Development of a computer program to predict working and performance of a centrifugal compressor.

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
Bachelors of Mechanical Engineering

by

Daud Iqbal

Bilal Ghani

Muhammad Safi Ullah

Muhammad Asad ur Rehman

June 2021

EXAMINATION COMMITTEE

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

Daud Iqbal	210343
Bilal Ghani	212096
Muhammad Safi Ullah	207159
Muhammad Asad Ur Rehman	221914

Titled: "Development of a computer program to predict working and performance of a centrifugal compressor" be accepted in partial fulfillment of the requirements for the award of BACHELOR OF MECHANICAL ENGINEERING degree with grade ___

Supervisor: Abdul Naeem Khan, Assistant Professor SMME, NUST	______________ Dated:
Committee Member: Shamraiz Ahmad, Assistant Professor SMME, NUST	______________ Dated:
Committee Member: Dr. Riaz Ahmad Khan, Assistant Professor SMME, NUST	______________ Dated:

______________________ ____________

(Head of Department) (Date)

COUNTERSIGNED

Dated: ________________ ______________________

(Dean / Principal)

ABSTRACT

In the domain of compressors, a variety of them are employed according to their respective needs. For instance, axial compressors are being used heavily in high flow rate applications on the other hand offering large compressor ratios we have centrifugal compressors and so on. But the idea of designing one, catering one to your needs has always been an intimidating element of turbomachinery.

The solution of this resides within using commercial software but they cater primarily to OEM requirements. Our project attempts to create a software that would cater its input requirements according to the user thus minimizing prior extensive knowledge and, in the end, provide them with geometric and performance parameters while also displaying their respective charts.

Within this project, a software capable of understanding user requirements and giving them a set of dimensions and plotting performance charts was created. The program calculates these parameters while paying heed to the dimensional constraints of the user. The program uses isentropic efficiency, work isentropic and specific diameters to begin its initial results, then by appropriate assumptions it proceeds by calculating geometric parameters at each stage. Finally, alongside giving output parameters it plots performance charts that give an indication of surge and choke limits. Finally, the program was tested by verifying against Design point data using CFD (ANSYS CFX) and a commercial software (CTrend).

ACKNOWLEDGMENTS

We are extremely grateful to Allah Almighty for satisfactory progress in our Final Year Project. We have learnt significantly about project management, analytical and critical problem solving, modern engineering technologies and other important aspects while progressing in our FYP. We are thankful to our supervisor Asst Prof. Abdul Naeem Khan who provided us an opportunity to learn and gain practical knowledge. We would also like to acknowledge the help of our fellow Wasif Rehman for the development of MATLAB code. We are also very thankful to Dr. Sami- Ur- Rehman Shah and other faculty members for guiding and helping us. We hope this fruitful project will enable us to apply our skills and knowledge to polish ourselves as good mechanical engineer.

ORIGINALITY REPORT

Final report

ORIGINALITY REPORT

4%	3%	2%	2%
SIMILARITY INDEX	INTERNET SOURCES	PUBLICATIONS	STUDENT PAPERS

PRIMARY SOURCES

1	dx.doi.org Internet Source	1%
2	watermark.silverchair.com Internet Source	<1%
3	Submitted to City University College of Science and Technology Student Paper	<1%
4	www.conceptsnrec.com Internet Source	<1%
5	Submitted to Engineers Australia Student Paper	<1%
6	Johann Friedrich Gülich. "Centrifugal Pumps", Springer Science and Business Media LLC, 2020 Publication	<1%
7	curve.carleton.ca Internet Source	<1%
8	Fluid Mechanics and Its Applications, 2015. Publication	<1%

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

ABBREVIATIONS

CFD	Computational Fluid Dynamics
CFC	Centrifugal Compressor
CFX	Congregationum Fratrorum Xaverianorum
IGV	Inlet Guide Vane
DOR	Degree of Reaction
CCD	Centrifugal Compressor preliminary Design

NOMENCLATURE

D_tip	Tip diameter
D1	inlet mean diameter
M1	inlet Mach number
Beta1_geo mid	inlet geometric flow angle
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

Beta2_geo	outlet geometric flow angle
Muslip	slip factor
L	mean length
b2	blade height at outlet
Outlet.W2.ang	outlet vane angle
Nb	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
T4	static temperature at vaned diffuser
P4	static pressure at vaned diffuser
M4	vaned diffuser Mach number
Nb	number of diffuser blades
htis	Isentropic enthalpy change
eta_tt	end to end efficiency

CHAPTER 1: INTRODUCTION

Although the centrifugal impeller has been around for over two centuries, its perfection and quick development occurred only in the previous sixty years. This paper examines the early evolution of the centrifugal impeller in the final and early years of the twentieth century. Centrifugal impellers are commonly encountered in centrifugal pumps and compressors. Centrifugal pumps and compressors are thought to be the most utilized devices of our day, second only to the electric motor. The scope of application for these rotary machines is constantly expanding as they are designed to handle a larger range of liquids and gases at higher pressures and temperatures, and entire industries become more and more reliant on them.

