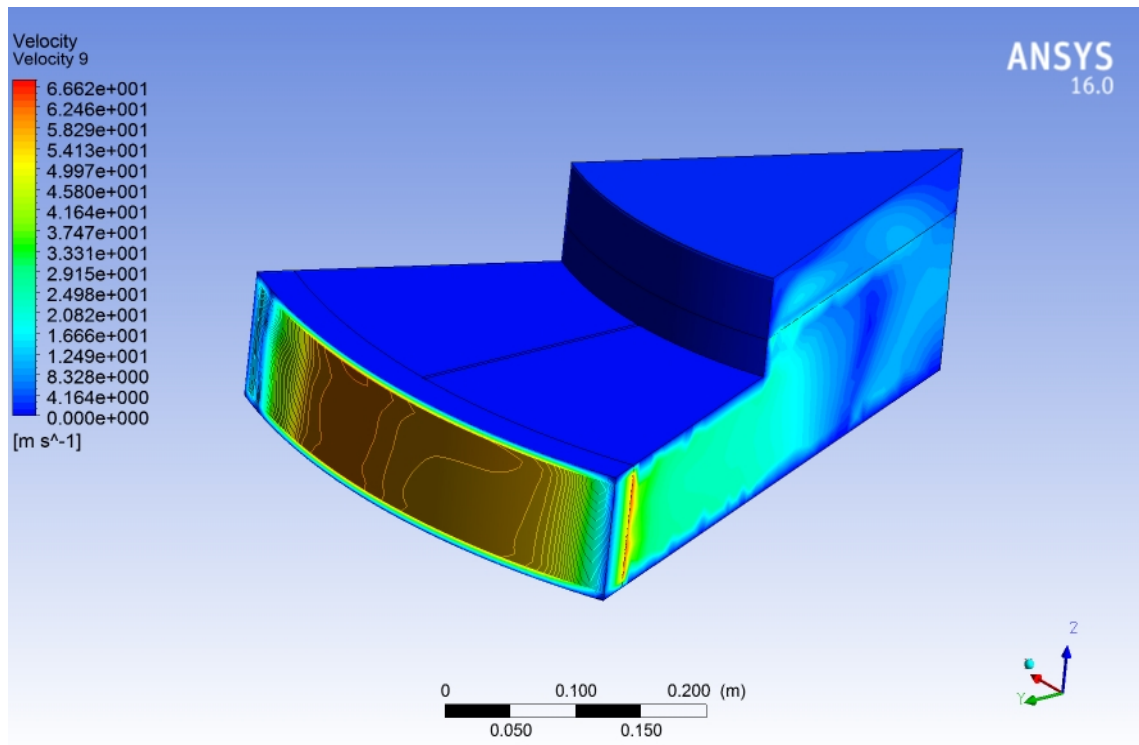


Figure 13: Velocity Contour for $u=2\text{m/s}$

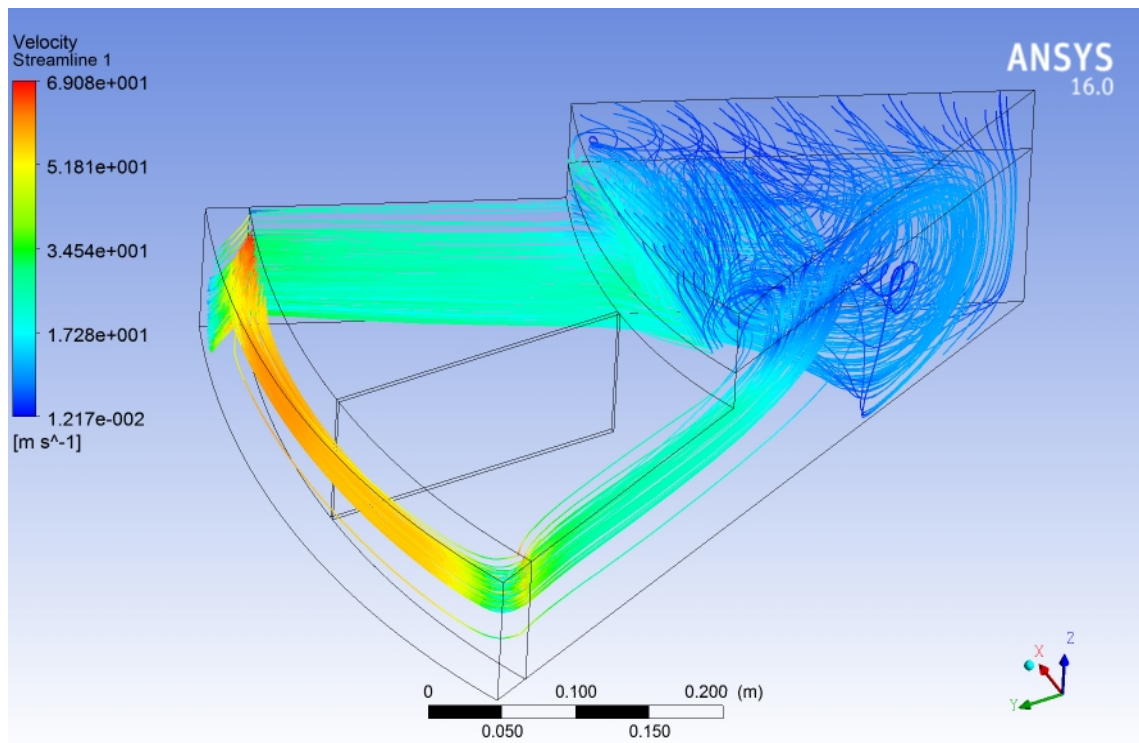


Figure 14: Streamline for $u=2\text{m/s}$

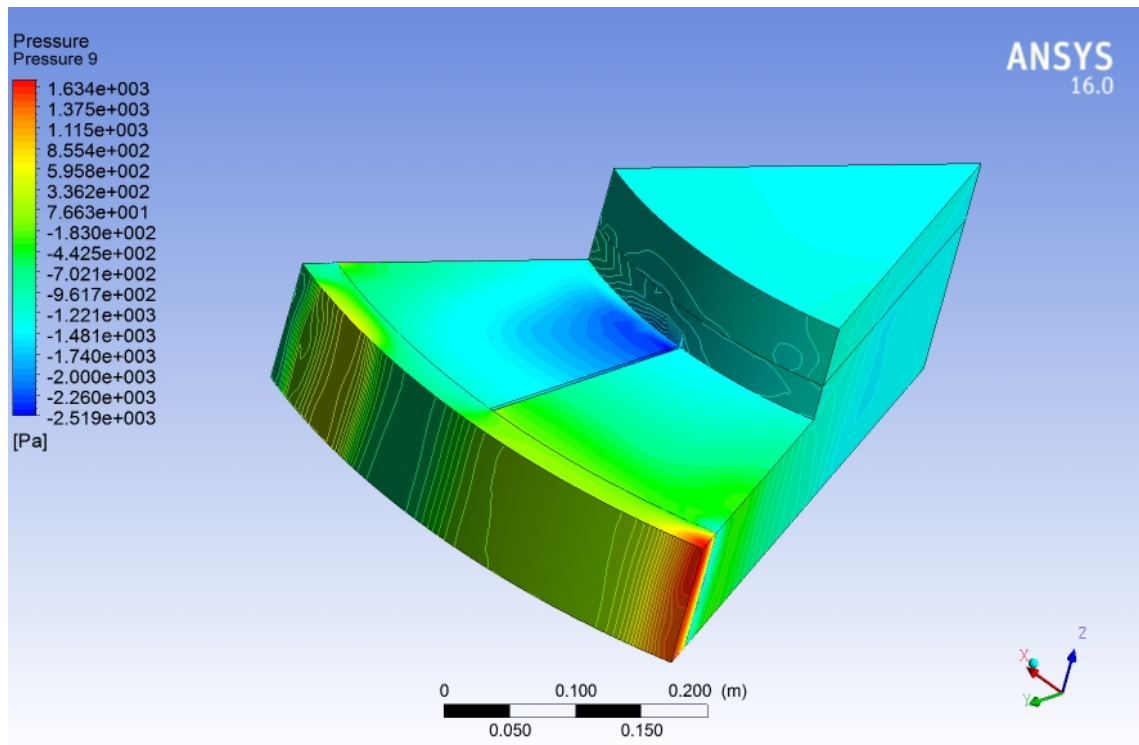


Figure 15: Pressure Contour for $u=4\text{m/s}$

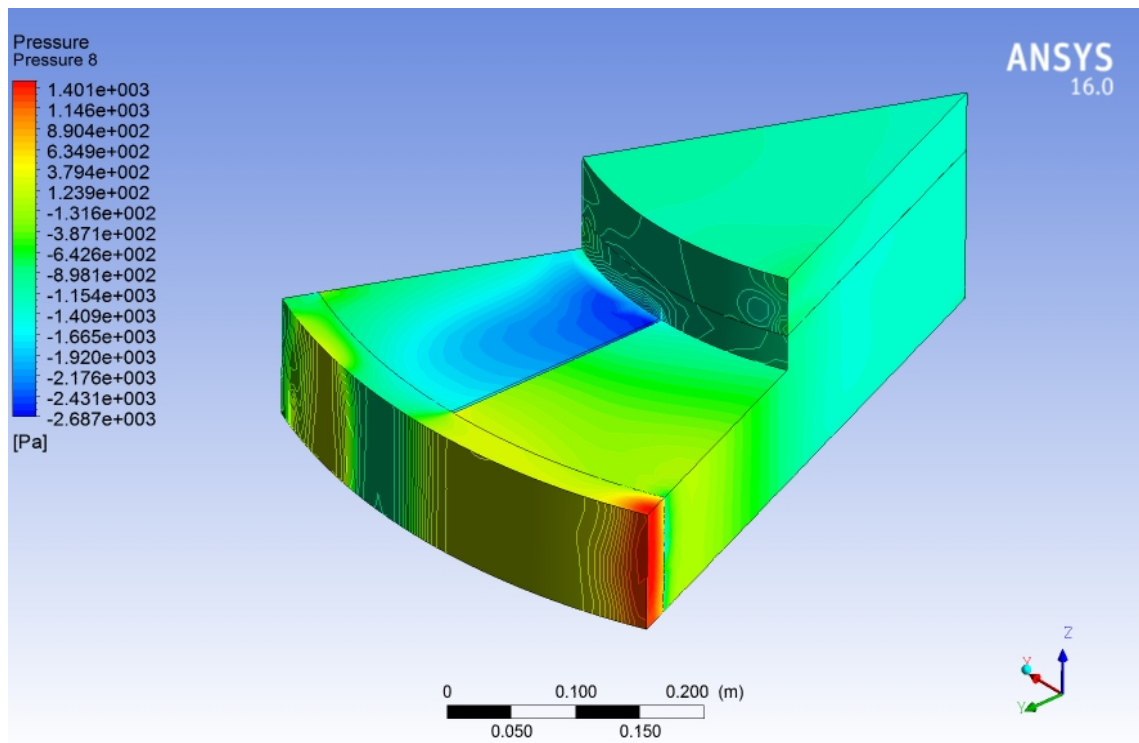


Figure 16: Pressure Contour for $u=6\text{m/s}$

3.2 Result Verification

The theoretical values obtained for pressure rise and power from 1D calculation as well as those obtained from the simulation are tabulated below for three different flow rates.

Table 3: Theoretical values

Axial velocity(m/s)	Flow rate(m^3/s)	Pressure rise(Pascal)	Power(Watt)
2	0.627	4286	2689
4	1.255	4286	5378
6	1.882	4286	8067

Table 4: Values from simulation

