Design of High-Speed Centrifugal Compressor
for High-Speed Turbomachinery

A Final Year Project Report

Presented to

SCHOOL OF MECHANICAL & MANUFACTURING ENGINEERING

Department of Mechanical Engineering

NUST

ISLAMABAD, PAKISTAN

In Partial Fulfillment
of the Requirements for the Degree of
Bachelor of Mechanical Engineering

By

Muhammad Adil Mumtaz

Muhammad Usman Shafique

Muhammad Usman Khan

Mohammad Mateen Ashraf

May-2022

EXAMINATION COMMITTEE

We hereby recommend that the final year project report prepared under our supervision by:

Muhammad Adil Mumtaz 00000255942

Muhammad Usman Shafique 000002265226

Muhammad Usman Khan 00000243975

Mohammad Mateen Ashraf 00000257975

Titled: "**Design of High-Speed Centrifugal Compressor for High-Speed Turbomachinery**" be accepted in partial fulfillment of the requirements for the award of Mechanical Engineering degree with grade ___

Supervisor: Mr. Abdul Naeem Khan (Assistant Professor)	______________________ Dated:
Committee Member: Name, Title (faculty rank) Affiliation	______________________ Dated:
Committee Member: Name, Title (faculty rank) Affiliation	______________________ Dated:

______________________ ___________

(Head of Department) (Date)

COUNTERSIGNED

Dated: ________________ __________________________

(Dean / Principal

ABSTRACT

One of the most valuable type of compressors is centrifugal compressor. This project aims at the design of centrifugal compressor for a high-speed turbomachinery to meet certain objectives and requirements. Apart from the turbomachinery, centrifugal compressor finds its application in many industries which include aerospace system, refrigeration systems, turbochargers, and automobile. This work includes the design, flow analysis and structural analysis of the centrifugal compressor.

Many pioneers of turbomachinery have worked on the centrifugal compressor which include Baljie, Robinson, Schultz. The design of centrifugal compressor remains one of the complex objectives in turbomachinery due to the variability of the parameters whose relations are mostly based on the assumption values. Similarly, there are many approaches to design a centrifugal compressor.

Our approach includes the design of compressor through preliminary calculations of parameters given the required output parameters. The preliminary calculations are used to create CAD model in software such as CFTURBO. The model is verified through computational fluid dynamics (CFD) in ANSYS with the help of CFX, Bladegen, and Turbo Grid. After gaining satisfactory results, the compressor was tested for structural integrity by performing Finite Element Analysis again using the software ANSYS. After conformance of the results to the required parameters, the model is 3D printed and a fully working prototype is made. Our verified results meet the 250 m/s velocity at the volute exit with structural integrity enough to sustain 20000 rpm in the form of structural steel.

A centrifugal compressor design is a complex one, yet our design approach has made it to be an uncomplicated procedure. The concerned application industry will handle the centrifugal compressor according to its use from us.

ACKNOWLEDGMENTS

At this stage of our project, we are extremely grateful for our Allah Almighty which has enabled us to achieve this progress in our project. The project inculcated, in us, a sense of commitment, responsibility, and taught us strong project management and analytical skills which will help us in our professional life.

As an acknowledgement, we are deeply thankful to our kind supervisor Assistant Professor Abdul Naeem Khan, whose graceful support at every stage has pushed use to achieve better, think better and taught us to fight for our right during the course of this project.

We want to acknowledge that this project has been progressed so far with the help of our esteemed supervisor and our faculty members which provided us support by teaching us valuable lessons during our study. We hope this fruitful project will enable us to apply our skills and knowledge to polish ourselves as good mechanical engineer.

ORIGINALITY REPORT

ORIGINALITY REPORT

14%	12%	4%	5%
SIMILARITY INDEX	INTERNET SOURCES	PUBLICATIONS	STUDENT PAPERS

PRIMARY SOURCES

1	dokumen.pub Internet Source	4%
2	www.coursehero.com Internet Source	2%
3	tudr.thapar.edu:8080 Internet Source	1%
4	epdf.pub Internet Source	1%
5	Submitted to Engineers Australia Student Paper	1%
6	Submitted to Higher Education Commission Pakistan Student Paper	<1%
7	thepipingtalk.com Internet Source	<1%
8	Submitted to Kuwait University Student Paper	<1%

9	asmedigitalcollection.asme.org Internet Source	<1 %
10	V V Neverov, Y V Kozhukhov, A M Yablokov, A A Lebedev. "Optimization of a centrifugal compressor impeller using CFD: the choice of simulation model parameters", IOP Conference Series: Materials Science and Engineering, 2017 Publication	<1 %
11	www.thomasnet.com Internet Source	<1 %
12	www.mdpi.com Internet Source	<1 %
13	www.hindawi.com Internet Source	<1 %
14	ebin.pub Internet Source	<1 %
15	Khalil Ekradi, Ali Madadi. "Performance improvement of a transonic centrifugal compressor impeller with splitter blade by three-dimensional optimization", Energy, 2020 Publication	<1 %

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

ABBREVIATIONS

CFD $\quad$ $Computation\ Fluid\ Dynamics$

CCD $\quad$ $Centrifugal\ Compressor\ Design$

CFX $\quad$ $Congregationum\ Fratrorum\ Xaverianorum$

CFC $\quad$ $Centrifugal\ Compressor$

IGV $\quad$ $Inlet\ Guide\ Vanes$

NOMENCLATURE

u	$tangential\ velocity$
c	$absolute\ velocity$
w	$relative\ velocity$
c_x	$axial\ component\ of\ velocity$
c_θ	$swirl\ component\ of\ velocity$
D_tip	$Tip\ diameter$
$D1$	$inlet\ mean\ diameter$
$M1$	$inlet\ Mach\ number$
$P1$	$static\ pressure\ at\ inlet$
$T1$	$static\ temperature\ at\ inlet$
$U1$	$Inlet\ peripheral\ speed$
$M2$	$Outlet\ Mach\ number$
$U2$	$Outlet\ peripheral\ speed$
$P2$	$Static\ pressure\ at\ outlet$

$T2$	*static temperature at outlet*
$D2$	*outlet diameter*
L	*mean length*
$b2$	*blade height at outlet Outlet*
N	*number of blades*
$D3$	*Vaneless space diameter*
$P3$	*static pressure at vanless space*
$T3$	*static temperature at vaneless space*
$M3$	*vanless space Mach number*
$b3$	*blade height at vaneless space*

CHAPTER 1: INTRODUCTION

Compressor:

It is dynamically serving machine which rises fluid pressure by decreasing the volume of fluid is called a compressor (Waqar, 2020). Outwardly operated compressors that need a sealant device where the shaft outlets the crankcase are called the open compressors (Hundy, Welch, 2016). . Compressors are same to pumps, but fluids in working are different. In the compressor we use the gas as air is the famous squeezed wind and other commercial squeezed gases like oxygen, natural gas, and nitrogen. In pump we use the liquid which both rises the force on a liquid and transports through a tube.; (Perry, Green.2007)

Basic Principle of Compressor:

The basic working principle of a gas or air compressor is easy to understand. Actually, it works by changing the volume of the gas or air. It uses a piston or diffuser to increase the pressure of the working fluid. When the working fluid enters a diffuser, it converts the fluid's velocity into pressure energy.

Working of Compressor:

The main working key of a gas or air compressor is very easy to understand. Actually, compressor works by lowering the volume of the gas or air and enhance the pressure. It has a diffuser (divergent channel) which enhances the force of the working fluid. As the gas enters in diffuser, it converts the velocity of gas into force energy

Types of Air Compressor:

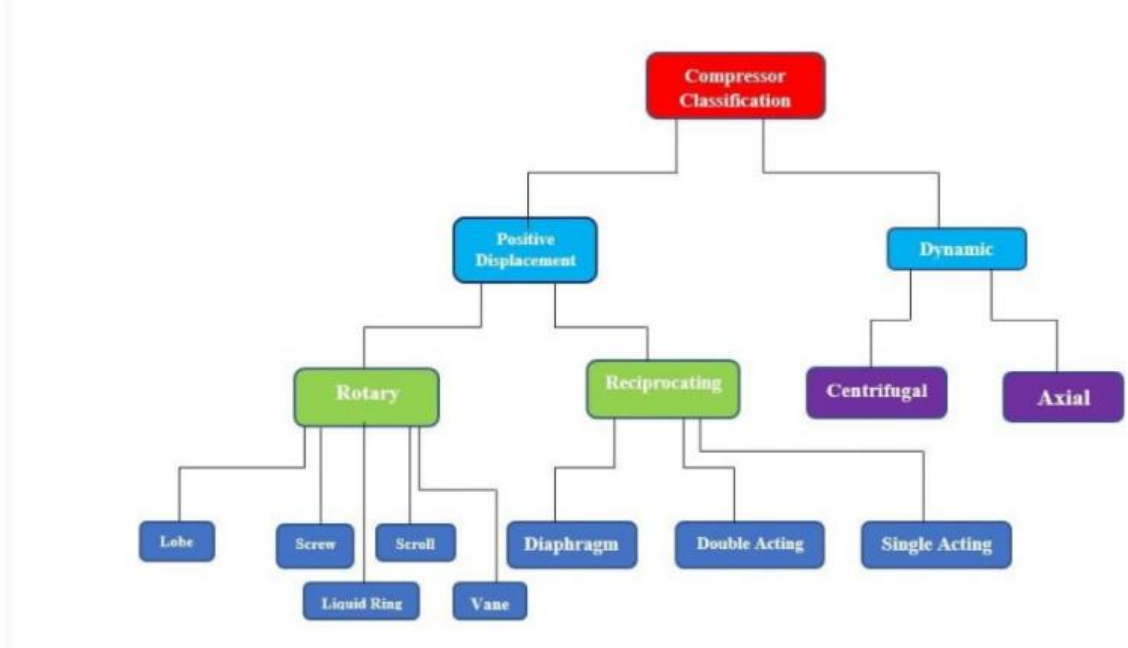
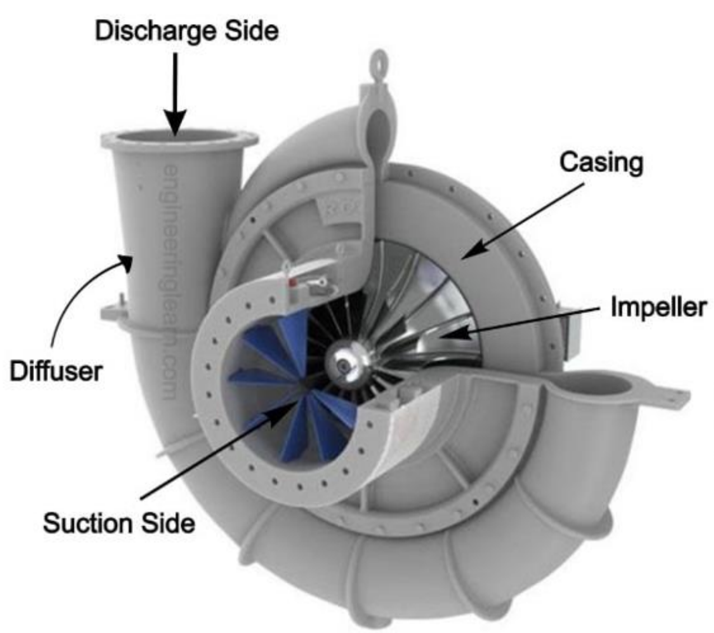


Figure 1: Types of Compressors

Centrifugal Compressor:

Centrifugal compressors use a rotating disc or impeller in a moulded casing to push the gas to the rim of the impeller and increase the gas's velocity. A diffuser section (divergent channel) changes the velocity energy into pressure energy. Centrifugal compressors raise the kinetic energy of the gas with a high-speed impeller, and then change this energy to increased pressure in a diverging outlet passage called a diffuser (Britanica, 2019).



The centrifugal compressor is one of the most popular types of air compressors. It uses a rotating impeller or disc in the casing to push the gas or air towards the impeller blades. The impeller blades increase the speed of the gas. The diffuser converts the gas velocity energy into pressure. After that, the gas is directed to the desired location.

Centrifugal compressors have lower compression ratios than positive displacement compressors, but they handle large volumes of gas. Many centrifugal compressors use multiple stages to improve the compression ratio. In these multi-stage compressors, the gas typically flows through intercoolers between stages.

These compressors are primarily used for constant, stationary applications in industries such as petrochemical plants, natural gas, chemical plants, and oil refinery processing plants. Its applications range from 100 horsepower to thousands of horsepower. High outlet pressures more than 6.9 MPa (1,000 psi) can be achieved with various upgrades. These types of air compressors are mostly used in air conditioning and HVAC systems.

Components of Centrifugal Compressor:

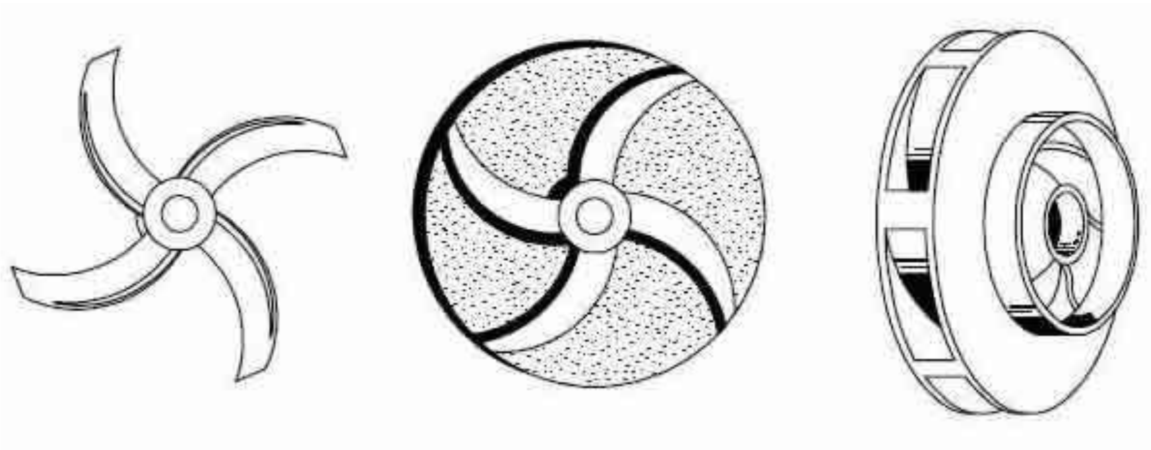
Impeller:

The impeller is the basic rotating detail of centrifugal compressor, which gives the speed to the air. The impeller via its blades transfers the shaft paintings to the fluid and will increase its strength level. it can be made in one piece together with both the inducer section and a largely radial element. The inducer receives the go with the flow between the hub and tip diameters (dh, d1) of the impeller eye and passes it directly to the radial portion of the impeller blades. The go with the flow coming near the impeller may be without or with swirl. The inducer phase can be regarded upon as an axial compressor rotor positioned upstream of the radial impeller. In some designs that is made one after the other and then installed at the shaft in conjunction with the radial impeller. In an excellent majority of centrifugal compressors, the impeller has instantly radial blades after the inducer segment. At high speeds, the impeller blades are subjected to high stresses which tend to straighten a curved impeller blade. consequently, the selection of radial instantly blades is sounder for higher peripheral speeds. The dimensions of impeller affect the pinnacle and glide traits of the compressor Impeller has three types:

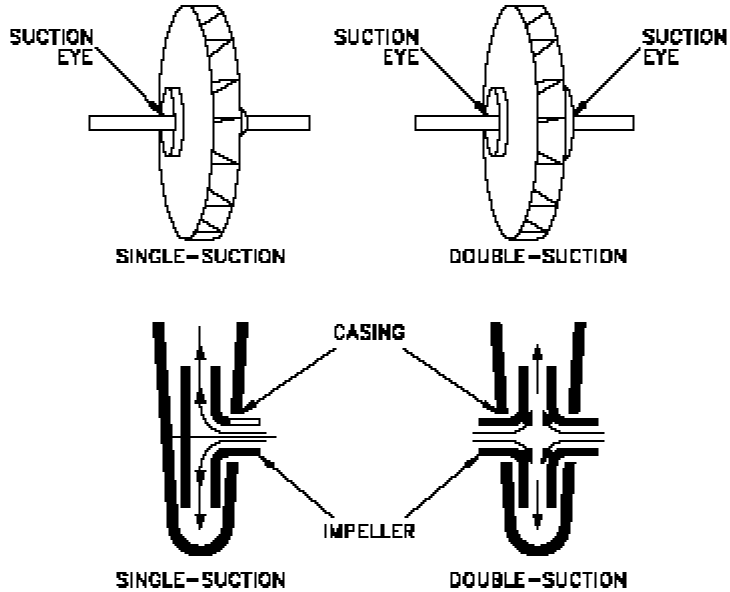
(a) Open impeller: It has no enclosing covers either in both sides. These can be used on single stage compressor for heavy flow rates. Open impellers are not commonly used in system industry apart from air compression and sometime as the first step in a multistage compressor.

(b) Semi enclosed impeller: It has only one side enclosed. The shroud (enclosing plate) adds mechanical power. Semi enclosed impellers have better efficiencies than open impellers. These can also be utilized for massive float rates in single stage compressor and as first step in a multistage compressor.

(c) Enclosed impeller: There are vanes positioned in between the two discs. Enclosed impellers are specially utilized in multistage compressors. The backward leaning, closed impeller which has a broad flow span is the maximum commonly used impeller.



The impeller can also have a single or double suction.



Diffuser:

The diffuser and the number of fixed channels where the air is lowered by continuous pushing at constant pressure. The static pressure of the fluid rises further on account of the deceleration of the flow. The absolute velocity of the gas at the impeller exit is high which is reduced to a lower velocity in the diffuser The amount of deceleration and the static pressure rise in the diffuser depend on the degree of reaction and the efficiency of the diffusion process. An efficient diffuser must have minimum losses maximum efficiency and maximum recovery coefficient. The diffuser may be merely a vaneless space or may consist of a blade ring. For high performance, the design of the diffuser is as important as that of the impeller.

Vanless Diffuser:

Because the name suggests the gas in a vaneless diffuser is subtle within the vaneless space around the impeller before it leaves the level via a volute casing. In a few applications the volute casing is overlooked. The gas within the vaneless diffuser gains static strain upward push truly because of the diffusion method from a smaller diameter d2 to a larger diameter d3. Such a go with the flow in the vane1ess space is a free-vortex flow wherein the angular momentum stays constant. The diffusion is proportional to the diameter ratio d3/d2. This leads to a rather large sized diffuser that is a severe downside of the vaneless type. In a few instances, the overall diameter of the compressor may be impractically big this is a critical quandary which prohibits the use of vaneless diffusers in aeronautical programs. Besides this the vaneless diffuser has a decrease efficiency and can be used most effective for a small pressure rise. However, for commercial programs in which big-sized compressors are acceptable the vane less diffuser is within your budget and provides a wider range of operation besides this it does no longer suffer from blade stalling and surprise waves.

Vaned Diffuser:

For a higher compression ratio throughout the radial diffuser, the diffusion technique has to be done across a distinctly shorter radial distance. This calls for the application of vanes which offer extra steerage to the go with the flow in the diffusing passages. Diffuser blade rings may be fabricated from sheet steel or cast in cambered and uncambered shapes of uniform thickness. To keep away from separation of flow the divergence of the diffuser blade passages in the vaned diffuser ring can be kept small by using using a large quantity of vanes. But this could result in better friction losses. For this reason, a maximum range of diffuser vanes have to be hired. The divergence of the float passages must now not exceed 12^0. The flow leaving the impeller has jets and wakes. When this type of drift enters a massive number of diffuser passages, the standard of float coming into one of a kind diffuser blade passages differs broadly and some of the blades may also face flow separation main to rotating stall and terrible performance. To keep away from one of these opportunities, it is more secure to offer a smaller quantity of diffuser blades than that of the impeller. In some designs the wide variety of diffuser blades is kept one-third of the variety of impeller blades. This association presents a diffuser passage with flows from some of impeller blade channels. Consequently, the nature of flow entering numerous diffuser passages does now not range drastically. Except this the absolutely speed (Mach no) of the flow is decreased at the diffuser access. That is an awesome advantage, especially if the absolute Mach range on the impeller go out is greater than unity. The supersonic drift on the impeller go out is decelerated in this vaneless area at constant angular momentum without shock. Every diffuser blade ring is designed for given flow conditions at the entry at which optimum performance is obtained. Therefore, at off-design operations the diffuser will give poor performance on account of mismatching of the flow. In this respect a vaneless diffuser or a vaned.

Diffuser with aero foil blades is better. For some applications it is possible to provide movable diffuser blades whose directions can be adjusted to suit the changed conditions at the entry. In some designs for industrial applications, a vaneless diffuser supplies the air or gas direct to the casing, whereas for aeronautical applications, various sectors of the vaned diffuser are connected to separate combustion chambers placed around the main shaft.

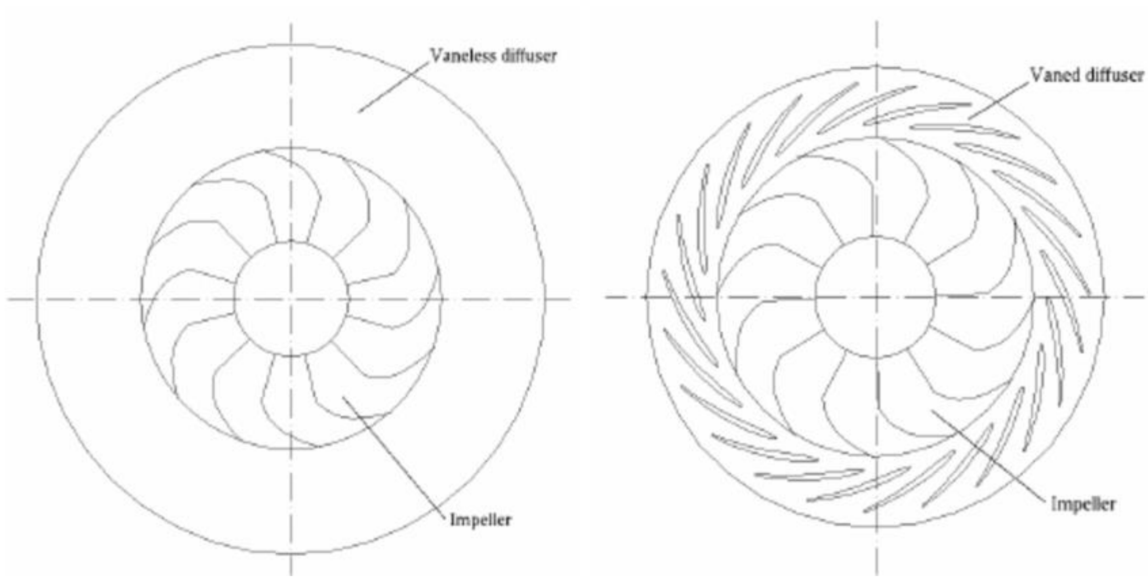


Figure 2: Vanless and Vaned Diffuser

Volute Casing:

The volute or scroll casing collects and guides the flow from the diffuser or the impeller (in the absence of a diffuser). The flow is finally discharged from the volute through the delivery pipe. For high pressure centrifugal compressors or blowers, the gas from the impeller is discharged through a vaned diffuser, whereas for low pressure fans and blowers, the impeller flow is invariably collected directly by the volute since a diffuser is not required because of the relatively low pressures. The volute base circle radius (r3) is a little larger (1.05 to 1.10 times the diffuser or impeller radius) than the impeller or diffuser exit radius. The vaneless space before volute decreases the non-uniformities and turbulence of flow entering the volute as well as noise level. Some degree of diffusion in the volute passage is also achieved in some designs, while others operate at constant static pressure. While the volute performance is dependent on the quality of flow passed on to it from the impeller or diffuser, the performance of the impeller or the diffuser also depends on the environment created by the volute around them. The non-uniform pressure distribution around the impeller provided by its volute gives rise to the undesirable radial thrust and bearing pressures.