The centrifugal impeller in pumps and compressors has occasionally aided each other's growth. In general, pumps were the major source of information in the nineteenth and early twentieth centuries, and compressors profited from it, but compressors have been the major source of information after 1945.

A centrifugal compressor is a mechanical device that compresses a fluid utilizing the radial acceleration of the impeller, which is surrounded by the compressor housing. The impeller in a centrifugal compressor increases the speed of the working fluid (gas or air) by transforming the air/kinetic gas's energy into speed. Furthermore, the diffuser turns the speed of the air or gas into pressure energy. Radial centrifugal compressors have a better high-pressure ratio at low flow rates, which gives them a significant advantage over axial compressors.

A centrifugal compressor is a device that compresses a certain vapor or gas using the compressor impeller. If further compression is required, these compressors can be configured in numerous stages. Each stage contributes to the overall pressure increase. Depending on the pressure needs for various functions, numerous stages can be connected in series to achieve the appropriate pressure. These multi-stage compressors are used in the processing, gas, and oil sectors. Sewage treatment plants, on the other hand, use single-stage low-pressure applications to achieve the requisite pressure ratio.

Centrifugal pumps and compressors are now available in sizes ranging from a few Watts to Megawatts, with efficiency levels exceeding 90%. The history of the centrifugal impeller is traced in this work. The factors that aided and hampered early development are also examined.

1.1. Problem Statement

To develop a program that can determine different parameter of a centrifugal compressor by taking minimal inputs from the user.

1.2. Objectives

Our main objective from this program is to calculate the parameters of a centrifugal compressors along with the performance curves of the centrifugal compressors. The results obtained from the program should be in close proximity to the results from the analysis by ANSYS CFX and should be close to the results generated by a commercially available software (Ctrend). The program should suggest the feasible geometrical sizes of different parts of the centrifugal compressor.

1.3. Motivation

The centrifugal compressor has their applications in many industries such as aerospace industry, automotive industry, and chemical industries etc. There is much more application of centrifugal compressors but there are less dedicated softwares to determine the parameters of the centrifugal compressors. Most of these codes do not have generalized loss models and do not account for off design performance of the centrifugal compressor. Our motivation is to develop a code to determine the parameters of the centrifugal compressor using generalized loss models moreover, to determine the off-design performance of the compressors.

1.4. Applications

The program has various application in different industries which incorporates the use of centrifugal compressors such as aerospace industry, automotive industries, and chemical industries etc. The program can be used to predict the performance of centrifugal compressors used in these industries.

The program can also be used by the students to help them studying about centrifugal compressors.

CHAPTER 2: LITERATURE REVIEW

2.1. Components of Centrifugal compressor stage

The various components in a centrifugal compressor are discussed below:

2.1.1. Impeller

The most significant component of a centrifugal compressor is the rotor or impeller. A centrifugal air compressor impeller, also known as a rotor, is a disc with a key assembly coupled to the compressor's shaft. This disc is attached to a variety of curved blades. These blades serve as a route for the working gas or air to diffuse through. The number of these bent blades in a single centrifugal compressor impeller varies between 15 and 20.

The compressor's impeller provides speed or velocity to the air or gas passing through the chambers linked to the rotating disc. Depending on the performance necessary, these blades can be inclined backward, radially, or forward.

2.1.2. Vaneless Diffuser

Vaneless diffusers provide a larger flow range but poor pressure recovery and efficiency. The geometry of the vaneless diffuser is basic. It is made up of parallel or nearly parallel walls that form a radial annular path from the impeller outlet radius to the diffuser outlet radius. The diffuser is frequently followed by a volute or a collecting chamber, which directs the flow to a single exit.

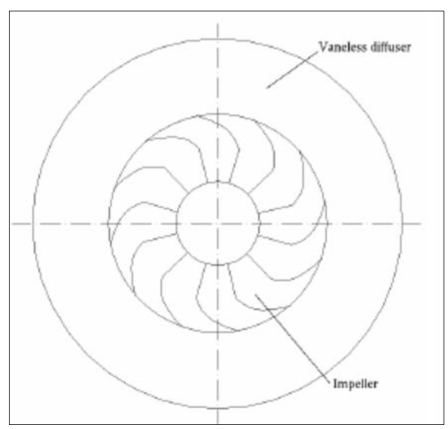


Figure 1: Typical Impeller and vaneless diffuser

2.1.3. Vaned Diffuser

Vaned diffusers have a greater pressure recovery and efficiency but a shorter flow range. The vaned diffuser outperforms the vaneless diffuser in terms of pressure rise and efficiency. The vaned diffuser, on the other hand, has a shorter operating range and more sophisticated shape, resulting in a more expensive design. Vaned diffusers are employed at high pressure ratio stages where a vaneless diffuser would be impractical, such as big turbochargers and gas turbine compressors.