Axial velocity(m/s)	Flow rate(m^3/s)	Pressure rise(Pascal)	Power(Watt)
2	0.627	3675	4214
4	1.255	4240	6667
6	1.882	4173	8955

From the tables above, it can be seen that the pressure rise obtained is lower than the theoretical values and the power consumption is higher than that required theoretically. This may have been due to the various reasons like friction loss, formation of eddies due to turbulence and other losses.

4 Conclusions and Remarks

From the 1D calculation and CFD simulation results, the percentage errors in pressure rise at axial flow velocity of 2m/s, 4m/s and 6m/s are calculated to be 14.26 percent, 1.07 percent and 2.64 percent respectively. From these 3 CFD results, the average percentage error is 5.99 percent.

Hence, three-dimensional computational fluid dynamics (CFD) codes are the most rigorous analytical techniques that can be used to calculate the flow through various turbo-machines. CFD analyses can account for all the facets of the aerodynamic component geometry and provide a far more comprehensive approximation of the flow physics than any of the analytical methods. As a result, using such analyses can lead to superior aerodynamic designs, and therefore superior performance.

5 References

The references used in this report are given bellow.

- Sorokes,J.M.,Kuzdzal,M.J.(2010).Centrifugal compressor evolution.Retrieved from <https://pdfs.semanticscholar.org/1b02/28f898c93ce7176b9b197b2cf8d02da91b4c.pdf>.
- Gorla,R.S.R.,Khan,A.A.(2003).Turbomachinery design and theory.New York: Marcel dekker.