Methods and Approach to design:

Centrifugal compressor design is not only dependent upon the aerodynamic factors involved in the modeling process but is also closely repressed by the non-aerodynamic factors involved. Some of these constraints include costs, application usage, the frame area, and the deployment service life. For example, to enhance the pressure ratio delivery, blades of a sturdier material than the standard aluminum would be required to withstand the stresses, consequently increasing the costs. Alternatively, the usage of a greater impeller diameter, or multi-staging the compressors would take a toll on the area constraints. Usually, the design of centrifugal compressors revolves around three major considerations: the design of the impeller, the guide vanes and the volute and its aerodynamic features.

The impeller speed is sometimes limited due to design restrictions. If a greater pressure ratio is required usually the geometry of the vane is modified in the design to introduce a skew. So, the vanes can be radial, forward swept or reverse swept. Hazby and Xu discovered that imparting a forward sweep to the vanes drastically reduces the shock impact on the compressor blades and thus increasing the range in which the compressor operates as well as its efficiency (Hazby, Xu,2007). However forward swept blades usually aren’t very efficient and have a lot of associated noise with them. Backwards swept blades can provide a medium range of pressure ratios, are highly efficient and do not produce as much noise (Pekka et al 2009).

In modern centrifugal compressors, which operate at very high speeds, the flow at the tip of the impeller becomes supersonic. In such cases, the guide vanes cause restriction in the designing of the compressor as its working region is narrowed to a very limited mass flow operating range. So, in centrifugal compressors, the operational range as well as many of the surge and stall related problems are associated with the vanes and not the impellers. This is quite the contrary to axial compressors.

It is experimentally seen that in the case of the axial flow compressors, most of the problems such as surge and stall are associated with the rotating blades or the rotors.

So, the design decision could be to remove the guide vanes to get rid of these problems. However, in some conditions, the guide vanes are needed to introduce an inlet swirl. IGV's avoid fluid from approaching supersonic speeds at inlet, because we sometimes obtain shocks on to the impeller face, which could create problems in terms of a flow separation, and we would have a lot of losses to cater to. So, in most cases we would like to avoid supersonic flow going into the impeller. Therefore, the selection of guide vanes also presents a vital design decision for the designer.

The flow leaves the compressor radially, so if the flow if it is to be supplied, let us say to the combustion chamber, in a gas turbine engine, it must take a radial turn and then get into the combustion chamber. Now, that requires a change of direction, at least by 90 degrees, sometimes more. Thus, we require a guided flow passage which is called a volute. The volute design needs to cater the typical losses, and other aerodynamic features of the volute are needed to be factored into the centrifugal compressor design, because whatever is being supplied from the centrifugal compressor is directed to the volute before it is delivered to the combustion chamber. The size of the volute is decided by the application to which the compressor output is being fed to. So, for the sake of this example, in which a compressor is providing air to an engine, the volute design is heavily dependent on the overall engine design criteria.

The efficiency of a centrifugal compressor is dependent on a multitude of factors such as the sweep of the impeller and the number of blades. Not just the impeller, but also the presence of guide vanes, diffuser, volute, and other components influence efficiency. As a result, both design and integral optimization approaches must be developed in order to increase the centrifugal compressor's performance further. Ch. Ganesh et. al. numerically studied the effect of the sweep in the leading edge of the vanes on the performance and efficiency of the compressor. They observed that the efficiency for forward swept blades is rather less in comparison with backward swept blades. For their study, they found that a maximum efficiency of 89.77% was achieved at an angle of 25 degrees and that mass flow rate and pressure ratio tend to increase when the reverse swept angle is increased from 20 degrees and maximizes at a maximum reverse angle of 25 degree. However, the highest stall margin was observed to be in the case of the forward swept vane geometry at an angle of 20 degrees (Hazby, Xu,2007). Similarly, the interaction of the fluid leaving the impeller towards the diffuser, the gaps between the diffuser and the impeller and the aerodynamic design of the volute casing (Cheng, Muller,2005) have a significant effect on the efficiency. C. Xu and R. Amano have discussed the usage of a various different and unique types of tongued volute designs. They observed that the usage of a large cut back volute tongue helps to get rid of circumferential flow vortices at high pressure near the impeller exit (Xu, C, Amano,2006)

Duccio bonaiut and other authors introduced an approach for the flexibility and invention of hypersonic centrifugal impellers developed using the DOE process in combination with a three-dimensional fluid solution. The geometric parameter is made using Bezier Angles and other mathematical variables named in mesh analysis to calculate the paramount bones. The extent of the variance variables is measured by considering the product conditions. The impact analysis of parallel variables on the performance of the main impeller was divided in two ways: first the authority of the variables operating on the blade formation was studied and a complete configuration was selected and the result of the three operating variables was observed by constraining the already optimized variables. The all approach aimed at the design process of compressor. From the current analysis it has been possible not only to describe the total value but also to see the effect of the variable input on an important machine action (Bonaiuti et al 2006).

Along with Yuri Biba and Peter Menegay have developed a method for determining the variability of the intermediate compressor levels between the limits of a particular action. This is often referred to as the opposite design system. The opposite procedure, to evaluate performance variables formed on statistical inputs, is often referred to as direct analysis or calculation. The mathematical programming and the computer code that forces the opposite path are defined. As a voluntary for the contrasting design cones available on the market, this implement is specially used for OEMs and require a reliable library of mislaying prototypes for each stage feature such as impeller, blades, diffusers, etc. The mathematical programming expands the intensity of the law of the opposite design. by ensuring energy savings in any operating system, such as bad deals. The concept of mislaying rate of moving impellers is developed in the advanced mislaying prototype. The ruling statistics for the preservation of every step element are introduced and explained in the language of a recurring process that measures the needed number of one dimensional. An easy-to-use stoner connector for storing and delivering problems is integrated., Menegay, 2004).

Apart from it, Xinwei shu chuangang gu introduced a complicated blade design diagram of the centrifugal compressor with the help of Unified design method (UDM), Computational fluid dynamics (CFD) simulation Method, Regression analysis method and Genetic algorithms (GA). UDM is used to attain geometric information for patterns to test their CFD method. The practical measure of sample data is made by means of the regression analysis method. At the end, globally improvement of approximate function is done the usage of Genetic Algorithms. This enhancement approach turned into used in the whole creation of specific centrifugal compressor blades thinking about the excessive isotropic performance as a goal feature. The comparison in assessment with the actual ones imply that the isentropic performance of the advanced impeller was improved reflecting the use of the proposed enhancement technique (Shu et al, 2008)

Designing under set parameters:

According to museum of Retro Technology, the basic function of compressor, being able to push air at high pressures, became modern when Viktor Popp designed and installed the first ever compressor plant in Paris in 1988 (Anon., 2018).

Design for Fixed Parameters:

Design of a centrifugal compressor under a given or defined set of parameters has been adopted and presented by different authors under different conditions. Soo-Yong Cho and others adopted an approach by defining fixed design variables for hub and shroud and equal variables for impeller blade optimization and then by using Artificial Neural Networks (ANN) and CFD analysis achieved their desired outputs (Cho, et al., 2012). When designing for the defined outputs, 1D analysis becomes of significance importance which uses the theoretical already present calculations to define geometry and then subsequent aerodynamic and stress analysis are performed (Ibaraki, et al., 2003).

In an effort to design centrifugal compressor for specific speed and desired efficiency, Balje presented plots of efficiency against specific diameter and speed. The findings included that the efficiency decreased with the decreasing specific speed as the density of gas decreased on low pressures with respect to low specific speeds (Baljé, 1962). Dixon presented the thermodynamic approach to the compressor design, by which, we can define the inlet and outlets pressures and then the velocities can be found which further proceed in the help of impeller design (Dixon, 1998).

Along with design of centrifugal compressor for specific parameters, the specific applications also affect the design strategy. De Villiers developed the centrifugal compressors using one-dimensional design strategy to be used in the micro-gas turbine initiated for aero applications which required more efficiency and less noise. With the help of CFD optimization, he was able to obtain efficiency of 76.1% with static pressure ratio of 3.0 (Villiers, 2014).

Application of centrifugal compressors not only influences the design methodology, but also, the influences which geometry and external components to be considered while losing some of the desired outcomes. Dennis and David concluded that a particular compressor combined with a high efficiency electric magnet motor, having low windage and bearing losses, can achieve an EER about one-fourth more than current operating compressors (Pandy & Brondum, 1996). Hyosung Sun and Soogab Lee numerically predicted noise of centrifugal compressor and studied the influence which design parameters have on the noise of the compressor through the Body Element Modeling (BEM) method which is a Euler study implemented approach (Sun & Lee, 2004).

Overall, even the slightest change in the design parameters is going to affect the overall outputs and therefore, it is essential to consider the inputs and outputs and the operating conditions

Teemu, Turunen-Saaresti, Ahti Jaatinen described the impact of many dissimilar drawing variables on top clearance drop of centrifugal compressors. The perceptivity of the centrifugal compressor to top clearance drops changes with dissimilar sketches. Still, it' is crucial for the developer to know the impact of tip clearance drop to be suitable to originally assess the standard of dissimilar designs. The data showed by studding that there's no clear correlation between tip concurrence loss perceptivity and certain velocity, dispersing rate, number of blade and splitter height rate (Saaresti, Jaatinen, 2012)

Wenyang Shao, Xiaofang Wang and other authors proposed to exploration the design parameters for centrifugal compressors with supercritical CO2 under several conditions. The Brayton cycle using (SCO2) has attracted further and further attention worldwide in latest times due to its top cycle effectiveness and closed-packed factors. The compressor has veritably significant factors in cycle. Due to discrepancy to rationally operating fluids, there's a peril of moisture on impeller inlet due to the special parcels of SCO2 closer to the pivotal end. For calculate the probability of moisture, a conception suitable for SCO2 called Condensation Margin (CM) is introduced. A layout parameter referred to as impeller inlet speed ratio (ISR) is described to check the phase of the operating fluid on impeller inlet grounded on CM. The impeller effectiveness, running range and processing method, especially in small length instances, the layout parameters on the impeller outlet are studied by means of chancing a feature of outlet variety wide, variety of blades, rotational velocity, perpendicular velocity measure at outlet and meridional speed measure at outlet. An initial design result of a sco2 low influx degree air compressor is offered as an example of operation of layout parameter disquisition results; additionally, a cfd simulation is done and harmonious effects are attained as compared to the disquisition effects. (Shao et al, 2016)

Role of simulations/CFD in designing of centrifugal compreesor for turbomachinery:

According to V.V.N.K. SatishK., Emanuele Guidotti, CFD plays an adding part in the performance vaticination evaluation in the sketch of ultramodern centrifugal compressor steps with high effectiveness and wide working range. Nonetheless, exploratory fact and figure is precious and obligatory to evaluate the execution of the steps and to more recognize the inflow parcels in detail

Brilliant trials are currently being made to enhance the precision of the arithmetical prototype and the inquiry dimension precision throughout both the sketch and affirmation steps of air compressor steps. This study presents influx analysis of air compressor steps using top dedication computational fluid dynamics with precise attention to melancholy inflow modeling and assessment to exploratory facts

Centrifugal compressor steps with dissimilar inflow portions were used for the confirmation of the arithmetical protypes with specific attention to impact of space inflow on the inflow procedure.

Time- mean fact and figure from the CFD analysis were contrasted to extended exploratory data deduced from the flash pressure inquiry for overall performance of mainstream rates at sketch and off- sketch states. It was constructed that enhancement in computational precision with full data modeling including leakage overflows was significant and the outputs of the computational model accepted well with the exploratory fact and figure. Additionally, the combination of advanced computing calculations and exploratory ways handed deeper perceptivity into the inflow field features (Satish, Guidotti, 2013)

Brett Dewar, Jonna Tiainen, Ahti Jaatinen-Vrri and Mike Creamer proposed that we have several unstable and use of a mathematical model to predict the effects of turbulence which may be tried in analysis like RANS simulations. Simulations are carried out with the assist of varied models. The outputs showed that SST-CC and RSM-Omega carried out properly close by overvoltage. The simulations are accomplished with the industrial cfd software program ANSYS CFX model 171 within the consistent phase. Authors observed on the quit that to forecast the nearby flow phenomena within the air compressor the k-omega sst model is virtually strong analysis tool. In general, the CFD exhibited first-rate reputation with the exploratory information over the entire divergent location, while compressors are designed with CFD for sketch or study functions, generally a single blade bypass is designed (Dewar et al, 2019).

Consistent with Mohammad Omidi Shu-Jie liu Soheil Mohtaram genetic set of rules and CFD techniques increase performance and optimize centrifugal compressor layout. GA is the value of the objective function calculated at randomly chosen points of the domain. In this text a 3-d version of the compressor will be created in bladegen. CFX is the software used on this simulation. It is an Ansys module and is the core for turbomachinery design. The temporary rotor-stator and frozen-rotor simulations are used for connecting distinct elements. The SST (Menter's shear strain transport) version is used for a huge improvement in flow separation. SST also recommends simulating the boundary layer with excessive accuracy. The effects of turbulence are modeled the usage of the usual k-turbulence version. Every time brief interaction effects want to be computed at a sliding frame changeover interface a transient rotor simulation is used. As the brand-new compressor is synthetic by using GA and CFD models its performance became 2% better than that of the bottom compressor. The compressor should be simulated the use of a low mass go with the flow charge to take a look at stall in brand-new compressor simulations imply a 7% put off in mass drift boom in the new compressor (Omidi et al, 2019)

According to V.V. Neverov, Y.V kozhukhov, AM Yablokov, AA Lebedev, optimization the usage of CFD performs a crucial component inside the design technique of turbomachinery. Nonetheless for a hit and powerful optimization it’s important to define a simulation version rightly and rationally. Go with the flow course layout quality is one of the basic factors that determine the general effectiveness of centrifugal compressors. Computerized increment patterns are used for the aerodynamic and mechanical layout of turbomachine elements. There are numerous enhance strategies which might be amazing for chancing a maximum suitable consequence. Utmost CFD software program programs encompass the overall cycle optimization set of guidelines with geometric model parameterization determine and mesh regeneration equipment.

This permits cfd operators to fluently use geometric CFD optimization software application to mechanically set off design versions. (Neverov et al, 2017)

S. Amano took help the industrial (CFD) model ANSYS CFX-11.0 for the determine in Meridional Considerations of the air Compressor growth, R (Amano, 2012)

Mei Han and other authors used an associated variable and tolerance sketch to optimize the assertion values and flexibility of a ten mathematical commutative air compressor sketch formed on a fluid-based computer simulation. The standard of air compressor is measured by the compatibility of the 3 KPI and correcting the simulation straight takes time. Gaussian process gp emulators are designed to predict PMs. Eight tolerances will be relaxed and two strengthened. The impact of design flexibility on quality is better understood. The top-quality way to the formulated IPTD trouble acquired by using the proposed approach presents an improved and within your budget compressor sketch than the existing sketch. New knowledge which may be very beneficial for engineers, is likewise received from the most suitable compressor sketch. The suggested iptd methodology can be carried out to other layout packages with time-consuming computer fashions in which its miles affordable to equate the anticipated degradation to a linear feature of failure probabilities including other aerospace engineering implementations (Han et al, 2019).

Performance prediction methods for centrifugal compressor:

According to Hongsheng Jiang and other authors, centrifugal compressors are a usual kind of turbomachinery, which are very commonly used in various engineering fields. For sketch, computing and effective of the centrifugal compressor, performance is repeatedly need at the compressor ratio, productivity, work, and other heat motion variables. So, forecasting execution of the centrifugal compressor is one of the important stages in the computations

For predicting compressor performance among from many models like kriging model is often used. It gives greatest linear impartial prediction of the middle numeric and notably decreases the computational difficulty. It is combined with various types of permutation models to build an exact precision compressor characteristic prediction prototype. (Jiang et al,2019)

CFD approach is very usual approach to measure the compressor execution. In many cases, however, the outcomes points are frequently very insecure to densely protect the entire workspace due to time and computational limitations. As the result, a satisfactory method is required to reassemble the chart with the help of restricted number of selected figures for efficiently give big-accurate figure at less amount (Neverov et al, 2017).

According to Zhang Chaowei and other authors describe that if you design the effective centrifugal compressor initial performance prediction is a very important stage aspect. Some undisclosed variables in one dimensional design forecast process for example: frictional losses, stage losses, vibrational effect, slip factor rely on experimental connections or supposition. From variables, loss connections play a crucial role in forecasting the centrifugal compressor execution. This article gives us many loss connections for the similar loss procedure. If we want to check adequate precision centrifugal compressor actions, it is essential to measure the good loss connection proposition.

The loss connection proposition has the shock loss too. But Aungier gave the methods in which it has not any shock loss, but the compression ratio of the centrifugal compressor has just less than 3.6 in model making circumstance, (Chaowei et al, 2019)

In addition, Gong and Chen proposed many losses connection by measuring the available one-dimensional loss connections, consisting of the above author shock loss connection, however the compression ratio just comes up to 2.2. Greater compression ratio and greatly forced centrifugal compressors are usual utilized right now, but they do not have adequate precision in execution forecast process according to their loss connection. Until now requirement for higher precision and trustworthy loss connection proposition. (Gong, Chen,2020)

There are many losses connections proposition suggested. Like, Galva's suggested the one-dimensional design to forecast the execution of a centrifugal compressor along the divergent channel having unique sketch circumstances. Sadly, the compressor design has general inadequate precision. (Galvas,2019)

Authors proposed the design of air compressor to research the complete impeller losses in comprehension. Whitfield and Baines established such a centrifugal compressor design for execution forecasting which has compressor ratio comes up to 2.4. Another author described the loss connections proposition to forecast centrifugal compressor execution with trying out interior and exploitative loss connections. With these methods which has the compressor ratio comes up to 2.1. In the A. Whitfield article, the analysis assumed state-of-the-art efficiencies, and while the design process sought to reduce the comparative Mach numbers for reduce losses, it is essential to carry on the analysis by including actual loss designs to proof the supposed effectiveness's (Conrad et al,2019)

Within the article Via Elkin I Gutierrez Vela Squez,it may be concluded that making use of an inappropriate set of correlations can lead to performance loss predictions which could break out with values as excessive as 8% in the efficiency of the air compressor phase, which represents a major mistakes inside the initial stages of the manufacture of that compressors.(VELA SQUEZ, 2017)

According to MingYao Ding, Clinton Groth, Suresh Kacker and Douglas Roberts, they described the methods for better performance prediction in designing of centrifugal compressor, like

First, they generated CAD geometry in CAD software. Then, mesh formation was made with the help of Ansys mesh formation model. They produced the two meshes (for impeller and diffuser). The CFD solver used was ANSYS CFX. CFX was chosen to tackle the complex problems faced. Then they applied boundary conditions. In entry, the boundary condition was applied as a Mach number less than 1 at inlet. At the impeller periodic conditions were implemented. The Vanes, hub and shroud were implemented as adiabatic walls with the proper circular speed. Unstable processes are very important and the mixing plane supposition below solid. Surge forecasting is also an unstable process. The outcomes of the unstable centrifugal compressor calculations show that totally unstable flow examination can produce greater execution for castings specifically at top revolving circular velocity (Robbert et al, 2001)

Hongsheng Jiang, and others author generated the representative figures with the help of Ansys model Vista CCD, which forecast or prediction the execution of the centrifugal compressor. Authors collected such a reversion design, which was integrated with the compressor's loss design. Such design-based model preserves the effectiveness and compression ratio trends and forecasting outcomes are so nearby the genuine facts and figures. The outcomes present that the forecasting outcomes of such loss model design and the kriging model are comparative steady. (Jiang et al, 2019) empirical loss models to verify the assumed efficiencies (Jiang et al, 2019)

CHAPTER 3: METHODOLOGY

There are various approaches that can be adopted to design a centrifugal compressor. The main factor in the selection of design approach is the parameters that are available for the design. The aim and goal of the design is another thing that defines which way will be used for the design of centrifugal compressor. Yet, one thing that remains constant in the overall process is the correlation between different parameters of the compressor. Most of the correlations of the compressor’s parameters are given by turbomachinery pioneers such as Stodola, Japikse, Hawthorne, Baljie and Robinson. The correlation results used by these researchers are used today and define the fundamental approach for design of centrifugal compressor.

The approach used in this work is fairly based on the works of all the pioneers except the fact that it is aimed to design the compressor for fixed inlet and outlet conditions requirements. The design process will begin by setting design points that will indicate the output required from the compressor. This, merged with input, gives a track of different components of compressor that are required to be designed. The secluded design of different components of compressors depends on the parameters of the design. The major components of the compressor are shown in figure. The most important design step is the integration of all the major components of the compressor.

(Thorat, 2022)

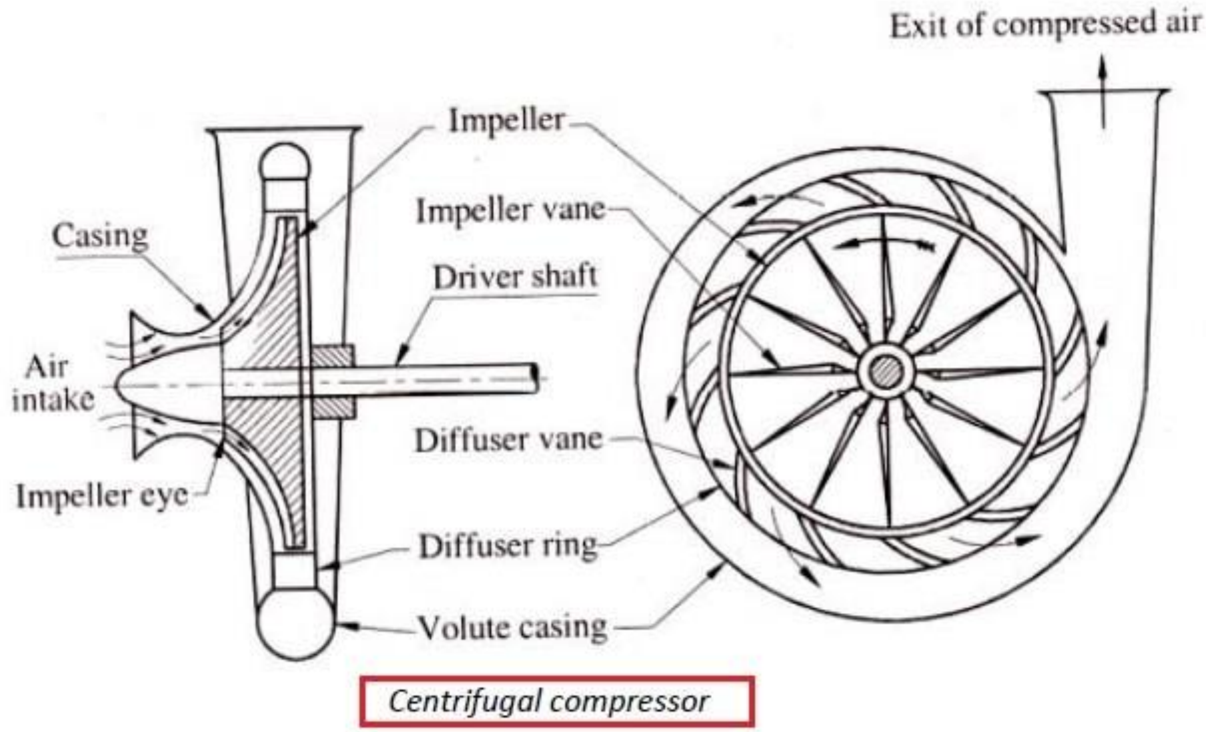


Figure 3: Different parts of a compressor

Preliminary Calculations:

Design of Impeller:

An impeller is the major component of a centrifugal compressor and thus follows a difficult design approach as compared to other components of compressor. The impeller is the rotating component of the centrifugal compressor which is made of blades and thus the motor is also attached to this component. The process of compression majorly happens in the impeller too. The compressor inlet is the first part of the centrifugal compressor, and the operation of compression ideally starts from here.

Impeller Inlet:

The impeller inlet, also known as inducer, is the entry of fluid into the compressor. The fluid enters the compressor. The impeller transfers its own rotational energy i.e., the shaft work into the fluid by increasing its overall energy level. It can be consisted of a single piece or part of an inducer section and a radial portion. The basic function of impeller inducer is to receive the fluid and provides a path towards the radial portion of the compressor. The impeller inlet design applies the use of inlet conditions of fluid and configures the inlet according to the structurally integrated equations and relations. The impeller diagram along with inducer is shown below in figure below.

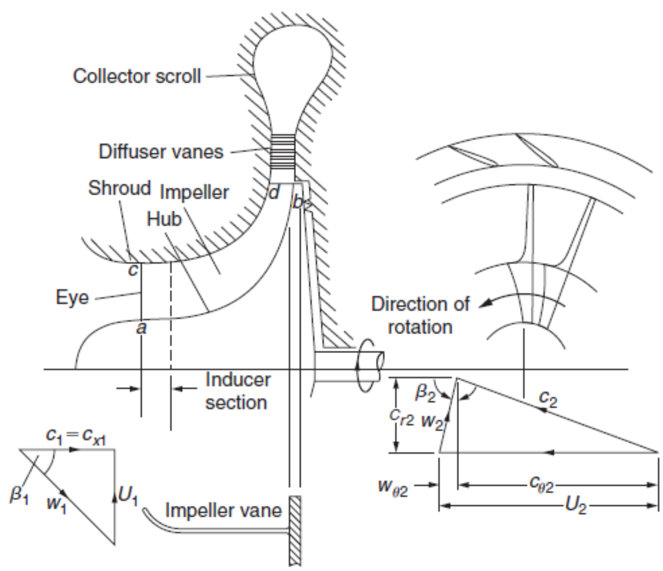


Figure 4: Impeller of compressor

Inlet Guide Vanes:

An addition at the impeller inlet is the use of inlet guide vanes (IGVs). The inlet guide vanes, evident by their name, provide the function of directing the flow of fluid into the compressor at specific angle. The angle of the inlet guide vanes actually gives the flow a direction to move into the impeller. The use of inlet guide vanes is also considered as the introduction of pre-swirl. The pre-swirl addition helps the compressor from the larger incidence angle at the inlet. The incidence angle is advised to be always less than $4^o or\ 5^o$. There is no accurate calculation of incidence angle at the inlet, rather, an estimation of results is used through software tools.

The usage of inlet guide vanes is shown in the figure below.

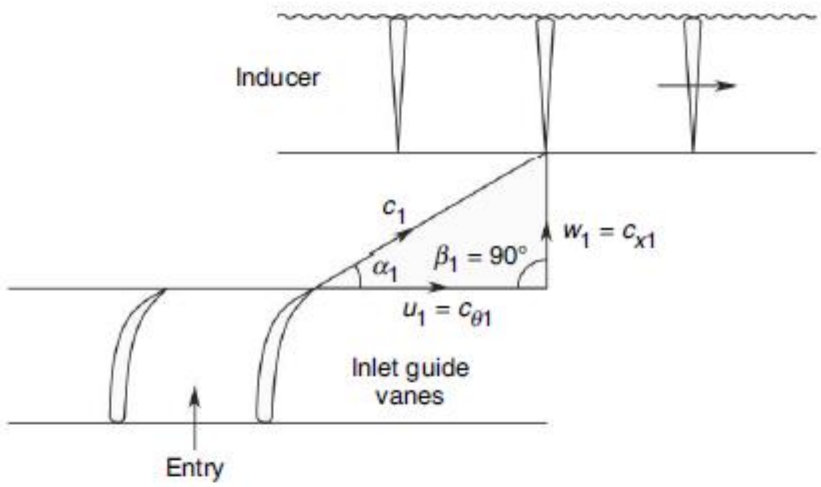


Figure 5: Inlet Guide Vanes usage

Impeller Blade Configurations:

The impeller blades can be radially extending from hub towards the shroud or can be curved in either way. The unique configurations form the three types of impeller blades being radial, forward blades and backward blades. The forward blades are inclined in the forward direction such that the leading edge of the blade is at a point ahead of the trailing edge when viewed on the hub axis. The backward inclined blade is opposite to the forward curved blades.

Velocity Diagrams:

The inducer section is designed by considering the concept of velocity diagrams. The velocity diagrams relate the absolute, relative and axial velocity. For this work, the basic design of impeller starts from the consideration of velocity diagrams. The velocity diagrams form an integral component of the design process. Moreover, both inlet and outlet of the impeller features velocity triangles for the compressor. Therefore, the velocity components will be analyzed both at the inlet and outlet. Two angles are specified at the inlet and outlet. The angles are measured from tangential direction at any given point. The absolute angles at inlet and outlet are defined as α_1 and α_2 while relative angles at inlet and outlet are given by β_1 and β_2 respectively.

The velocity diagrams used for the purpose of analysis are shown below in figures.

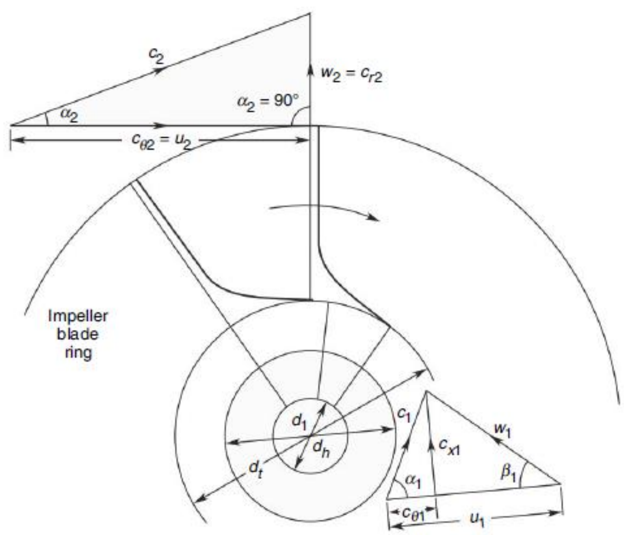


Figure 6: Velocity Diagram for radial blades

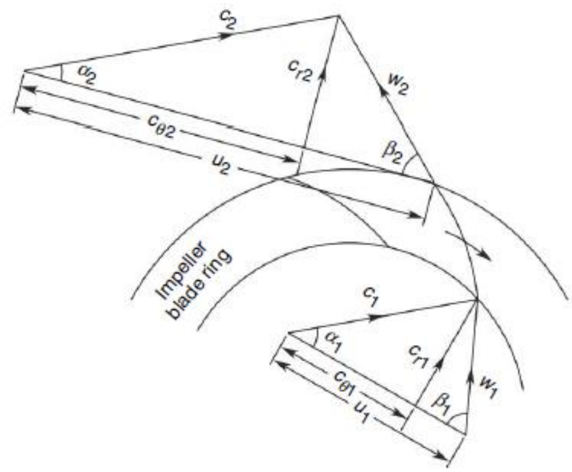


Figure 7: Velocity Diagrams for forward blade impeller

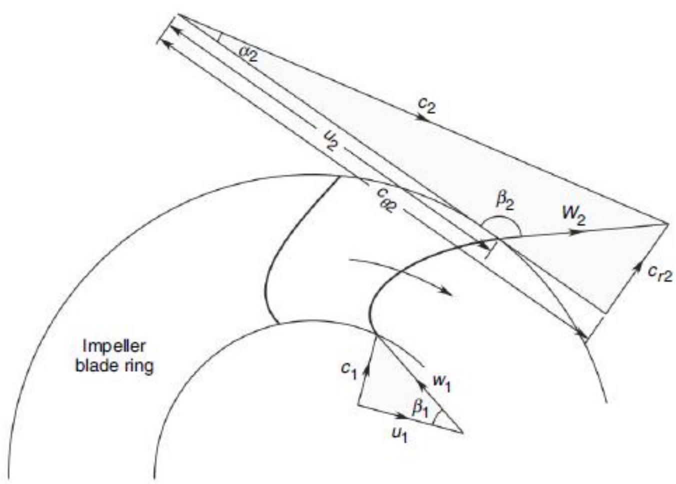


Figure 8: Velocity Diagram for Backward blade impeller

Were,

$$u = tangential\ velocity$$
$$c = absolute\ velocity$$
$$w = relative\ velocity$$

$$c_x = axial\ component\ of\ velocity$$

$$c_\theta = swirl\ component\ of\ velocity$$

The relations used for the inducer inlet design are given as:

$$u_1 = \frac{\pi d_1 N}{60}$$

$$u_2 = \frac{\pi d_2 N}{60}$$

All other components of velocities are related by trigonometric relations and are justified by the velocity diagrams at inlet and outlet.

Hub and Should:

The hub is component of the impeller that forms the inner wall of flow path of the compressor. The shroud forms the outer wall of the impeller. Both are the integral component of the compressor and during passing through the impeller, the flow actually passes between hub and shroud. The blades of the impeller are also located between these two. The hub and shroud design are the main component for the impeller.

The outer radius of impeller can be found from the specific work. While the specific work is given as:

$$Specific\ Work = \frac{Power\ Input}{Mass\ flowrate}$$

$$\Delta W = \frac{P}{\dot{m}}$$

Since,

$$\Delta W = U_2 c_{\theta 2}$$

While from the Stanitz expression, we have, for fixed number of blades Z,

$$\sigma = 1 - \frac{0.63\pi}{Z} = \frac{c_{\theta 2}}{U_2}$$

Here the outer radius can be easily found as follows:

$$U_2 = \sqrt{\frac{\Delta W}{\sigma}}$$

$$r_2 = \frac{U_2}{\Omega}$$

The inlet of the impeller design begins by specifying the design for the hub and shroud. This can be done by selecting a definite ratio of shroud radius to the impeller outer radius r_{s1}/r_2. From literature, we know that this ratio lies in the range of 0.35 to 0.65 and so based on this ratio, we can select another ratio between axial velocity and the blade speed at the tip c_{x1}/U_{s1}. The second ration lies in the range of 0.4 to 0.5. This leads use to calculate ratio of hub-tip radius using the equation of continuity. The goal in this analysis is set to be minimum relative Mach number at the inlet $M_{1,rel}$. The procedure can be repeated to get a satisfactory value for the Mach number.

A better yet direct method is usage of a literature theory developed. According to it, the relative Mach number at the inlet is a function of certain parameters and given as:

$$f\left(\mathrm{M}_{1,\mathrm{rel}}\right) = \frac{\Omega^2 \dot{m}}{\pi k p_{01} \gamma a_{01}} = \frac{\mathrm{M}_{1,\mathrm{rel}}^3 \sin^2 \beta_{s1} \cos \beta_{s1}}{\left(1 + \frac{1}{5}\mathrm{M}_{1,\mathrm{rel}}^2 \cos^2 \beta_{s1}\right)^4},$$

Were,

$$k = 1 - \left(\frac{r_{h1}}{r_{s1}}\right)^2$$

Considering the absolute angle at input to be 0, i.e., $\alpha_1 = 0$ and thus for a fixed value of $\mathrm{M}_{1,\,\mathrm{rel}}$, the value of relative angle at inlet, β_{s1}, can be found for a maximum value of the function stated above. Differentiating the above factor, it can understand that the maximum value of this function occurs when,

$$\cos^2 \beta_{s1} = X - \sqrt{X^2 - 1/\mathrm{M}_{1,\mathrm{rel}}^2}$$

Were,

$$X = 0.7 + \frac{1.5}{\mathrm{M}_{1,\,\mathrm{rel}}^2}$$

Different values of angles can be calculated for fixed values of relative Mach number. A table will be printed and shown in results section. An optimum value of relative Mach number is then chosen which usually corresponds to lower satisfactory value. The dimensions of other parameters at the inlet can be found from the continuity equations and its correlated equations.

As, continuity equation is given as,

$$\dot{m} = \rho_1 A_1 c_{x1}$$

$$r_{s1}^2 = \frac{\dot{m}}{\pi k \rho_1 c_{x1}},$$

Were,

$$\rho_1 = \frac{\rho_{01}}{\left[1 + \frac{1}{5}\mathrm{M}_1^2\right]^{2.5}}$$

$$c_{x1} = \mathrm{M}_1 a_1,$$

So, the absolute Mach number is given as,

$$\mathrm{M}_1 = \mathrm{M}_{1,\mathrm{rel}} \cos\beta_{s1} = 0.7 \times \cos 57.94 = \mathbf{0.3716},$$

The speed of sound is given as,

$$a_1 = \frac{a_{01}}{\left[1 + \frac{1}{5}\mathrm{M}_1^2\right]^{0.5}} = 338.5\ \mathrm{m/s},$$

and

$$c_{x1} = 0.3716 \times 338.5 = 125.8\ \mathrm{m/s}.$$

The density relations can be utilized as,

$$\rho_{01} = \frac{p_{01}}{(RT_{01})} = \frac{1.249\ \mathrm{kg}}{\mathrm{m}^3}$$

$$\rho_1 = \frac{1.249}{1.0704} = \frac{1.1669\ \mathrm{kg}}{\mathrm{m}^3}$$

Thus,

$$r_{s1}^2 = \frac{7.5}{(\pi \times 0.8037 \times 1.1669 \times 125.8)} = 0.02024 \text{ m}^2$$

$$r_{s1} = 0.1423 \text{ m and } r_{h1} = 0.0630 \text{ m.}$$

Efficiency considerations for the impeller:

The efficiency of the impeller is also considered in certain analysis to make sure that the results conform to the reality. For our analysis, we have assumed efficiency of 92%. This value is fairly acceptable for optimized specific speeds from 0.6 to 0.7. The construction of impellers always directly impacts the efficiency of the impeller. It has been observed by Rodgers (1980) that the impellers which have backward blade swept blades have an efficiency usually 2-3% more than the radial blade impellers (Rodgers, 1980).

Considering the specific speed as,

$$N_s = \frac{\phi^{0.5}}{\psi^{0.75}}$$

Where flow coefficient is given as,

$$\phi = \frac{c_{x1}}{U_2} = 118.7/373.4 = 0.3179$$

And pressure coefficient is given as,

$$\psi = \frac{\Delta W}{U_2^2} = 126.3 \times 10^3/373.4^2 = 0.9058$$

$$N_s = 0.607$$

Impeller exit:

The radial component of velocity at exit is often chosen equal tot eh axial component of velocity at the impeller entry such as

$$c_{r2} = c_{x1}$$

The absolute velocity at the outlet is therefore given as,

$$c_2 = \sqrt{c_{\theta 2}^2 + c_{r2}^2} = 360.8 \text{ m/s}$$

The flow angle will be given as,

$$\alpha_2 = \tan^{-1}\left(\frac{c_{\theta 2}}{c_{r2}}\right) = \mathbf{69.60}^{\circ}$$

For calculation of width of impeller exit, density is required at the exit. Therefore, this value is determined as,

$$\rho_2 = p_2/(RT_2)$$

For density, we will use the efficiency correlations as:

$$\eta_i = \frac{h_{02s} - h_{01}}{h_{02} - h_{01}} = \frac{T_{02s}/T_{01} - 1}{T_{02}/T_{01} - 1}$$

and

$$\frac{T_{02}}{T_{01}} = \frac{\Delta W}{C_p T_{01}} + 1 = \mathbf{1.4289}$$

Based on the value of T_{01}, we can find the value of T_{02} from the ratio of $\frac{T_{02}}{T_{01}}$.

Using efficiency relation, the value of $\frac{T_{02s}}{T_{01}}$ can be found as thus pressure ratio can also be found as:

$$\frac{p_{02}}{p_{01}} = \left(\frac{T_{02s}}{T_{01}}\right)^{\frac{\gamma}{\gamma-1}}$$

And thus, value of p_{02} can be found for density calculations.

$$T_2 = T_{02} - \frac{c_2^2}{2C_p} = 353.9 \text{ K so } T_2/T_{01} = 1.2080 \text{ and } T_{02}/T_2 = 1.1830,$$

$$p_2 = p_{02}/\left(\frac{T_{02}}{T_2}\right)^{\gamma/(\gamma-1)} = 186.7\text{kPa}.$$

Hence,

$$\rho_2 = \frac{p_2}{RT_2} = 186.7 \times 10^3/(287 \times 353.9) = 1.838 \text{ kg/m}^3$$

According to continuity equation, we have,

$$\dot{m} = \rho_2 A_2 c_{r2}$$

While,

$$A_2 = 2\pi r_2 b_2$$

So,

$$b_2 = \frac{\dot{m}}{(2\pi\rho_2 c_{r2} r_2)} = 0.0195 \text{ m} = 1.95 \text{ cm},$$
$$\frac{b_2}{r_2} = \frac{1.95}{26.5} = 0.0736.$$

At impeller exit the Mach number,

$$\mathrm{M}_2 = \frac{c_2}{a_2}$$

Were,

$$a_2 = \sqrt{\gamma R T_2} = 377.1 \text{ m/s}$$

$$\mathrm{M}_2 = 360.8/377.1 = \mathbf{0.957}.$$

Flow in the vanless space:

When the fluid leaves the impeller, it enters a vaneless region. This region exists between a moving part of the compressor and fixed or non-moving part of the compressor. To avoid any form of leakage in this part, the flow is monitored carefully for analysis. Typically, the flow that is leaving the impeller is usually turbulent with flow having regions of separation. This non-uniform flow can have strong effects on the performance of the diffuser. Before entry into the diffuser, the flow gets some diffusion due to the vaneless space and therefore a significant reduction of flow irregularities takes place in this region.

The radius of the vanless space, denoted here as r_{2d}, is proved to have a relation with the impeller outer diameter. This radius is considered to be 1.1 times the impeller outer radius as the minimum. The increase in this ratio results in the reduction of Mach number of fluid when it enters the vanes of the diffuser. The axial width of the vaneless space is considered to be constant as of the impeller exit b_2.

The flow in the vanless space is considered to be frictionless by assuming that the tangential momentum is this space is conserved within the vaneless space. The flow, when assumed to be incompressible, can be approximated by a logarithmic spiral flow path described by following equation.

$$\Delta\theta = \theta_3 - \theta_2 = \tan\alpha_2 \ln\left(\frac{r_3}{r_2}\right)$$

The vaneless space radius can be assumed as the minimum.

$$r_{2d} = 1.1 r_2$$

Comparing the radial and swirl components rations, we have,

$$\frac{c_{\theta 2d}}{c_{\theta 2}} = \frac{r_2}{r_{2d}}$$

$$c_{\theta 2d} = \frac{338.2}{1.1} = \mathbf{307.5}\,\frac{\mathbf{m}}{\mathrm{s}}$$

$$c_{r2d} = \frac{r_2}{r_{2d}} c_{r2} = \frac{114.36\ \mathrm{m}}{\mathrm{s}}$$

$$\alpha_{2d} = \cos^{-1}\left(\frac{114.36}{307.5}\right) = \mathbf{68.16}°$$

$$c_{2d} = (c_{2d}^2 + c_{r2d}^2)^{0.5} = 328.1\ \mathrm{m/s}$$

$$T_{2d} = T_{02} - \frac{c_{2d}^2}{(2C_p)} = 418.7 - 328.1^2/2010 = 365.2\ \mathrm{K}$$

$$a_{2d} = (\gamma R T_{2d})^{0.5} = 383.0$$

$$\mathrm{M}_{2d} = \frac{328.1}{383} = \mathbf{0.856}$$

Density Changes in Vanless Diffuser space:

The fluid exiting the impeller is, as we know, an irregular flow and suddenly enters a space which as sudden area increase too. An iterative procedure can adopt to predict the change in the density and the Mach number. One iteration will be carried out as given in the following structure. The radius of vanless space will be used to find the velocity in the vaneless space.

$$c_{r2d} = c_{r2}\left(\frac{r_2}{r_{2d}}\right) = 125.8/1.1 = 114.3.$$

Hence,

$$c_{2d} = \left(c_{\theta 2d}^2 + c_{r2d}^2\right)^{0.5} = (307.5^2 + 114.3^2)^{0.5} = 328.06\ \mathrm{m/s}.$$

Then, the values of T_{2d} and p_{2d} will be determined as follows,

$$T_{2d} = T_{02} - \frac{c_{2d}^2}{2C_p} = 418.7 - \frac{328.062}{2010} = 365.2\ \mathrm{K},$$

$$p_{2d} = p_{02}/\left(\frac{T_{02}}{T_{2d}}\right)^{\frac{\gamma}{\gamma-1}},$$

$$\mathrm{p}_{2\mathrm{d}} = 336.3 \times 10^3/(418.7/365.2)^{3.5} = 208.4\mathrm{kPa}$$

$$\rho_{2d} = \frac{p_{2d}}{RT_{2d}} = \frac{208.4 \times 10^3}{287 \times 365.2} = 1.988 \text{ kg/m}^3$$
$$A_{2d} = 2\pi r_{2d} b_2 = 2\pi \times 0.2915 \times 0.0195 = 0.03572 \text{ m}^2$$

so that

$$c_{r2} = \frac{\dot{m}}{(\rho_{2d} A_{2d})} = 7.5/(1.988 \times 0.03572) = 105.6 \text{ m/s}$$
$$c_{2d} = (105.6^2 + 307.5^2)^{0.5} = 325.1 \text{ m/s}$$

The iterations will be repeated until a converging result is achieved for the Mach number. The values of Mach number and angle will be calculated as follows,

$$\mathbf{M}_{2d} = \frac{c_{2d}}{\sqrt{\gamma RT_{2d}}} = 324.9/\sqrt{1.4 \times 287 \times 366.1} = \mathbf{0.847}$$
$$\alpha_{2d} = \tan^{-1}\left(\frac{c_{\theta 2d}}{c_{r2d}}\right) = \tan^{-1}(307.5/104.98) = 71.15°$$

When the flow is approximated with a spiral flow analysis, it can be observed that their insignificant change int eh density and Mach number.

The volute:

The volute is last component of the compressor, and it features the flow going out from the compressor. The design of volute is often dependent on the application as it feeds the flow from the compressor to the attached part. For example, in a refrigeration cycle, the fluid exiting the compressor will be sent to the condenser section.

CAD Model:

The preliminary design is converted into the live form through the use of CAD Software. Our CAD model is one the of the basic elements of our project. The model will be used for further analysis, 3D printing and prototype development. For the turbomachinery components, the availability of CAD software is an essential problem and gets in the way of effective designing. The complexity of the centrifugal compressor requires more digitized designing methods. The CAD model for this project, therefore, will be completed in CFturbo application.

CFTUBRO is one of the major CAD modeling software for the turbomachinery. The process of designing a turbomachinery equipment is truly straightforward and in accordance with all of the literature. The method employs the literature developed models and curves from Cordier Diagram to Bezier Curves. The basic interface of CFTUBRO shall be indicated in appendix. The process of CFturbo design begins with specifying the dimensions for the main component i.e., impeller. The impeller defines the relationships with other components of the compressor. This, in terms of the design methodology gives a step-by-step overview of the modeling.

The advantage of the CFTURBO is its usability in the overall design process. The exported model from CFTURBO can be utilized to perform computational fluid dynamics (CFD) analysis in other software and for the structural analysis in software like

ANSYS workbench or COMSOL. The availability of these tools is helpful in the overall design process.

The generated model from the CFTURBO is show below in meridian plane. The utility of CFTURBO can be visualized in the form of all components designed perfectly. It can be visualized from the model that a pipe is first connected with the impeller and then the impeller is attached with the diffuser. The volute, which is the last component of the compressor, is attached with the diffuser. It can be seen that the area increase in the volute is significant. This marks how varied the overall flow comes out of the compressor.

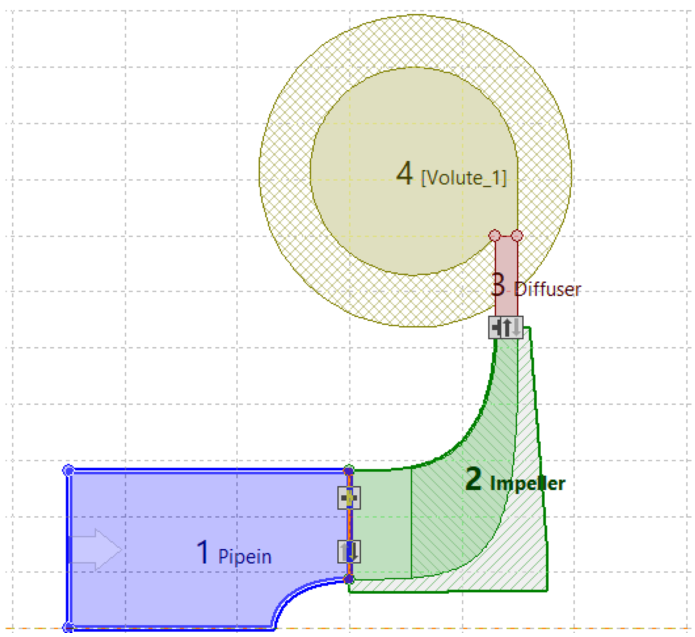


Figure 9: Meridional Figure of compressor from CFTurbo

The CAD model exported for further analysis is shown below.

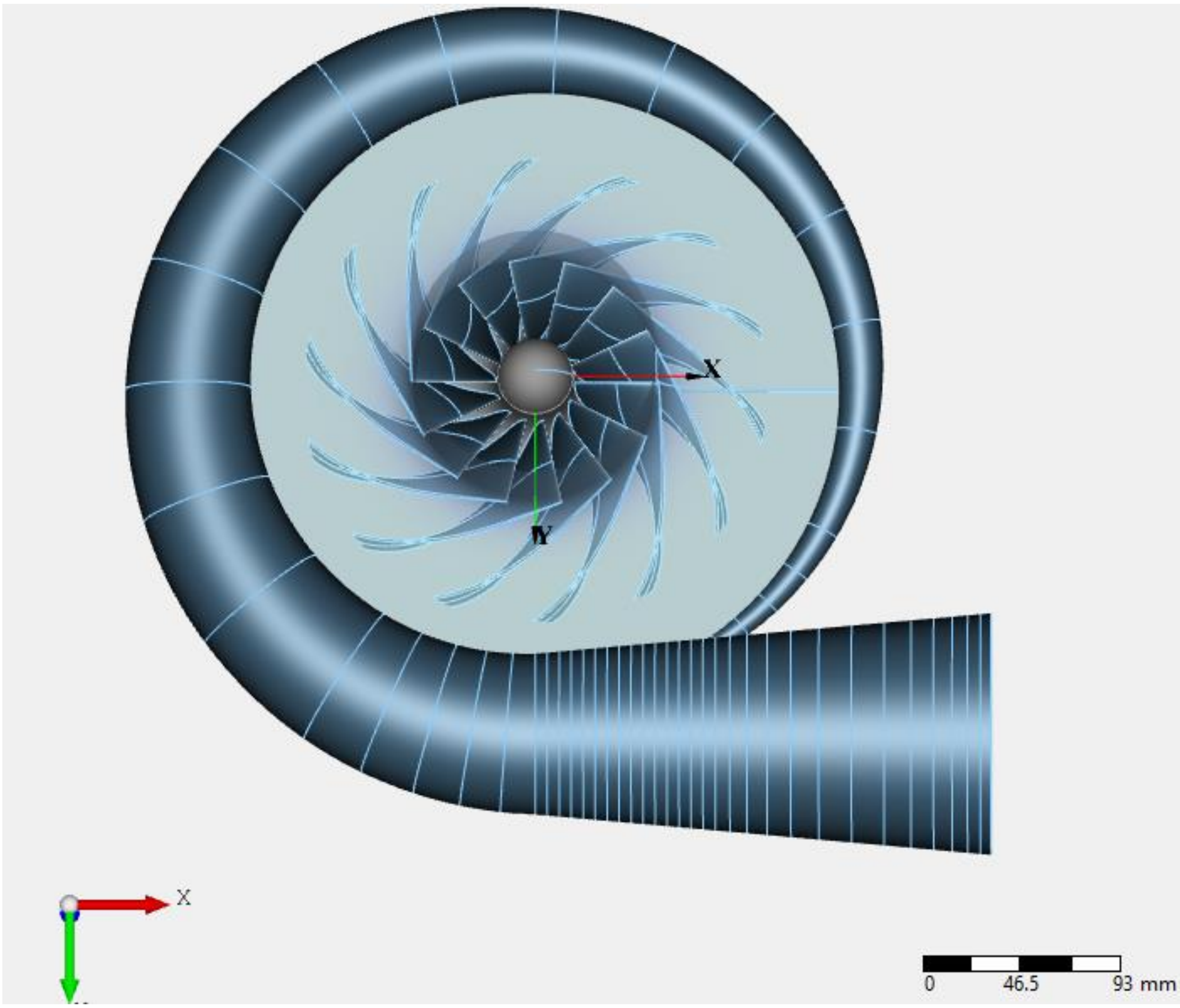


Figure 10: CAD model of compressor

CFD Analysis:

After successfully modeling the centrifugal compressor on CFturbo, we narrowed down to ANSYS FLUENT and ANSYS CFX as our major choices for software based computational fluid dynamics analysis. While ANSYS FLUENT has the advantage of being able to expand the study on a broad range of studies, it however necessitates a thorough mesh analysis. In this respect, CFX is unique as it is mainly intended for the examination of turbomachinery and provides a much more tailored experience, which is especially useful when it comes to geometry generation and meshing. Another benefit to CFX is that it requires less computational power in comparison to its counterpart. Therefore, we used the later of the afore mentioned two options in order to perform our CFD analysis. We performed the CFD analysis using the ANSYS Vista CCD module. The results from CFturbo were obtained at design point and were input into ANSYS as the parameters for our CFD study.

Table 1: Parameters of CFD analysis

Design Specifications	
Pressure Ratio	1.5
Mass flow Rate	1 kg/s
RPM	20000
Efficiency	0.85

Geometric Parameters	
Impeller Outer Diameter	300 mm
Inducer Hub Diameter	80 mm
Inducer Tip Diameter	193 mm
Overall Length	140 mm

For the initial step of sizing the design, VISTA CCD was used. This module is especially helpful in assisting design development from a crude conceptual phase to a 3D parametric model. This also enabled us to bypass the arduous task of defining the geometry of the compressor. In order to model the conditions in Vista CCD, we input the parameters as mentioned in the following table:

Design Creation for Analysis:

In the next step, it is required to specify the behavior of the working fluid at our specific design point. As we are modeling for air and are considering only single stage compression, ideal gas behavior can be confidently chosen as air can be approximated as an ideal gas at the specified pressure and temperature after only a single stage of compression. Using these parameters, Vista CCD generated the following results:

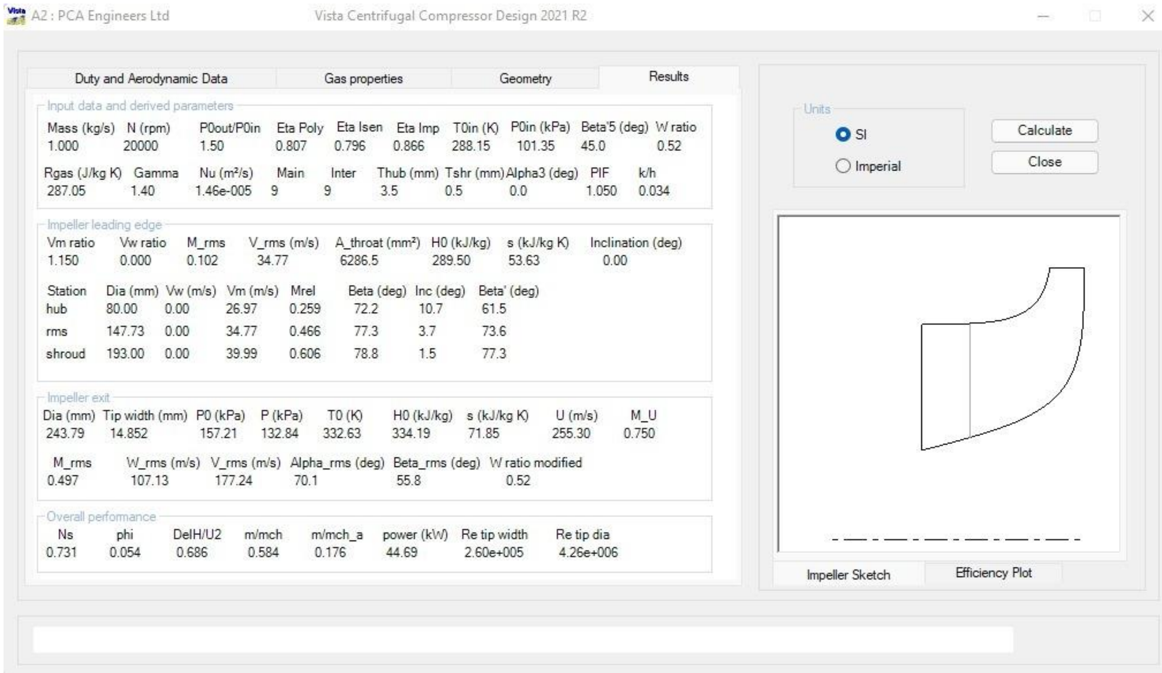


Figure 11: ***Input Data and Derived Parameters***

In the next step, we utilize the functionality provided by Blade-Gen in order to adjust the geometric parameters and modify the control points on the Bezier curves that enable us to adjust the blade geometry and tweak it for optimal performance. Adjusting the control points of the Bezier curves enables us to tweak the geometric parameters such as the variation in the angle along the progression of the meridional plane or along the span, thickness of the normal layer. In this module, we can also adjust the areas through the flow traverses at the inlet and outlets. The tweaked geometry obtained from Blade-Gen is then imported into Turbo-Grid where it can be meshed.

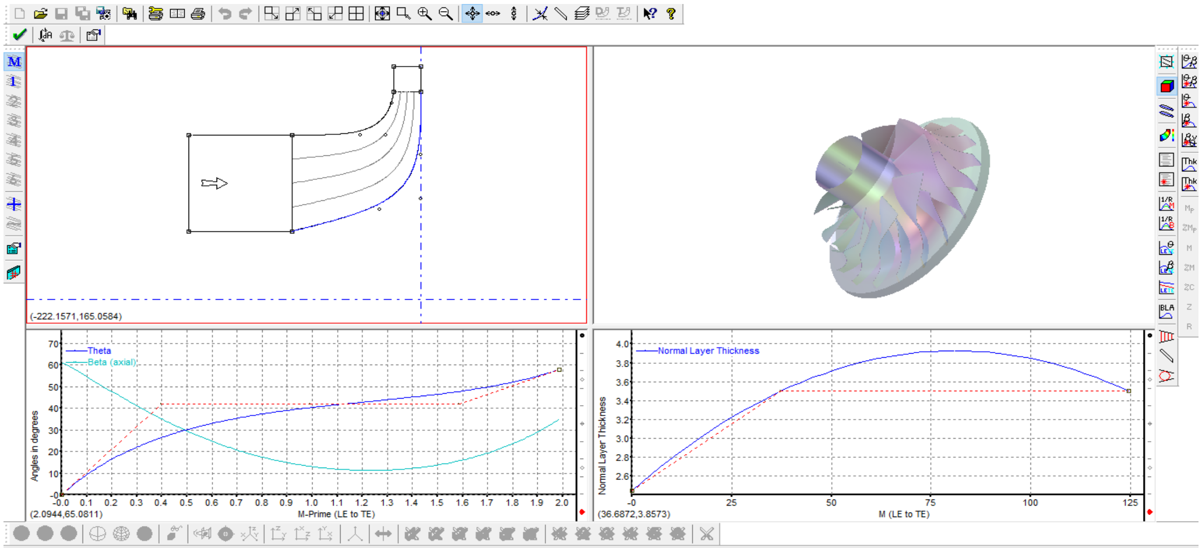


*Figure 12: **Impeller Design***

Meshing For Analysis:

Turbo-Grid is a module that is used for high quality meshing of turbomachinery. It generates a mesh grid for a single blade of our compressor, which can be patterned circularly to consider the geometry of the whole compressor. The entry and the exit behavior of the fluid is observed. The Hub and shroud of the compressor are also highlighted, where the fluid behavior can be observed. Y+ variable is also introduced to the calculation near the boundaries in order to ensure that the correct grid is generated. For a flow with higher Reynold number values, the interaction between the blade and the fluid must also be considered and is not negligible. A high quality (fine) mesh is generated by employing mesh control. The topology is suspended, and the inlet and outlet areas are trimmed to focus on the fluid's interaction with the vanes. This generated mesh is then exported to the CFX module.

So, using Turbo-Grid, we did the meshing of Impeller and Volute separately. For this we had to import the 3D model of compressor from CF-Turbo, but it was not possible to do it directly, so we imported the model to Solidworks in which we solidified all the surface by giving them suitable thickness. After that, the files were converted into IGIS form which were then imported into Ansys for CFD Analysis. Then the network of different modules for complete analysis was created in this way:

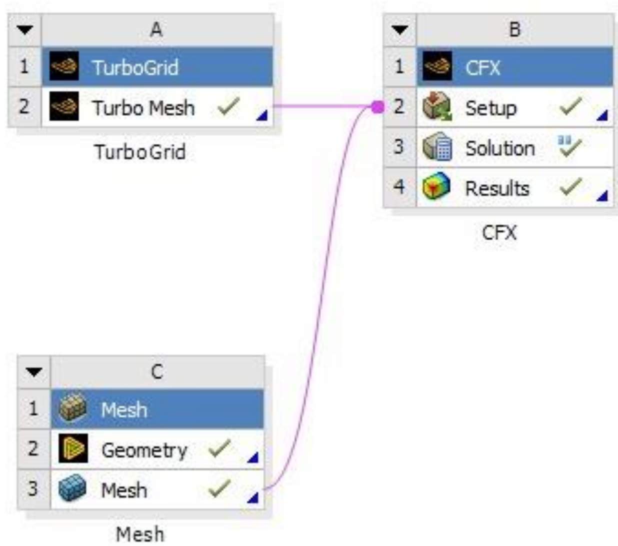


Figure 13: Tree of Analysis

Network Diagram

In Turbo-Grid, the blade profile was further checked and optimized for meshing:

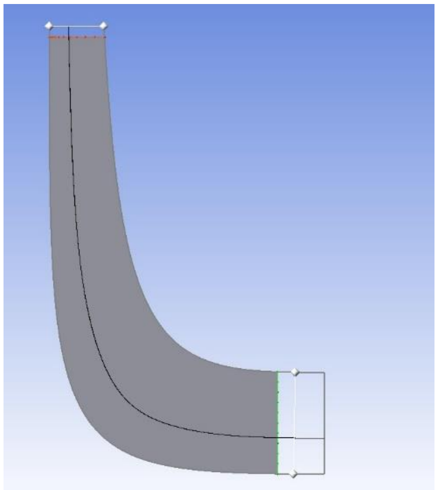


Figure 14: Mean Blade Profile

Blade Profile

Then the meshing of Impeller and Volute was done in Turbo-Grid in parallel. For which following data was used:

Mesh Data

Note that the meshing we did was not uniform because at inlet and outlet it had to be finer so the mesh element at these areas were smaller in size and hence larger in number. Below is the figure which shows how refinement of the mesh was done to get more accurate results:

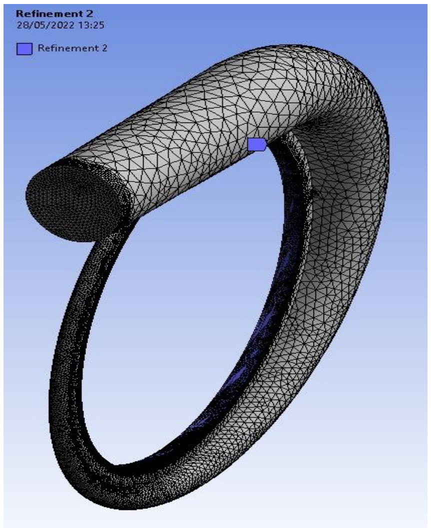


Figure 15: Meshed Volute

Also, at interfaces and at some critical points the element size was reduced again to get accurate results. The meshing is not equal instead it is program driven. The mesh element size 10^ {-3}.

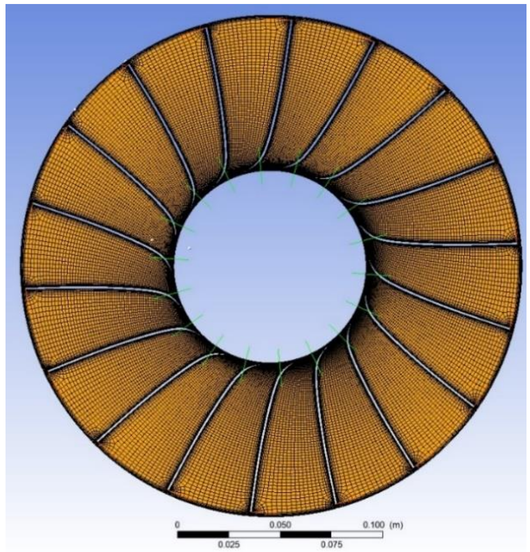


Figure 16: Meshed Impeller for CFD

After successful meshing, the inlet and outlet passages were created to define the path for the flow. The inlet path is the inlet of the impeller from where the flow passes through the blades and after gaining the velocity from the impeller, it enters the volute where it further gains some velocity as per geometry of the volute. In the figure below, the inlet is represented by the green area having small arrows while outlet is represented by the larger arrows at the end of the volute:

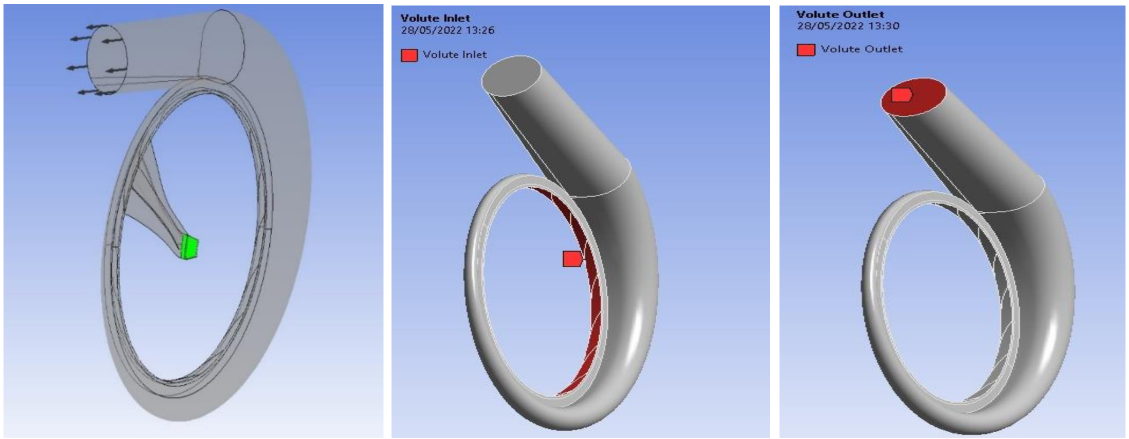
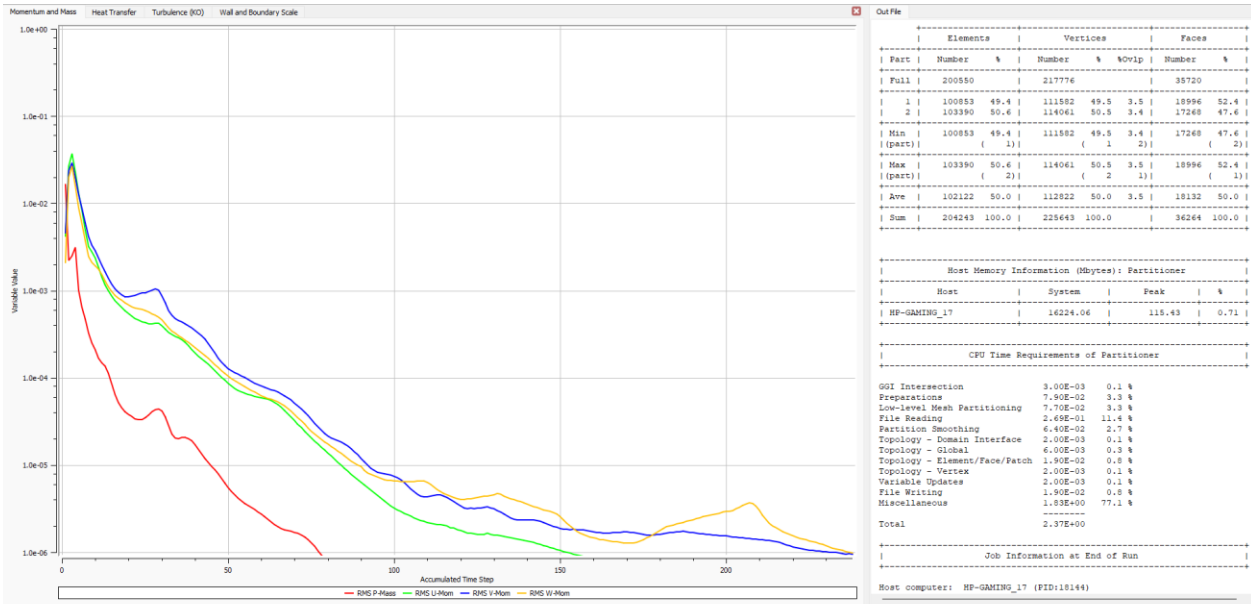


Figure 17: Boundary Conditions

Then we used CFX setup. In the CFX module of ANSYS, we need to set up the inlet and outlet conditions as well as specifying the rotations per minute. Other physics parameters include the fluid properties that must be specified before we can solve the model. In turbo mode, we specify that the component for which the analysis is being performed is a centrifugal compressor. The rotational axis is specified, and then steady state analysis type is opted. The R1 component is specified to be rotating at an angular frequency of 20,000 RPM, interacting with air at $25^{o}C$. Total energy heat transfer is specified for the model with shear stress transport turbulence model instead of k-epsilon method. The shear stress transport model is a combination of the k omega and epsilon modeling strategies. The eddy viscosity in this model is restricted, allowing for enhanced performance in wake zones. In the totally turbulent zone, we use the k-model, and in the near wall region, we use the k-model. The relevant bounds are then double-checked to make sure they don't go too far or are specified incorrectly. The total inlet pressure and static outlet pressure model is selected to specify the inlet and outlet pressures with the flow normal to the boundary. The study was done for frozen rotor as per instructed by our supervisor. The passages for rotor and stator we specified for study. After the study was setup in CFX, we specify the solver settings with 1000 iterations, with residual type set as RMS with target of 1×10^{-4}.

A high-resolution advection scheme is selected with high resolution turbulence numeric. In the convergence control settings, we specify a maximum of three hundred iterations. Auto time scale control with Conservation length scale is selected to control the fluid timescale with a unitary timescale factor. RMS Residual type convergence criterion is set with a residual target of 10^{-6}. In order to create the run definition, we select a double precision run type with direct start submission type and Intel MPI Local Parallel Run Mode in the Parallel environment. The solver is set to run till convergence which is achieved after 238 iterations.



Structural Analysis:

Structural analysis is an integral part of any mechanical system. Only a structural integrate system can survive all the required harsh environments and unprecedented loads that may occur in the future. The structural analysis is performed using Finite Element Analysis (FEA). FEA progress and calculates the stresses and strains by dividing the whole geometry into smaller elements and then calculating the results on individual elements and nodes. This results into a clear body with all the results merged. The development of modern tools has pushed the applicable conditions to the fullest by providing real environment coherence of the results. The results can be considered verified for the sake of implementation and while considering a good safety factor.

The major determinants of a good quality Finite Element Analysis it the use of effective boundary conditions. The boundary conditions respond to how the external factors will be acting on the compressor body in the environment. The second most important condition in effective analysis is the mesh size. Mesh size corresponds to the dimensions of the elements used for the solution. The smaller the mesh, more accurate will be the solution and vice versa.

The last deliverable before the manufacturing of the centrifugal compressor, either prototype or the commercial model, applies the FEA to test the structural integrity of the compressor body. For the prototype, we have analyzed the shaft and the impeller as these two components are the major moving parts of the compressor and therefore most of the analysis are due to these only. The compressor is analyzed with structural steel being applied as the material for all the analysis. This assignment of material is based on the final application of the compressor as required.

Analysis of Impeller:

Impeller integrity is the major element of this design. Our approach has been to identify the constraints that are acting on the impeller and then simulate those in our used tool Ansys Mechanical to get the results. Following table better illustrates the use of boundary conditions on the impeller.

Table 2: Inputs for FEA of impeller

Rotational Velocity	20000 rpm or 2148.4 rad/s	Applied on the impeller hub inlet where the shaft resides.
Material	Structural Steel	
Gravitational Load	With respect to mass	Applied directly to include the gravitational effects
Mesh Size	5 mm	The mesh size is adjusted based on available computational power and required results.
Frictionless Support		Applied on the inlet of hub to simulate a rotation.
Evaluated Parameters	• Total Deformation • Equivalent Stress • Strain Energy • Maximum and	

	Minimum Principal Stresses	

Meshed Body:

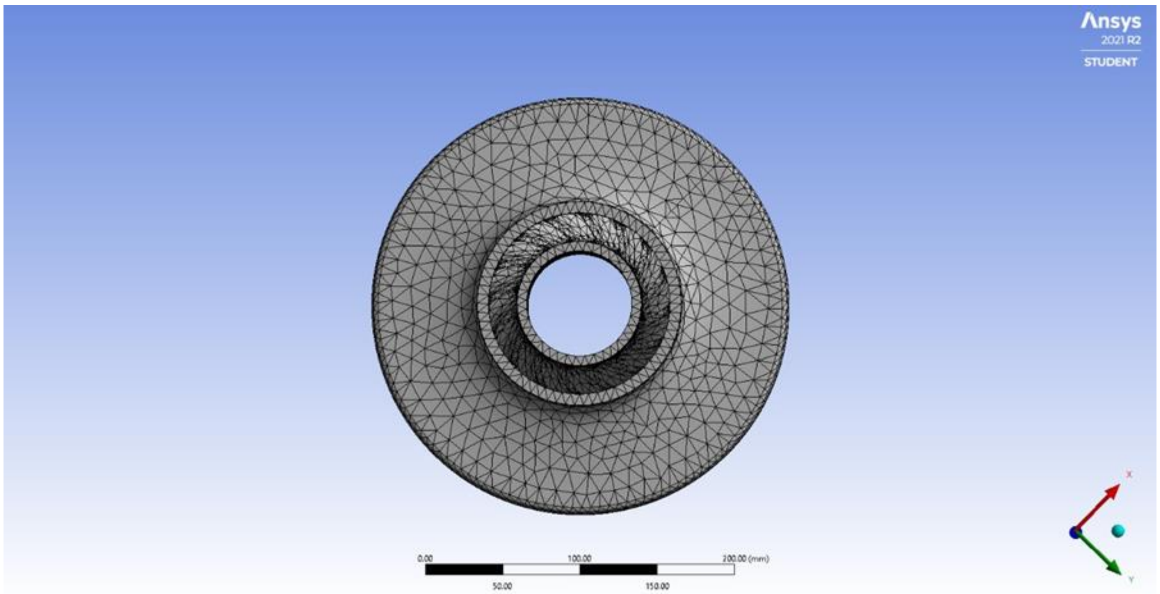


Figure 18: Meshed Body of Impeller

Structural Analysis of the Shaft:

For our prototype, a structural steel shaft is analyzed, and various parameters are adjusted based on the requirements as will be practiced in the real time environment of our prototype testing. The shaft is the main structural element that is responsible for all the operation and working of centrifugal compressor. Therefore, its design is an important feature in the overall design and its structural integrity eventually ensures a long life for compressor. The original model of the shaft used for analysis is given below:

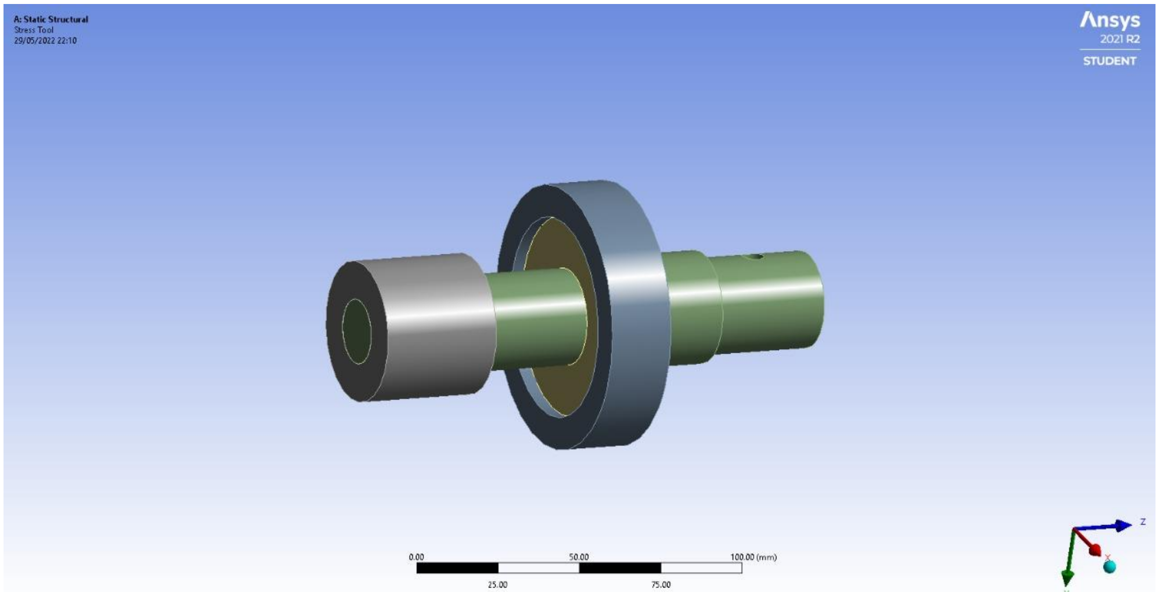


Figure 19: Shaft Model

The important features of the FEA of shaft are given below:

Table 3: Inputs for FEA of Shaft

Material	Structural Steel	
Rotational Velocity	20000 rpm or 2148.4 rad/s	
Mesh Size	3 mm	This mesh size is tested, and the optimum results can be obtained with this mesh size.
Frictionless Support	On the motor shaft side	This is applied to ensure a

		smooth operation of the shaft.
Fixed Joint	Applied between the bush and the shaft	This bush is designed to save impeller in case of shaft problems.
Bearing	Applied on the shaft body.	The bearing is inserted on the backplate of the casing of the compressor.
Studied Parameters	• Total Deformation • Maximum and Minimum Principal Stress • Strain Energy • Equivalent Stress	

Meshed Body:

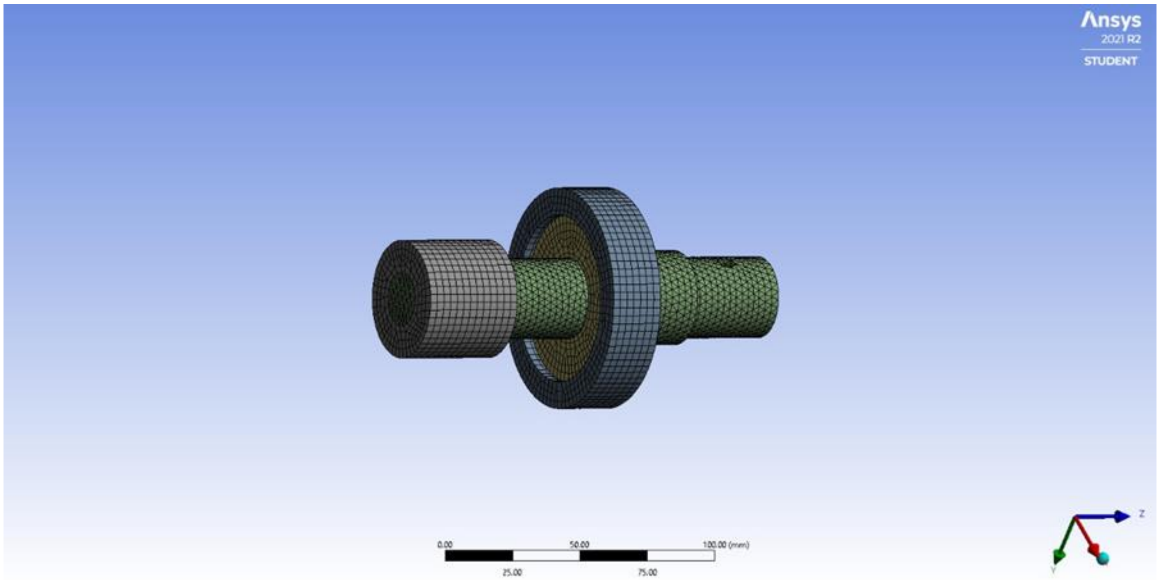


Figure 20: Meshed Body of Shaft

CHAPTER 4: RESULTS AND DISCUSSIONS

Preliminary Calculation Results:

The results follow our proposed methodology, and it features the results from preliminary calculations to the analysis in CFD. The results of the preliminary calculations are obtained for the following parameters, only major parameters are listed here.

Table 4: Preliminary Calculation Results

$Parameter$	$Value$
$Specific\ Work$	$44690\ m2/s2$
U_2	
$Outer\ diameter$	$214\ mm$
$Relative\ Mach\ Number\ at\ inlet$	0.7
$Relative\ angle\ at\ inlet$	$57.94\ degrees$
$Hub\ Radius$	$24\ mm$
$Shroud\ Radius$	$192\ mm$
$Slip\ Factor$	0.89004
$Angular\ velocity\ of\ impeller$	$2094.39510\ rad/s$

Absolute Mach number at inlet	0.37163
Inlet density from stagnation density	1.10621 kg/m^3
Flow Coefficient	0.56117
Work Coefficient/Power Input Factor	0.89004
Specific Speed	0.81750
Radial Outlet velocity	125.74490 m/s
Relative Outlet Velocity	199.43941 m/s
Outlet velocity	235.77078 m/s
Absolute Flow angle	57.76891 *degrees*
Isentropic efficiency of impeller	092%
Outlet stagnation temperature	342.46766 K
Outlet stagnation pressure	158948.39466 Pa
Actual outlet temperature	314.81201 K
Actual Outlet pressure	118376.95967 Pa
Outlet density	1.30960 kg/m^3
Blade Width	0.00903 m
Outlet absolute Mach Number	0.66277

$Radius\ for\ vaneless\ space\ r\_2d$	$0.11769\ m$
$Tangential\ velocity\ at\ radius\ r2d$	$181.30855\ m/s$
$Radial\ Velocity\ at\ radius\ r2d$	$114.31355\ m/s$
$Flow\ angle\ at\ radius\ r\_2d$	$50.91359\ degrees$
$Velocity\ at\ radius\ r\_2d$	$214.33707\ m/s$
$Temperature\ at\ r\_2d$	$319.61175\ K$

These results clearly indicate that our objectives till now have been met. The process will be verified in the CFD analysis to make sure that our analysis holds empirical value. The results of this step are utilized in the CAD modeling of our centrifugal compressor.

CFD Analysis Results:

Following the development of solution, we employ CFD-Post to create a graphical representation of the findings of the solver. CFD Post helps to plot the blade to blade and meridional plots so that we can better understand the interaction of the fluid. We switch to the turbo tab and select color plot type for the blade to blade and meridional plots of pressure, temperature, velocity etc. in the local R1 domain and with a span of 0.5. The results are attached as follows:

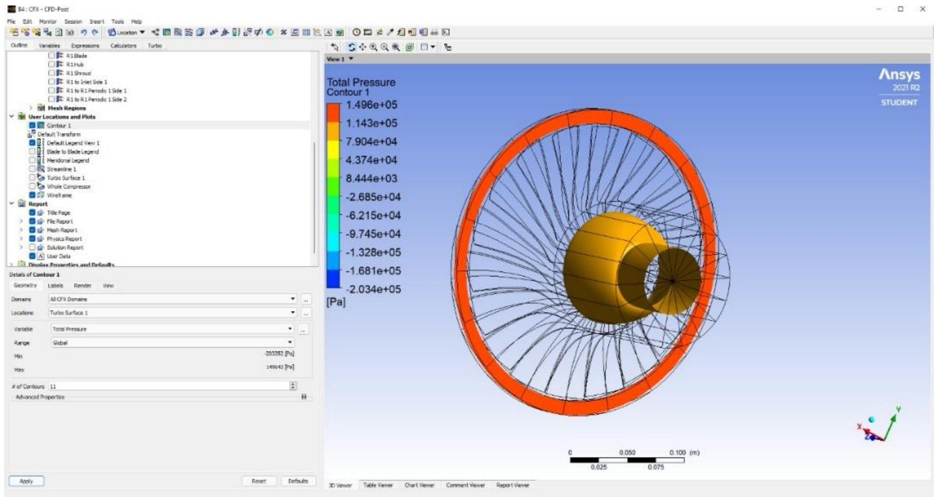


Figure 21: Pressure Distribution on Blade Profile

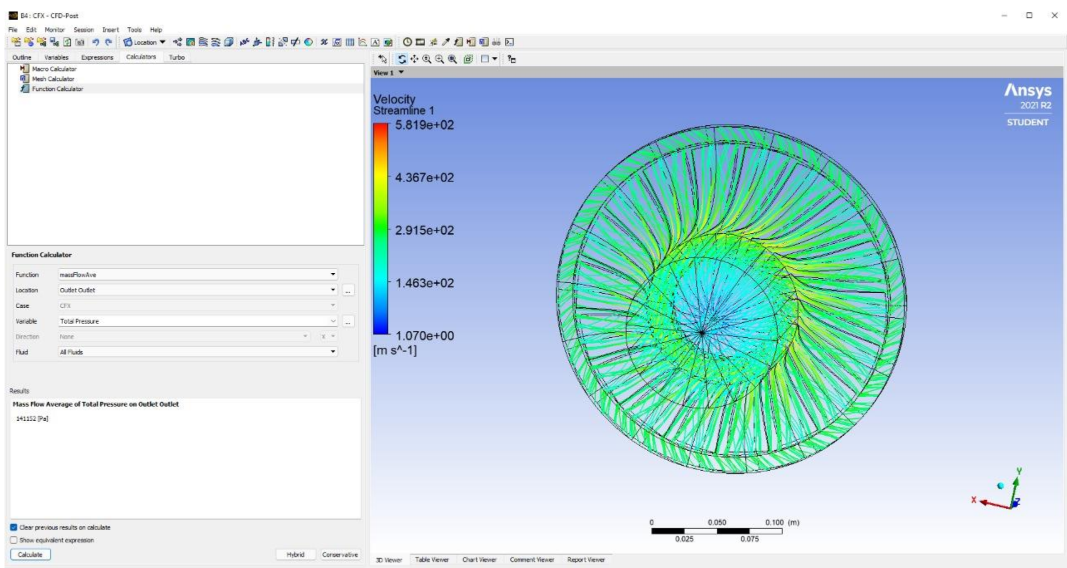


Figure 22: Velocity Distribution on Blade Profile

It is evident from the results shown above that the pressure and velocity values that were expected and desired as a design criterion are obtained after the CFD analysis with minimal errors. The errors can be attributed to the losses that occur within the compressor. Pressure is the first and most important metric. We see that a pressure ratio of 1.286 is achieved which is within the desired range of 1.2 to 1.5, resulting in an output pressure of 114kPa. The next metric to be considered is the fluid velocity at the outflow. We see that an average velocity of 300 ms^{-1} is obtained which is also above the acceptable level of 250 ms^{-1}.

Structural Results:

Results of Impeller FEA:

A suitable simulation from FEA has proved structural integrity of the impeller. The structural steel has about tensile strength of around 350 to 450 MPa. The proposed impeller model at 20000 rpm can sustain the loads easily as the FEA results are purely in range. The other results of FEA are denoted below:

Table 5: Result Parameters of Impeller FEA

Total Deformation Minimum	$9.343\mathrm{E}-06$	mm
Total Deformation Maximum	0.00042589	mm
Total Deformation Average	0.00023936	mm
Equivalent Stress Minimum	$1.1371\mathrm{E}-05$	MPa
Equivalent Stress Maximum	11.195	MPa
Equivalent Stress Average	1.0497	MPa

Strain Energy Minimum	$1.1512\ \mathrm{F}-12$	mJ
Strain Energy Maximum	0.0041972	mJ
Strain Energy Average	2.73	mJ
Safety Factor Minimum	15	
Shear Stress Minimum	-3.7093	MPa
Shear Stress Maximum	3.7511	MPa
Shear Stress Average	0.0020169	MPa
Maximum Principal Stress Minimum	-1.0293	MPa
Maximum Principal Stress Maximum	15.488	MPa
Maximum Principal Stress Average	1.1085	MPa

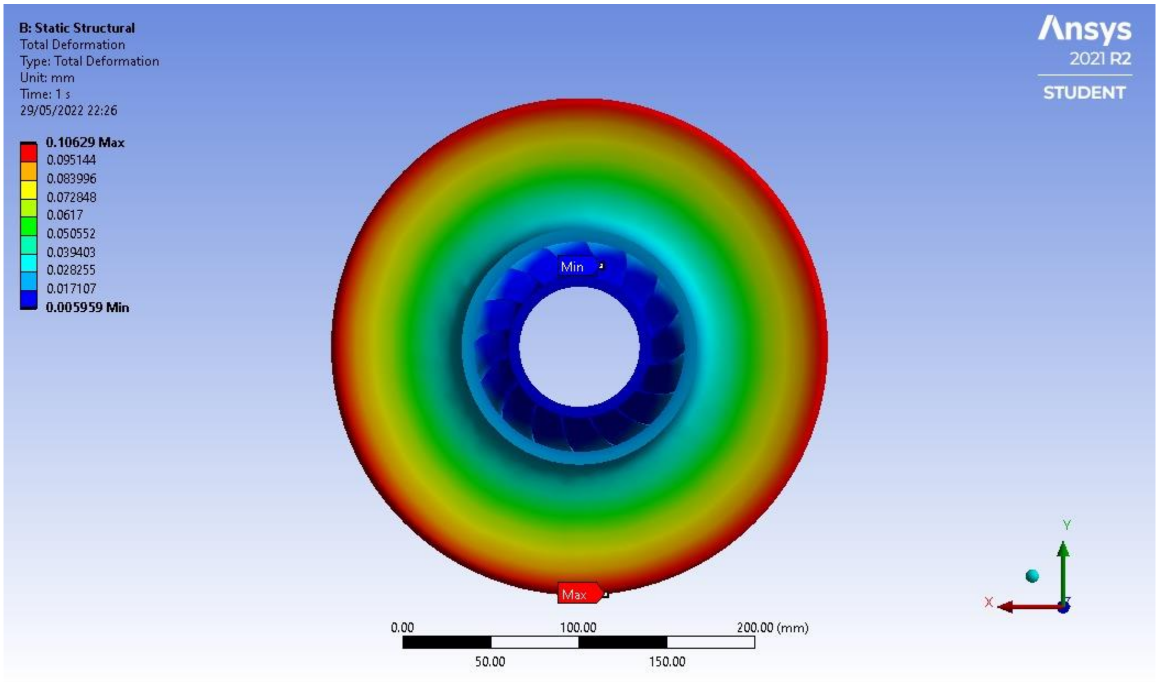


Figure 23: Total Deformation of Impeller

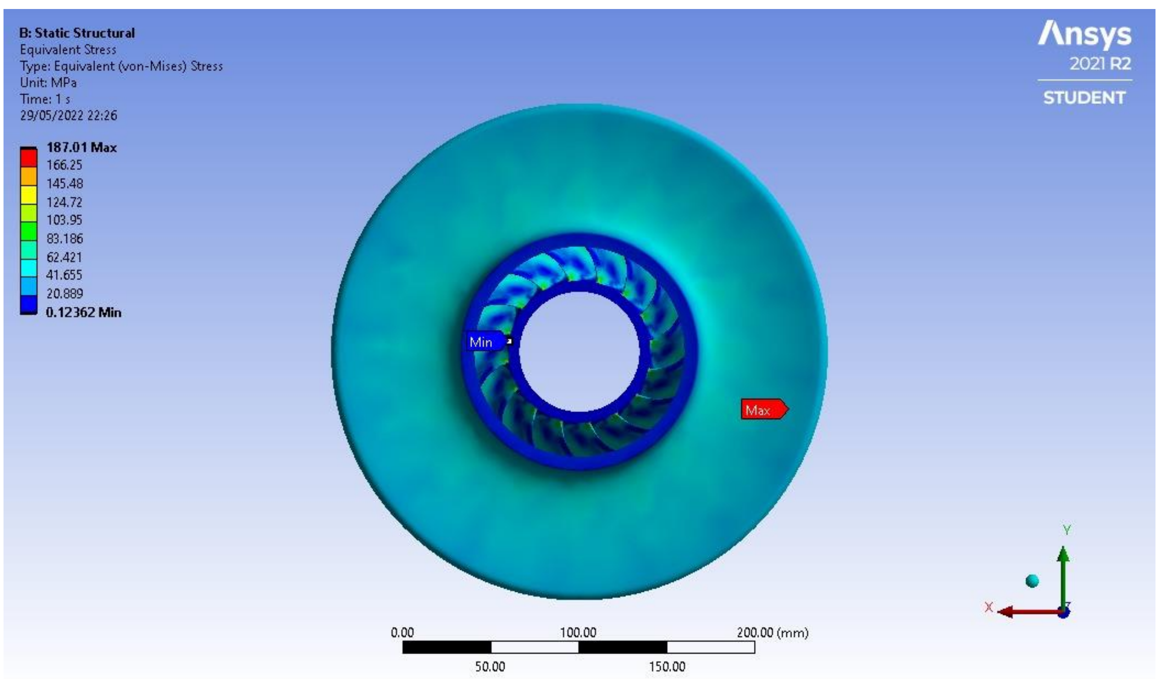


Figure 24: Equivalent Stress on Impeller

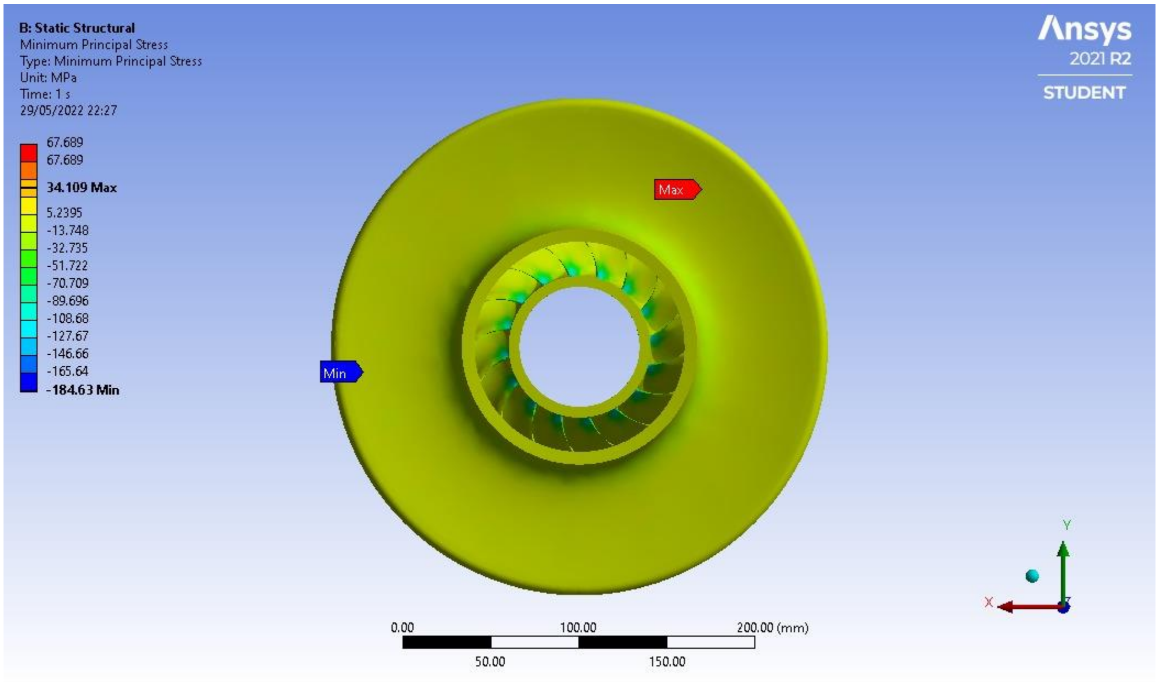


Figure 25: Minimum Principal Stress

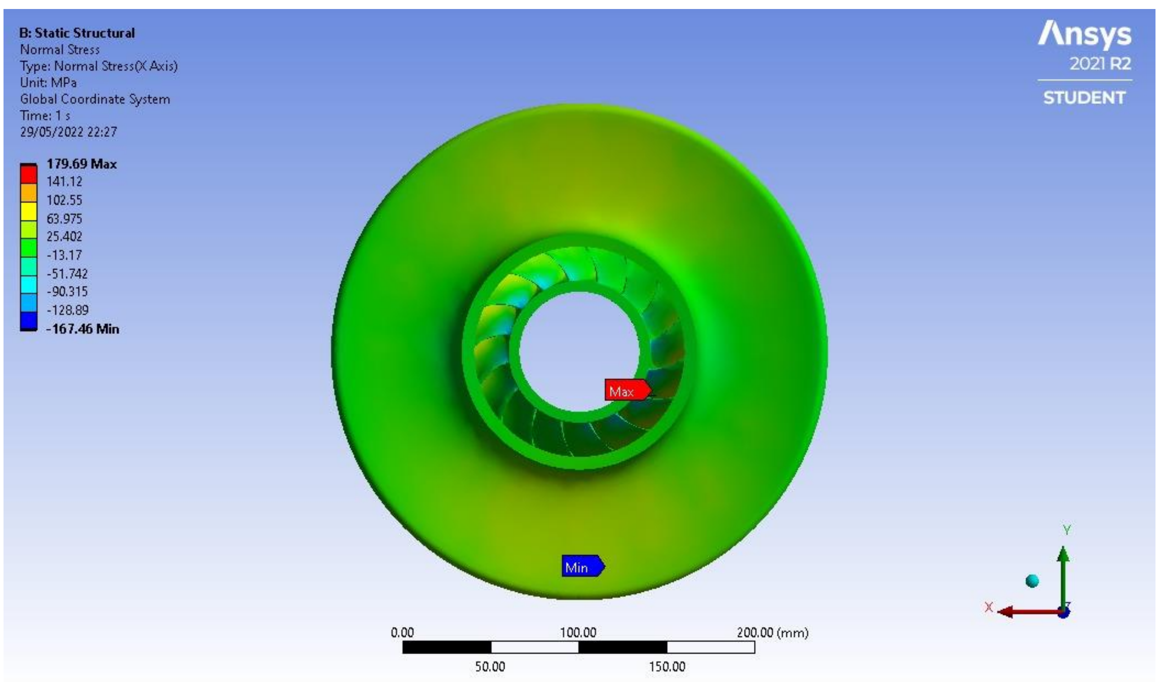


Figure 26: Normal Stress in X-axis

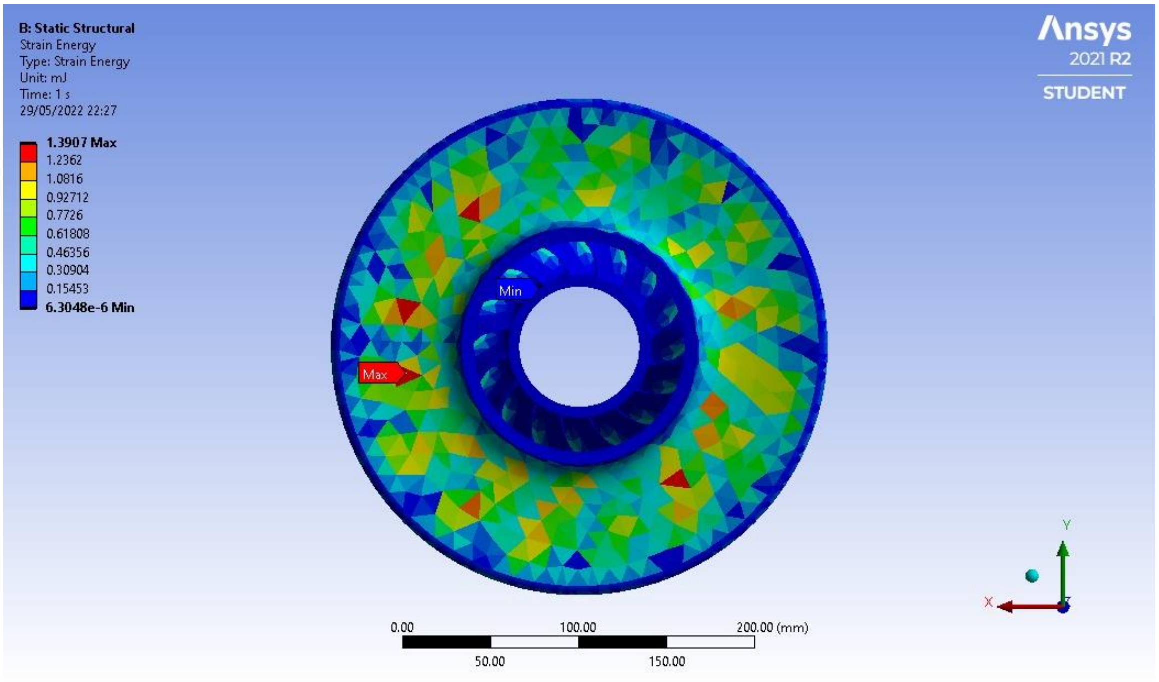


Figure 27: Strain Energy

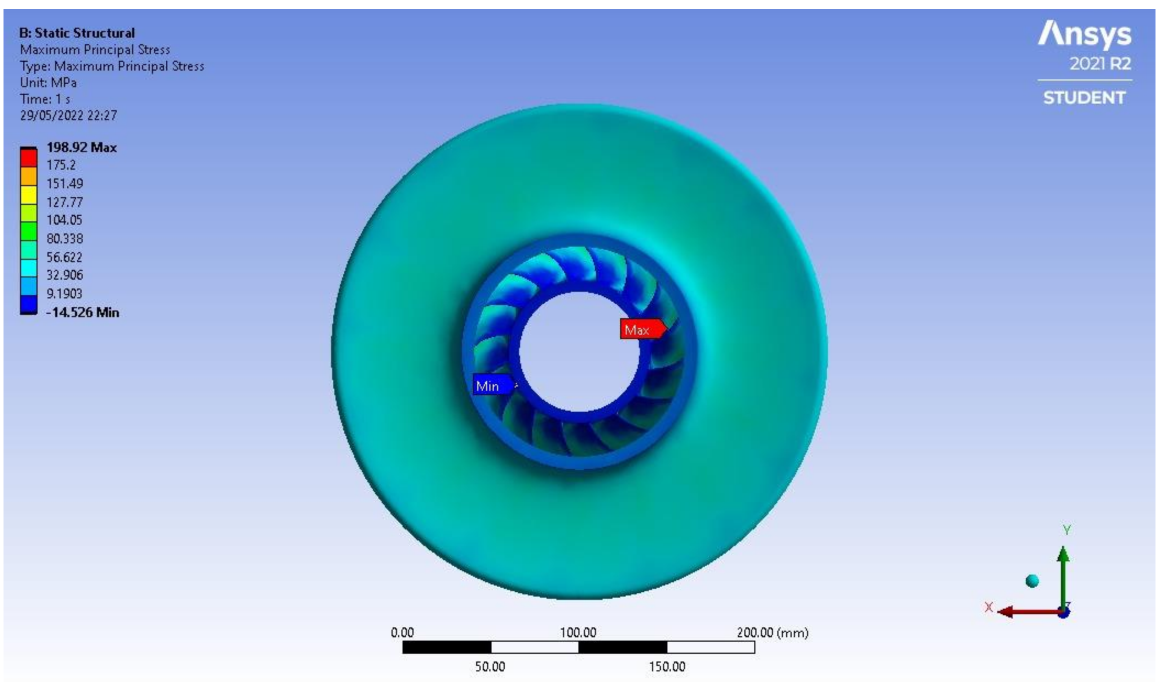


Figure 28: Maximum Principal Stress

Results of Shaft FEA:

Or shaft has sustained under the given FEWA results. Even the shaft is stronger than the impeller as the values of stresses given below are much less than which further proves the integrity of the shaft. The tabular results of shaft are given below:

Table 6: Result Parameters of Shaft FEA

Minimum Principal Stress Minimum	-184.63	MPa
Minimum Principal Stress Maximum	34.109	MPa
Minimum Principal Stress Average	-7.1493	MPa
Equivalent Stress Maximum	187.01	MPa
Equivalent Stress Minimum	0.12362	MPa
Equivalent Stress Average	44.04	MPa
Total Deformation Average	0.033169	mm
Total Deformation Maximum	0.10629	mm
Total Deformation Minimum	0.005959	mm

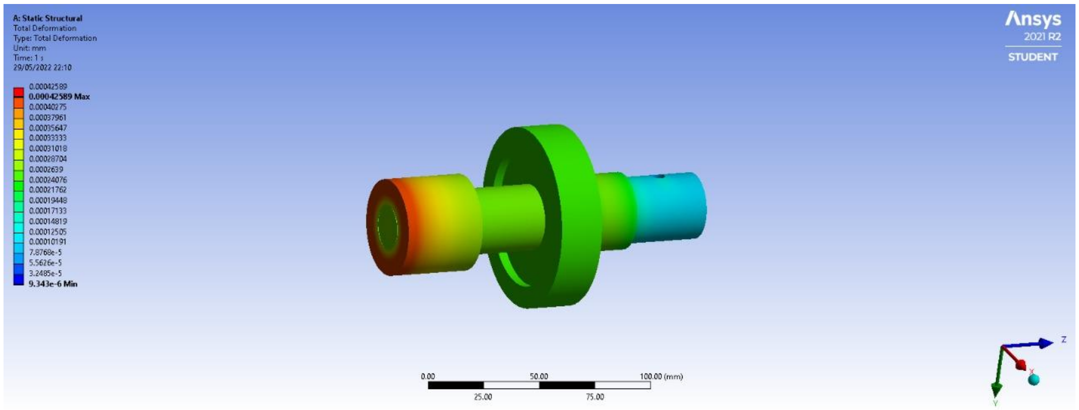


Figure 29: Total Deformation

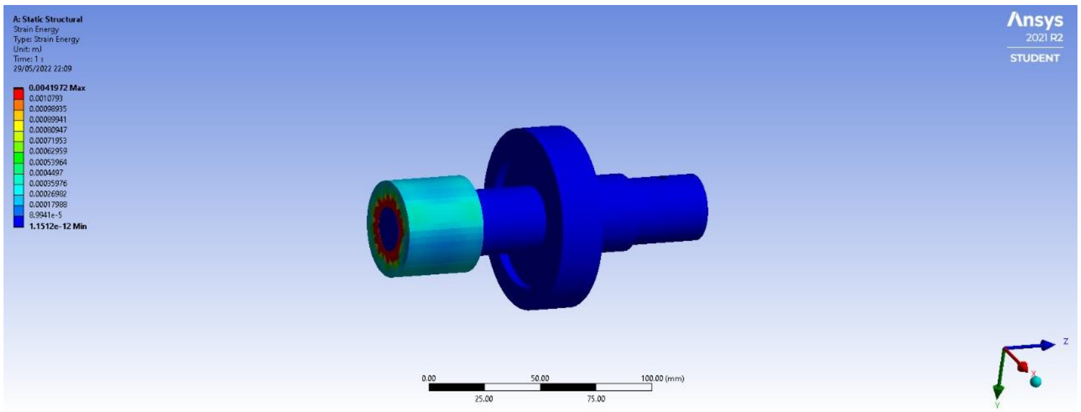


Figure 30: Strain Energy

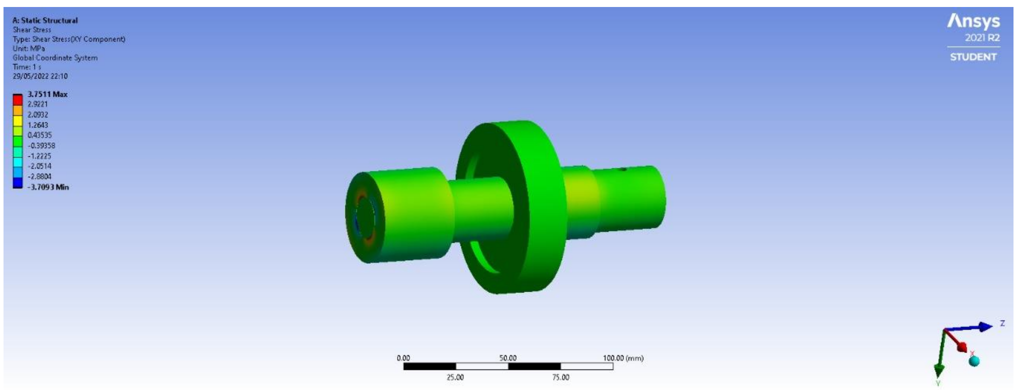


Figure 31: Shear Stress on Shaft

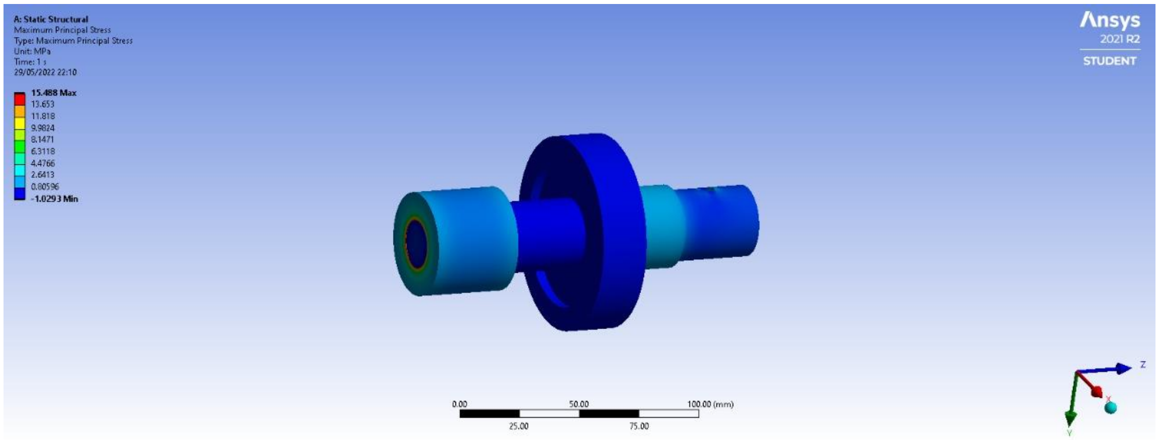


Figure 32: Maximum Principal Stress on Shaft

Comparison with other works:

Centrifugal compressor design has been a fundamental part of most engines. The work has been done in great deal to design the most efficient centrifugal compressor. A lot of research has been done as well. In the design of volute, Xu and Muller approached numerically and verified through experimental data. Our approach of design is similar except that we have verified the volute through CFTurbo numerical calculations and then verified through computational fluid dynamics (Xu and Müller, 2005). Meroni and others similarly followed a mean line design approach as we have discussed in our theoretical part (Meroni, Zühlsdorf, Elmegaard and Haglind, 2018).

A very identical design to our approach is the design done by Eckardt who worked on impellers with tip speeds of above 400 m/s. The author further investigated the turbulence effect as well on the impeller. The similarity with our approach lies in the high velocity result attainment as in our case the tip velocity was nearly 373 m/s (Eckardt, 1976).

Prototyping

After successfully designing the 3D Model, completing its CFD Analysis and then FEM Analysis which were the required deliverables, our next task was to give our project a physical shape and show its working. The designed components in this project re 3D printed because this was the only feasible option for us that is why it won't be able to show the original working of the compressor and hence the required outputs will not be achieved, and this prototype is the scaled model only to show the working of our project.

Impeller:

The Impeller originally modeled on CF-Turbo was solidified on Solidworks and then it was 3D Printed using ABS material.

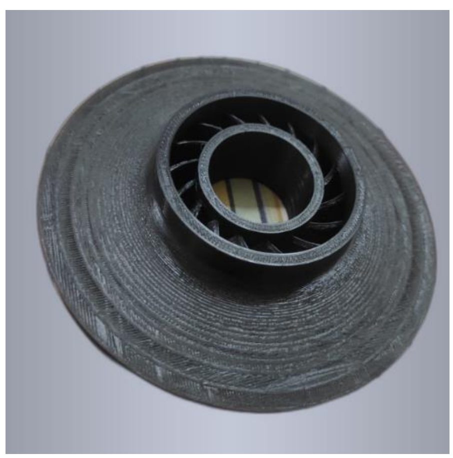


Figure 33: Physical Impeller

Volute Casing:

The Volute casing again whose volute was designed in CF-Turbo and then given the shape of casing on Solidworks was 3D printed using ABS material.

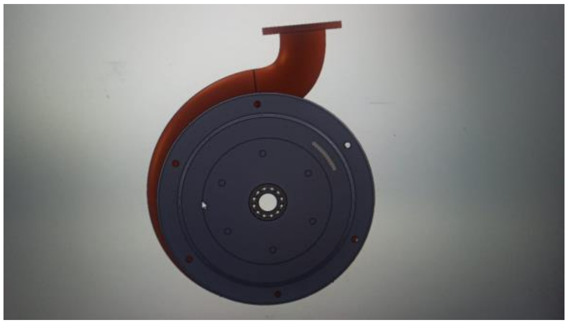



Figure 34: Physical Volute Casing

Motor:

The next important mechanical component which basically drives the impeller inside the casing is motor. It was a challenge for us to select a motor that could do the job for our project as the rpm we required was 20000, but in market this kind of motor was not available and the industry with which we were doing the project also said that the motor will be available after some time. So, we went for a 2800 rpm motor whose model is shown below:

Figure 35: Motor

Shaft:

The motor drives the impeller using shaft. The drawing of the shaft we used is shown below. We used the material aluminum for it, and we designed it will variables diameters to make it as light weight as possible and not to reduce its strength at the same time.

Figure 36: Aluminum Shaft

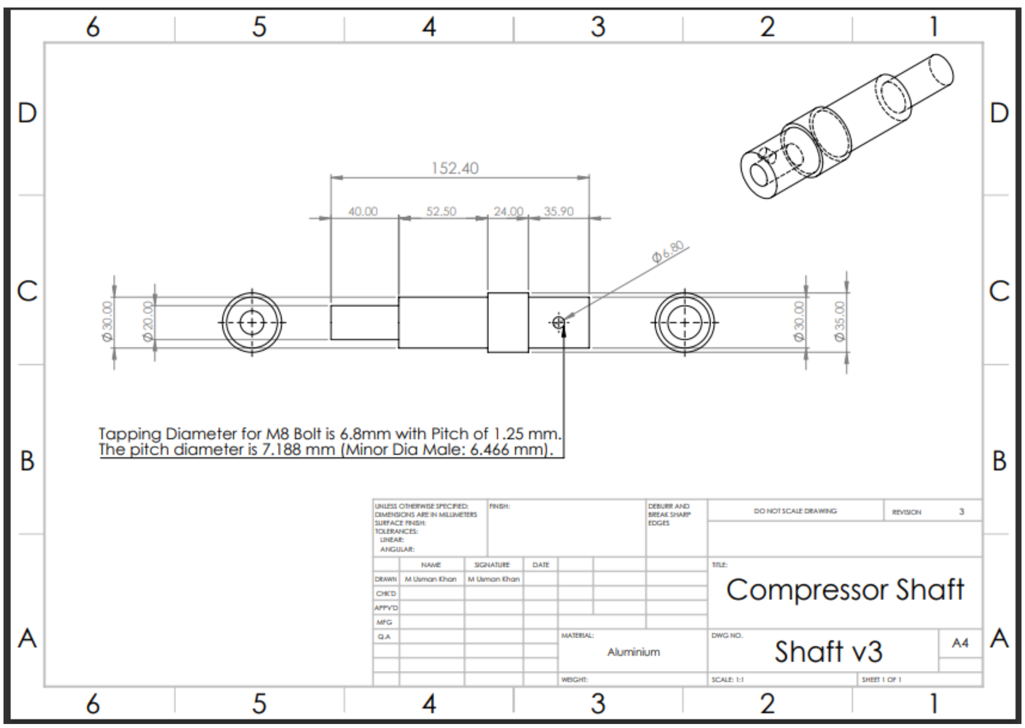


Figure 37: Shaft Drawing

Bearing:

The bearing is basically used to physically support the rotating shaft which passes through the back plate to go into the casing. It also makes the motion smooth by minimizing the friction.

The bearing we used is 6202 2RS with internal diameter of 30 mm and outer diameter of 62mm:

Figure 38: Bearing 6206

Nylon Bush:

Nylon is a material excellent strength and abrasion resistance. So, we used this material to use as an interface between Impeller and Aluminum Shaft and between Backplate Inner Hole and Bearing Outer Wall.

Its drawings are shown below:

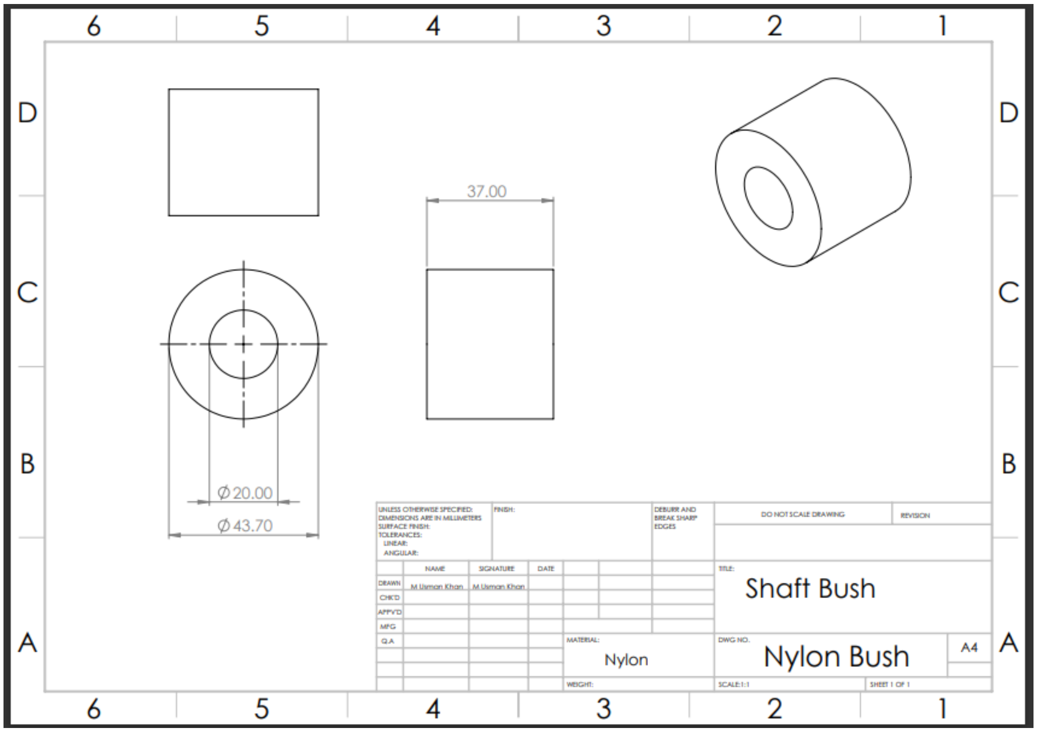


Figure 39: Bush Drawing

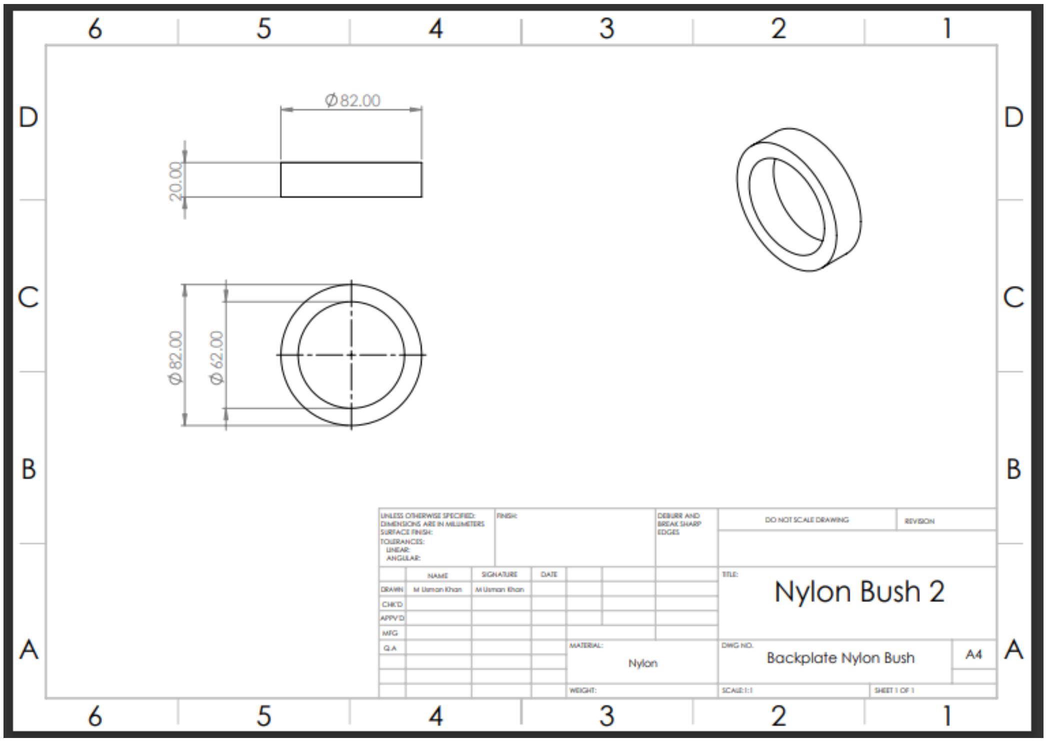


Figure 40: Backplate Bush Drawing

Metallic Frame:

Next comes metallic frame which basically is used to make a proper platform for our components on which they can be assembled to each other ensuring the alignment of the components which is very crucial for our compressor to work properly.

We used 30mm Mild Steel Rectangular Pipes for that along with 4.5 mm thick Mild Steel sheet for the solid platform. The drawings of which are shown:

Figure 41: Frame of Assembly

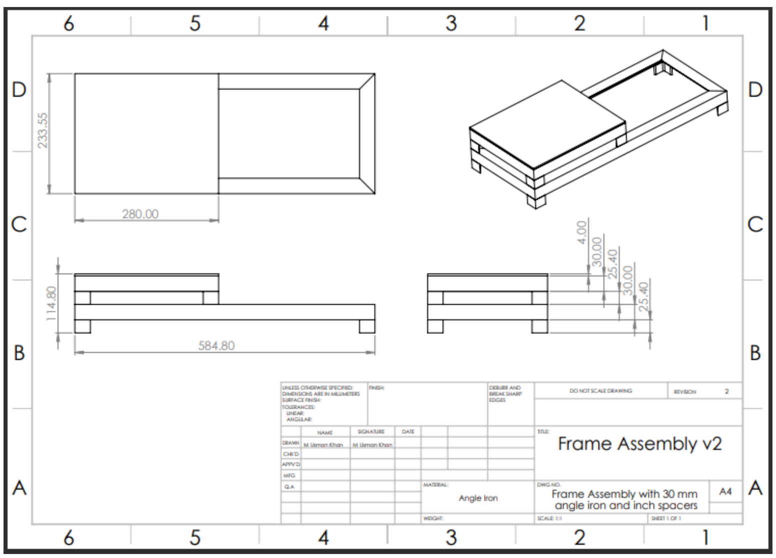


Figure 42: Drawing for frame of Assembly

Mounting Plates for Casing:

We needed to mount the volute casing with the platform to ensure some clearance between the surface of the frame and the casing so that no vibrations are induced in the frame.

The same Mild Steel sheet of 4.5 mm thickness was used to make these plates. The drawings are as shown:

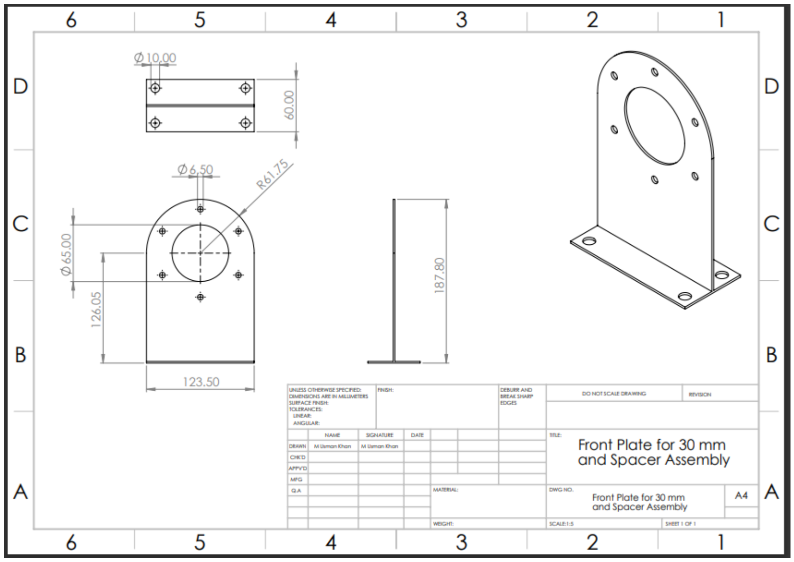


Figure 43: Volute Front Plate Drawing

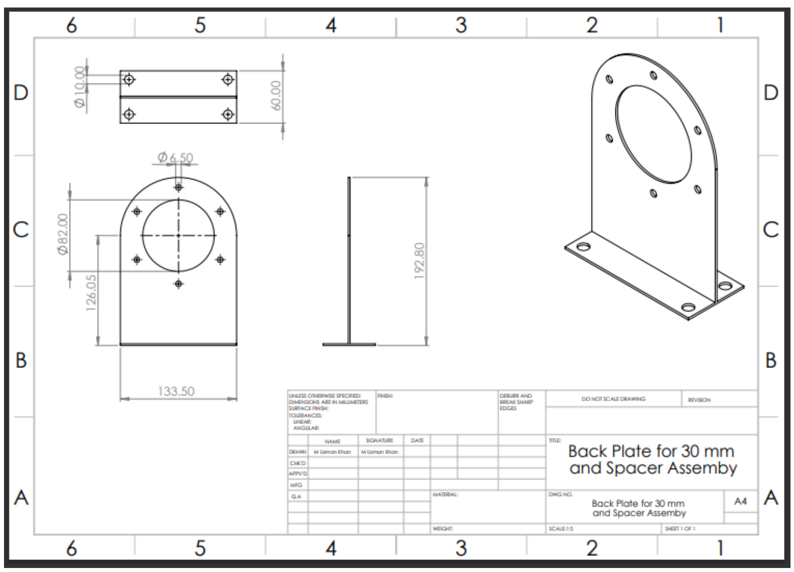


Figure 44: Volute Backplate Drawing

Final Assembly:

After designing and manufacturing all the components, our next task was to make a completely assembled working model. It was assembled as shown:

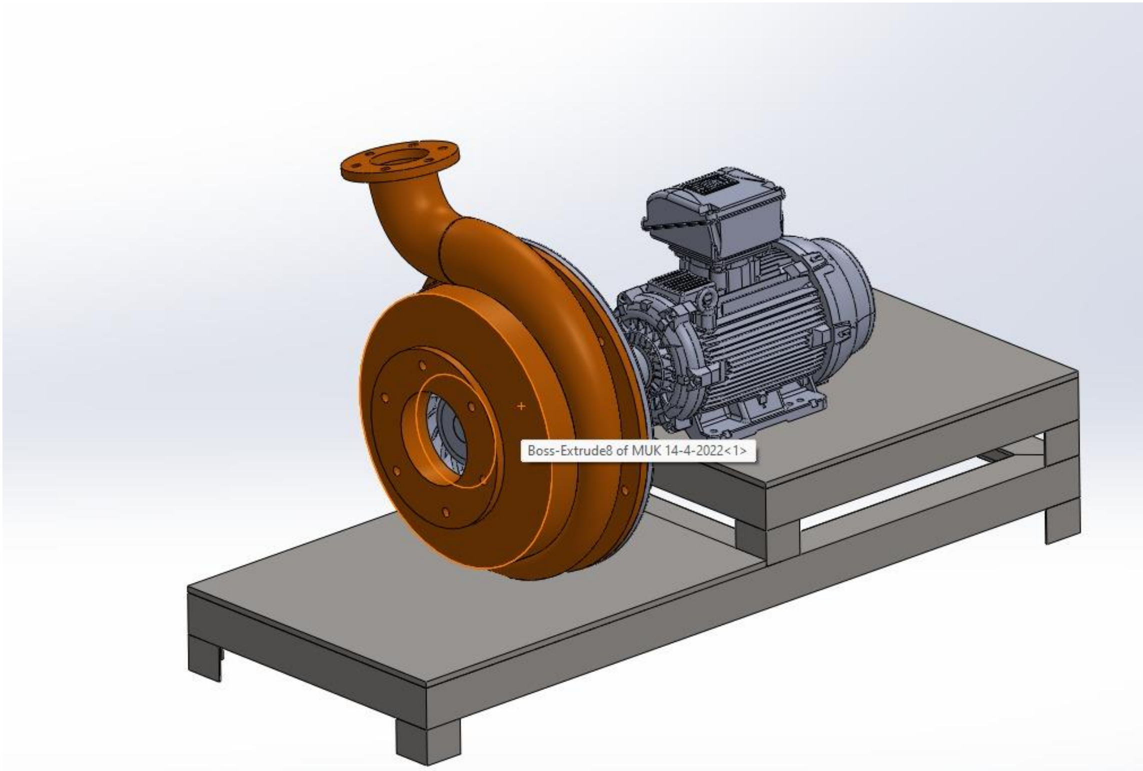


Figure 45: Final Assembly Model

CHAPTER 5: CONCLUSION AND RECOMMENDATION

Conclusion

The purpose of our project is to design a high-performance centrifugal compressor bases on the specified given parameters and required output. A centrifugal compressor is generally for low to medium pressure ratio and high-volume flow rate. And the purpose of our centrifugal compressor is not to actually provide the compression but high exhaust velocity at very low-pressure ratio of 1-2-1.5. So, it is a big challenge for us to design such a compressor.

Now talking about what is our approach to design the centrifugal compressor, we first studied in detail the design procedure of a typical centrifugal compressor and then started working on the one-dimensional preliminary calculations. So, we did the required calculations to get the necessary geometric and design parameters. For this we used the one-dimensional flow approach.

After successfully performing the preliminary calculations to get the required geometric and design parameters, our next task was to design its preliminary 3D model that could be analyzed later and optimized and then can be used for manufacturing in the 3D printing manufacturing process. So, there are many 3D modelling or CAD software are available, but the best choice for us was to go for CF Turbo. This is because it is specifically designed for the modelling of turbomachinery including centrifugal compressor because using this tool, we do not have to manually draw any component of centrifugal compressor but instead we only had to input some specific geometric and design parameters and it automatically generates the model fulfilling the requirements. So, using this we were able to design a preliminary model successfully which could be easily used for CFD and finite element or structural analysis. The procedure of this is shown in detail in methodology chapter.

After designing the preliminary 3D model our next task was to check its performance for which CFD analysis was needed to be performed. Though we got import the model from CF Turbo easily, but we decided to design the impeller in the Ansys using Vista CCD because this procedure is widely followed to validate the design and calculations and also check the accuracy of the calculations previously performed.

So, using Vista CCD we basically used the input data to get all the other design and geometric parameters which were matching with a little difference to our one-dimensional calculations. After getting all the parameters we optimized the profile of the blades on the impeller using the translations of the control point of Bezier curves representing the meridional profile. After optimizing the profile of the blade and hence design of the impeller our tasks wo to do the meshing for flow or CFD analysis whose procedure is written in detail in methodology. After doing the analysis and getting the CFD results, we compared our CFD results with the results we got through calculations which show that our results were very much accurate because the difference that occurred was only due to the flow losses, we did not cater for in our one dimensional or software calculations. For example, our targeted efficiency is 85% and the efficiency we achieved so far is 74.5% which will be increases after optimizing the design.

So, summing up all of this, we were successfully able to design and analyze the flow and performance of our preliminary model and got satisfactory results but obviously we need to improve and optimize our design for which we will use iterative method that is it will be optimized gradually and not once.

Recommendations:

Since the manual calculations performed by the famous authors are one-dimensional calculation that is the flow is only considered in one dimension so it does not produce accurate results when compare to the results obtained from the software so this caused much problem as the accuracy of some parameters is very critical for the optimal design of the centrifugal compressor so it is recommended that the manual calculations should not be one dimensional to get more accurate results.

It is further recommended to improve the design by practicing various designs and blade angles to optimize the overall design of the compressor. A turbulence focused designed should be practiced in the future to more closely predict the environment of the working of compressor. Moreover, our aim is to further deep dive into comparison of various blade angle differentiated impellers.

REFRENCES

1. *Rodgers, C. (1980). Efficiency of centrifugal compressor impellers. AGARD Conference Proceedings,No. 282. Centrifugal Compressors, Flow Phenomena and Performance Conference. Buressels. Retrieved Jan 1, 2022*

2. *Thorat, S. (2022). Centrifugal Compressor – Diagam, Parts, Working, Efficiency, Advantages. Retrieved Jan 1, 2022, from Learnmech.com: https://learnmech.com/centrifugal-compressor-diagram-parts-working-advantages/*
3. *Britannica, T. Editors of Encyclopaedia (2019, March 20). compressor. Encyclopedia Britannica. https://www.britannica.com/technology/compressor*

4. *G.F. Hundy, ... T.C. Welch, in Refrigeration, Air Conditioning and Heat Pumps (Fifth Edition), 2016*
5. *Bonaiuti , D., Arnone , A., Ermini, M., and Baldassarre, L. (September 27, 2006). "Analysis and Optimization of Transonic Centrifugal Compressor Impellers Using the Design of Experiments Technique." ASME. J. Turbomach. October 2006; 128(4): 786–797. https://doi.org/10.1115/1.1579507*

6. *Yuri Biba, Peter Menegay, "Inverse Design of Centrifugal Compressor Stages Using a Meanline Approach", International Journal of Rotating Machinery, vol. 10, Article ID 910123, 10 pages, 2004. https://doi.org/10.1155/S1023621X04000089*

7. *Eckardt, D., 1976. Detailed Flow Investigations Within a High-Speed Centrifugal Compressor Impeller. Journal of Fluids Engineering, 98(3), pp.390-399.*

8. *Shu, X., Gu, C., Xiao, J. et al. Centrifugal compressor blade optimization based on uniform design and genetic algorithms. Front. Energy Power Eng. China* ***2,*** *453 (2008). https://doi.org/10.1007/s11708-008-0083-5*

9. *Turunen-Saaresti, T., and Jaatinen, A. (October 30, 2012). "Influence of the Different Design Parameters to the Centrifugal Compressor Tip Clearance Loss." ASME. J. Turbomach. January 2013; 135(1): 011017. https://doi.org/10.1115/1.4006388*

10. *Hazby H. R. and Xu L., Effect of Leading Edge Sweep on Performance of a Transonic Centrifugal Impeller, Proceedings of the International Symposium on Air Breathing Engines (ISABE), ISABE Paper-1225. 2007*

11. *Roytta, Pekka & Grönman, Aki & Jaatinen-Värri, Ahti & Turunen-Saaresti, Teemu & Backman, Jari. (2009). Effects of Different Blade Angle Distributions on Centrifugal Compressor Performance.*

International Journal of Rotating Machinery. 2009. 10.155/2009/537802

12. *Hazby, Hamid & Xu, Liping. (2007). Effect of Leading-Edge Sweep on Performance of a Transonic Centrifugal Impeller. SASTech Journal.*
13. *BONCINELLI., et al. "Impeller-diffuser interaction in centrifugal compressors: Numerical analysis of radiver test case." Journal of Propulsion & Power 23.6 (2012): 1304-1312.*
14. *Liu Yangwei. "Numerical investigation of impeller-diffuser interaction in a centrifugal compressor." Journal of Aerospace Power. 024.012 (2009): 2695-2702*
15. *Xu, Cheng, and Michael Muller. "Development and Design of a Centrifugal Compressor Volute." International Journal of Rotating Machinery 2005. 3 (2005): 190-196.*
16. *Xu, C. and R. S. Amano. "Eliminating Static Pressure Distortion by a Large Cut Back Tongue Volute." ASME Turbo*
17. *Expo 2006: Power for Land, Sea, and Air 2006*
18. *Shao, W, Wang, X, Yang, J, Liu, H, & Huang, Z. "Design Parameters Exploration for Supercritical CO2 Centrifugal Compressors Under Multiple Constraints." Proceedings of the ASME Turbo Expo 2016: Turbomachinery Technical Conference and Exposition. Volume 9: Oil and Gas Applications; Supercritical CO2 Power Cycles; Wind Energy.*

Seoul, South Korea. June 13–17, 2016. V009T36A008. ASME. https://doi.org/10.1115/GT2016-56820`

19. *Anon., 2018. The Paris Compressed-Air Power Network. [Online] Available at: http://www.douglas-self.com/MUSEUM/POWER/airnetwork/airnetwork.htm [Accessed 1 October 2021].*
20. *Baljé, o. E., 1962. A Study on Design Criteria and Matching of Turbomachines: Part B - Compressor and Pump Performance and Matching of Turbocomponents. J. Eng. Power, 84(1), pp. 103-114.*
21. *Cho, S.-Y., Ahn, K.-Y., Lee, Y.-D. & Kim, Y.-C., 2012. Optimal Design of a Centrifugal Compressor Impeller Using Evolutionary Algorithms. Mathematical Problems in Engineering, Volume 2012, p. 22.*
22. *Dixon, S., 1998. Fluid Mechanics, Thermodynamics of Turbomachinery. 4th ed. London: Oxford.*
23. *Ibaraki, S., Matsuo, T. & Kumo, H., 2003. Aerodynamics of a transonic centrifugal compressor impeller. Journal of Turbomachinery, 126(2), p. 6.*
24. *Pandy, D. R. & Brondum, D., 1996. INNOVATIVE, SMALL, HIGH-SPEED CENTRIFUGAL COMPRESSOR TECHNOLOGIES. International Compressor Engineering Conference.*
25. *Sun, H. & Lee, S., 2004. Numerical prediction of centrifugal compressor noise. Journal of Sound and Vibration, Volume 269, pp. 421-430.*

26. *Villiers, L. C. B. d., 2014. Design of a Centrifugal Compressor for Application in Micro Gas Turbines, s.l.: Stellenbosch University.*

27. *Brett Dewar, Jonna Tiainen, Ahti Jaatinen-Värri, Mike Creamer, Mariana Dotcheva, Jovana Radulovic, James M. Buick, "CFD Modelling of a Centrifugal Compressor with Experimental Validation through Radial Diffuser Static Pressure Measurement", International Journal of Rotating Machinery, vol. 2019, Article ID 7415263, 12 pages, 2019. https://doi.org/10.1155/2019/7415263*

28. *Zebiao Feng, Jianjun Wang, Yan Ma, Yizhong Ma. (2021) Integrated parameter and tolerance design based on a multivariate Gaussian process model. Engineering Optimization 53:8, pages 1349-1368.*

29. *Zebiao Feng, Jianjun Wang, Yizhong Ma, Yiliu Tu. (2021) Robust parameter design based on Gaussian process with model uncertainty. International Journal of Production Research 59:9, pages 2772-2788.*

30. *Yunxia Han, Yiliu Tu, Linhan Ouyang, Jianjun Wang, Yizhong Ma. (2020) Economic quality design under model uncertainty in micro-drilling manufacturing process. International Journal of Production Research 0:0, pages 1-19.*

31. *C. Xu, R. S. Amano, "Meridional Considerations of the Centrifugal Compressor Development", International Journal of Rotating*

Machinery, vol. 2012, Article ID 518381, 11 pages, 2012. https://doi.org/10.1155/2012/518381

32. *Omidi, M.; Liu, S.-J.; Mohtaram, S.; Lu, H.-T.; Zhang, H.-C. Improving Centrifugal Compressor Performance by Optimizing the Design of Impellers Using Genetic Algorithm and Computational Fluid Dynamics Methods. Sustainability **2019**, 11, 5409. https://doi.org/10.3390/su11119540*
33. *Satish K., VVNK, Guidotti, E, Rubino, DT, Tapinassi, L, & Prasad, S. "Accuracy of Centrifugal Compressor Stages Performance Prediction by Means of High Fidelity CFD and Validation Using Advanced Aerodynamic Probe." Proceedings of the ASME Turbo Expo 2013: Turbine Technical Conference and Exposition. Volume 6B: Turbomachinery. San Antonio, Texas, USA. June 3–7, 2013. V06BT37A040. ASME. https://doi.org/10.1115/GT2013-95618*
34. *V V Neverov et al 2017 IOP Conf. Ser.: Mater. Sci. Eng. **232** 012037*
35. *Chaowei Zhang, Xuezhi Dong, Xiyang Liu, Zhigang Sun, Shixun Wu, Qing Gao, Chunqing Tan,*
36. *A method to select loss correlations for centrifugal compressor performance prediction,*
37. *Aerospace Science and Technology,Volume 93,2019,105335,iSSN 1270-9638,https://doi.org/10.1016/j.ast.2019.10533*

38. *Hongsheng Jiang, Sujun Dong, Zheng Liu, Yue He, Fengming Ai, "Performance Prediction of the Centrifugal Compressor Based on a Limited Number of Sample Data", Mathematical Problems in Engineering, vol. 2019, Article ID 5954128, 13 pages, 2019. https://doi.org/10.1155/2019/5954128*

39. *lkin I. GUTIÉRREZ VELÁSQUEZ,Determination of a suitable set of loss models for centrifugal compressor performance prediction,Chinese Journal of Aeronautics,Volume 30, Issue 5,2017,Pages 1644-1650,ISSN 1000-9361, https://doi.org/10.1016/j.cja.2017.08.002.(https://www.sciencedirect.com/science/article/pii/S1000936117301772)*

40. *D. A. Roberts and S. C. Kacker. Numerical investigation of tandem-impeller designs for a gas turbine compressor. Report 2001-GT-324, Pratt and Whitney Canada for ASME, 2001.*

41. *Meroni, A., Zühlsdorf, B., Elmegaard, B. and Haglind, F., 2018. Design of centrifugal compressors for heat pump systems. Applied Energy, 232, pp.139-156.*

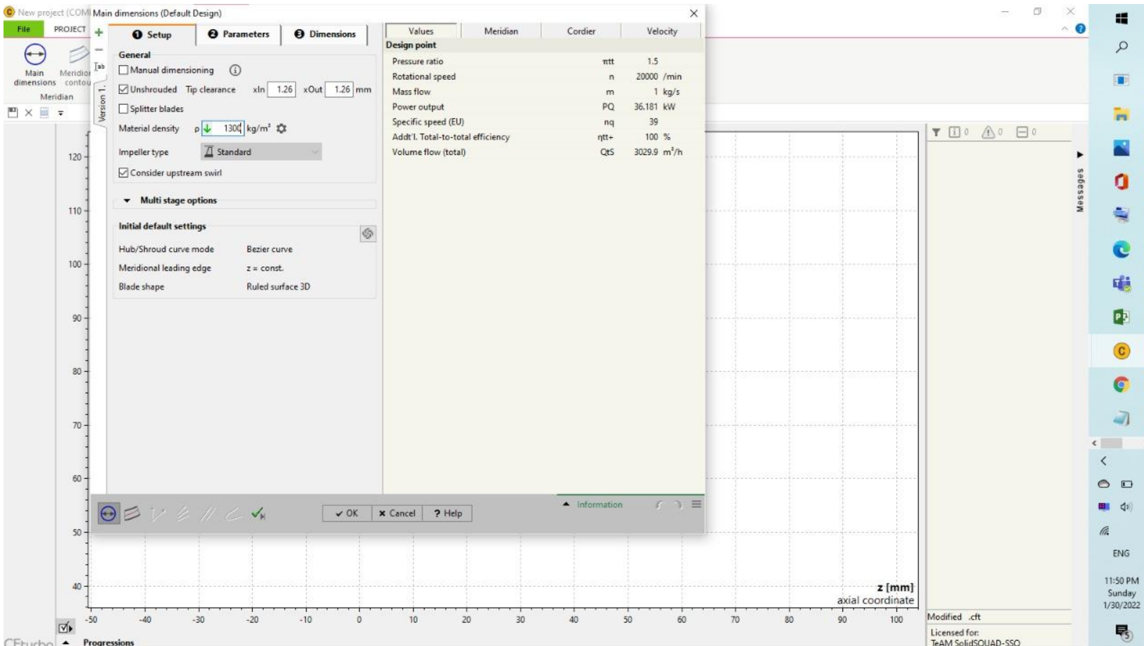
42. *Perry, R.H.; Green, D.W., eds. (2007). [Perry's Chemical Engineers' Handbook]() (8th ed.).*

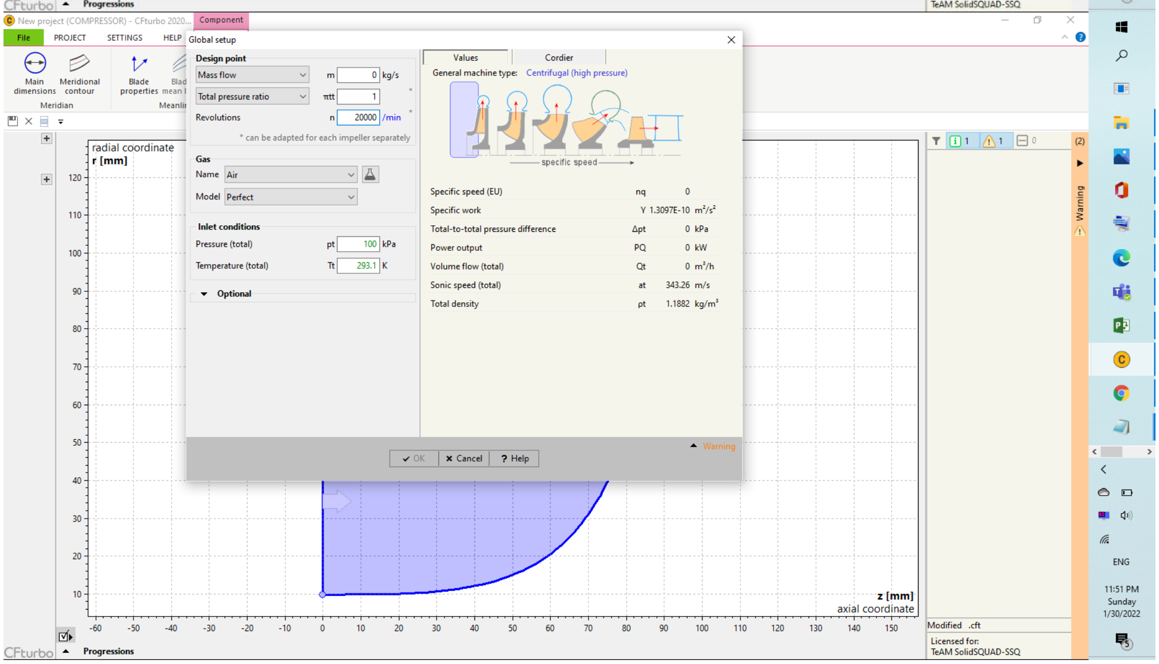
43. *McGraw Hill. [ISBN]() [978-0-07-142294-9](). https://en.wikipedia.org/wiki/Compressor*

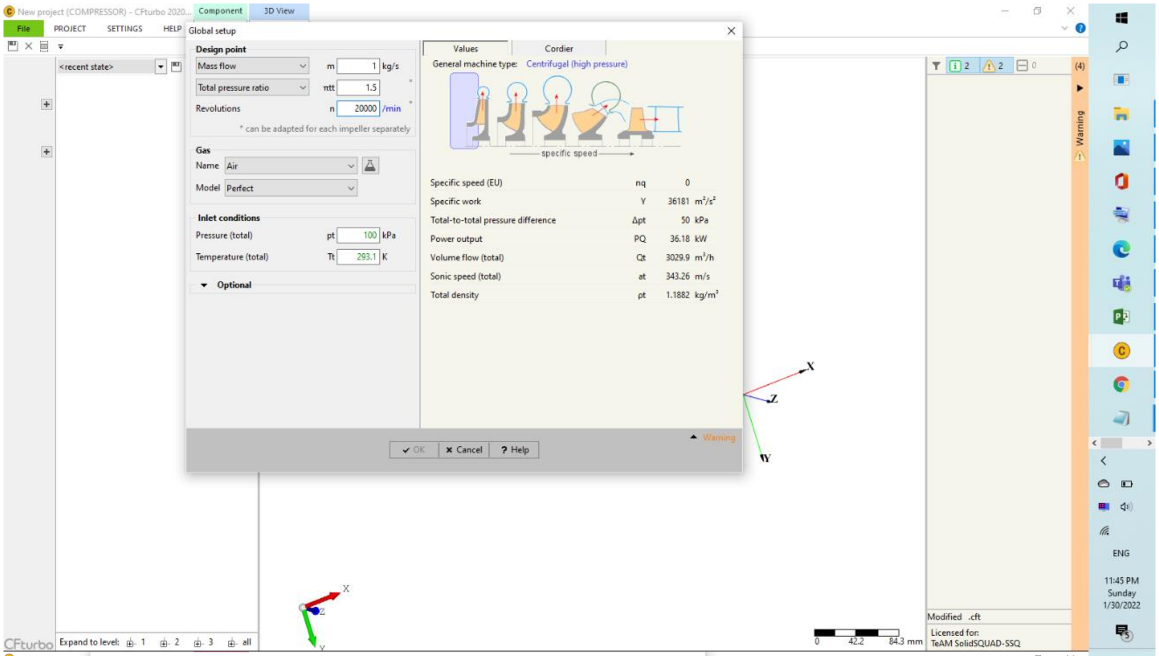
44. *Xu, C. and Müller, M., 2005. Development and Design of a Centrifugal Compressor Volute. International Journal of Rotating Machinery, 2005(3), pp.190-196.*

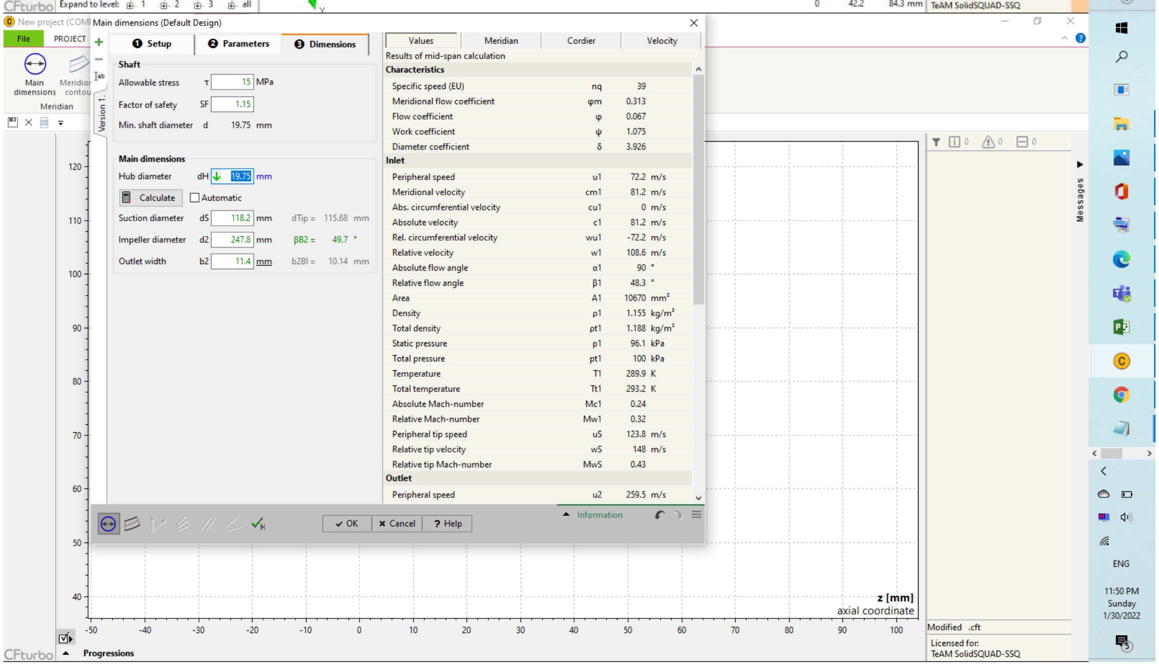
45.

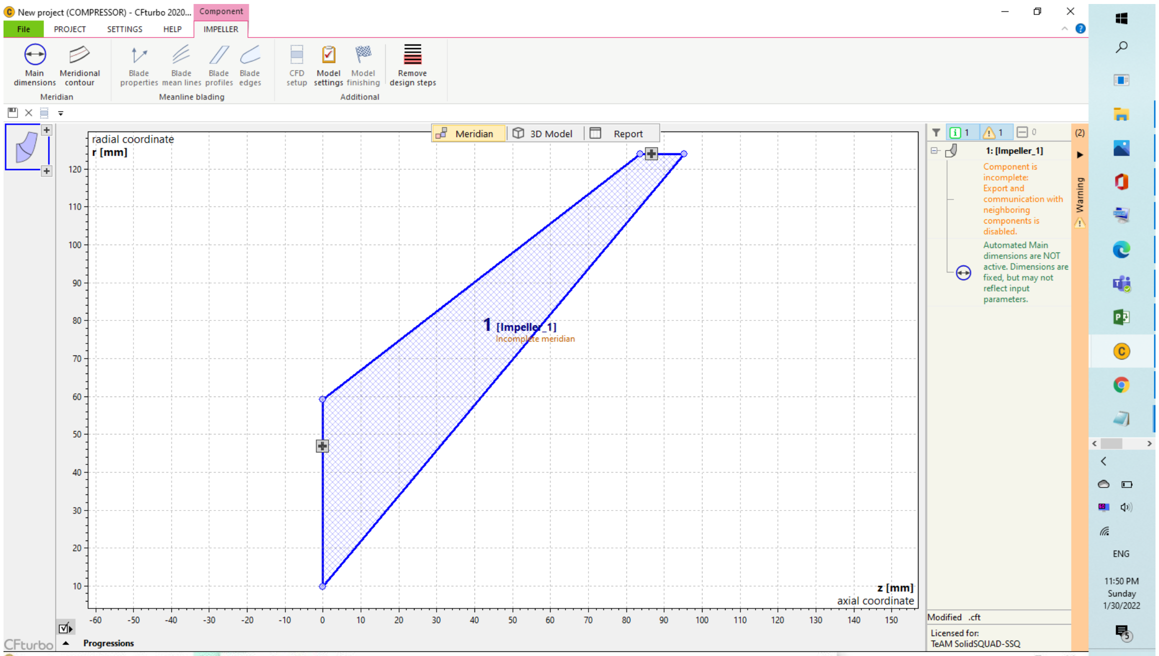
APPENDIX I: Interface of Cfturbo

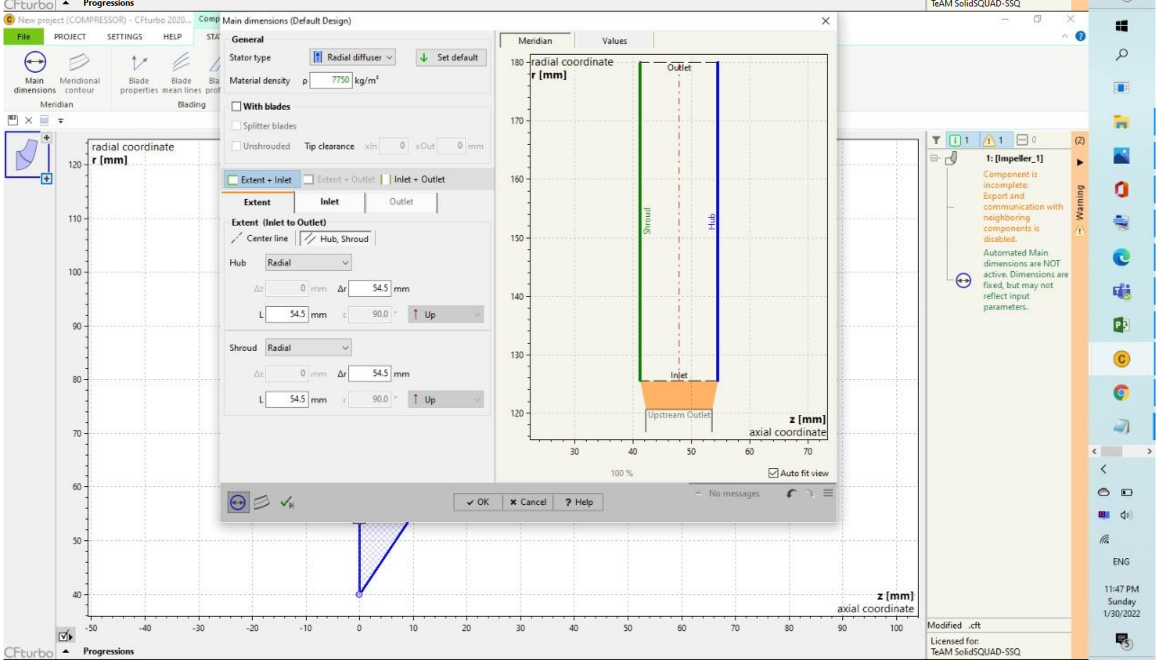





Global setup
Design point
Mass flow m 1 kg/s
Total pressure ratio πtt 1.5
Revolutions n 20000 /min
* can be adapted for each impeller separately
Gas
Name Air
Model Perfect
Inlet conditions
Pressure (total) pt 100 kPa
Temperature (total) Tt 293.1 K
Optional
Values
Cordier
General machine type: Centrifugal (high pressure)
specific speed
Specific speed (EU) nq 0
Specific work Y 36181 m²/s²
Total-to-total pressure difference Δpt 50 kPa
Power output PQ 36.18 kW
Volume flow (total) Qt 3029.9 m³/h
Sonic speed (total) at 343.26 m/s
Total density ρt 1.1882 kg/m³
OK
Cancel
Help
Warning
Expand to level: 1 2 3 all
Modified .cft
Licensed for: TeAM SolidSQUAD-SSQ
11:45 PM
Sunday
1/30/2022


Main dimensions (Default Design)
Setup
Parameters
Dimensions
Shaft
Allowable stress τ 15 MPa
Factor of safety SF 1.15
Min. shaft diameter d 19.75 mm
Main dimensions
Hub diameter dH 19.75 mm
Calculate
Automatic
Suction diameter dS 118.2 mm dTip = 115.68 mm
Impeller diameter d2 247.8 mm βB2 = 49.7 °
Outlet width b2 11.4 mm b2Bl = 10.14 mm
Values
Meridian
Cordier
Velocity
Results of mid-span calculation
Characteristics
Specific speed (EU) nq 39
Meridional flow coefficient φm 0.313
Flow coefficient φ 0.067
Work coefficient ψ 1.075
Diameter coefficient δ 3.926
Inlet
Peripheral speed u1 72.2 m/s
Meridional velocity cm1 81.2 m/s
Abs. circumferential velocity cu1 0 m/s
Absolute velocity c1 81.2 m/s
Rel. circumferential velocity wu1 -72.2 m/s
Relative velocity w1 108.6 m/s
Absolute flow angle α1 90 °
Relative flow angle β1 48.3 °
Area A1 10670 mm²
Density ρ1 1.155 kg/m³
Total density ρt1 1.188 kg/m³
Static pressure p1 96.1 kPa
Total pressure pt1 100 kPa
Temperature T1 289.9 K
Total temperature Tt1 293.2 K
Absolute Mach-number Mc1 0.24
Relative Mach-number Mw1 0.32
Peripheral tip speed uS 123.8 m/s
Relative tip velocity wS 148 m/s
Relative tip Mach-number MwS 0.43
Outlet
Peripheral speed u2 259.5 m/s
OK
Cancel
Help
Information
z [mm]
axial coordinate
Progressions
Modified .cft
Licensed for: TeAM SolidSQUAD-SSQ
11:50 PM
Sunday
1/30/2022


New project (COMPRESSOR) - CFturbo 2020...
Component
File
PROJECT
SETTINGS
HELP
IMPELLER
Main dimensions
Meridional contour
Meridian
Blade properties
Blade mean lines
Blade profiles
Blade edges
Meanline blading
CFD setup
Model settings
Model finishing
Remove design steps
Additional
Meridian
3D Model
Report
radial coordinate
r [mm]
1 [Impeller_1]
Incomplete meridian
z [mm]
axial coordinate
1: [Impeller_1]
Component is incomplete: Export and communication with neighboring components is disabled.
Automated Main dimensions are NOT active. Dimensions are fixed, but may not reflect input parameters.
Warning
Modified .cft
Licensed for: TeAM SolidSQUAD-SSQ
CFturbo
Progressions
ENG
11:50 PM
Sunday
1/30/2022


Main dimensions (Default Design)
General
Stator type
Radial diffuser
Set default
Material density
ρ
7750
kg/m³
With blades
Splitter blades
Unshrouded
Tip clearance
xIn
0
xOut
0
mm
Extent + Inlet
Extent + Outlet
Inlet + Outlet
Extent
Inlet
Outlet
Extent (Inlet to Outlet)
Center line
Hub, Shroud
Hub
Radial
Δz
0
mm
Δr
54.5
mm
L
54.5
mm
ε
90.0
°
Up
Shroud
Radial
Δz
0
mm
Δr
54.5
mm
L
54.5
mm
ε
90.0
°
Up
Meridian
Values
radial coordinate
r [mm]
Outlet
Shroud
Hub
Inlet
Upstream Outlet
z [mm]
axial coordinate
100 %
Auto fit view
OK
Cancel
Help
No messages
1: [Impeller_1]
Component is incomplete: Export and communication with neighboring components is disabled.
Automated Main dimensions are NOT active. Dimensions are fixed, but may not reflect input parameters.
Modified .cft
Licensed for: TeAM SolidSQUAD-SSQ
CFturbo
Progressions
ENG
11:47 PM
Sunday
1/30/2022