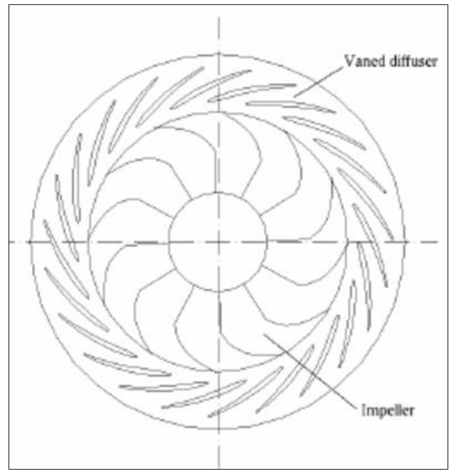


Figure 2: Typical vaned diffuser

2.2. Vaneless space

There is a certain amount of vaneless space that exists between the impeller discharge and the diffuser itself in any vaned diffuser design. It is needed not only to smooth out unsteady fluctuations, but also to disperse extremely high-speed flow to more manageable speeds. It is a very complicated and non-trivial matter to choose the extent of the vaneless space, as Japikse (1996) explains in detail. However, since the flow inside the channel diffuser is called steady and one-dimensional, the analysis becomes a little easier. Furthermore, along the channel, the velocity has only one component.

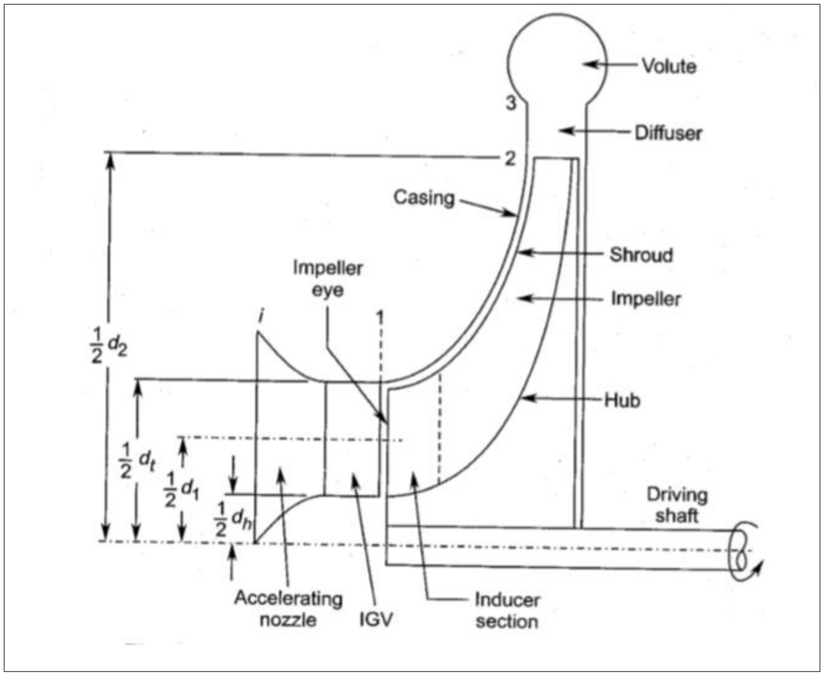


Figure 3: Cross Sectional view of a centrifugal compressor

2.3. Stage velocity triangles

In turbomachinery velocity triangles are of vital importance while analyzing their behavior. Usually, various components of velocity at the inlet and outlet section of a turbomachine are shown in them. The velocity triangles are generally drawn for the r, theta, x coordinate system. Below is a brief description of velocity triangle elements.

- Absolute and relative air flow angles denoted by α and β.
- Tangential velocity denoted by U due to rotation and because the change in radius is not negligible so values of inlet and outlet tangential velocity will be different.
- C_t is the tangential velocity component while C_r known as the radial component of C which is the absolute velocity.
- W is also known as the relative component of the velocity at inlet and outlet.

To analyze these velocity triangles are drawn in different planes. At inlet they are drawn using axial and radial velocity components. While at the outlet since the flow has been turned 90 degrees. The velocity triangle is drawn for radial and tangential components of velocity. Below is attached a sample picture of the velocity triangle at the compressor outlet for backward swept blades.

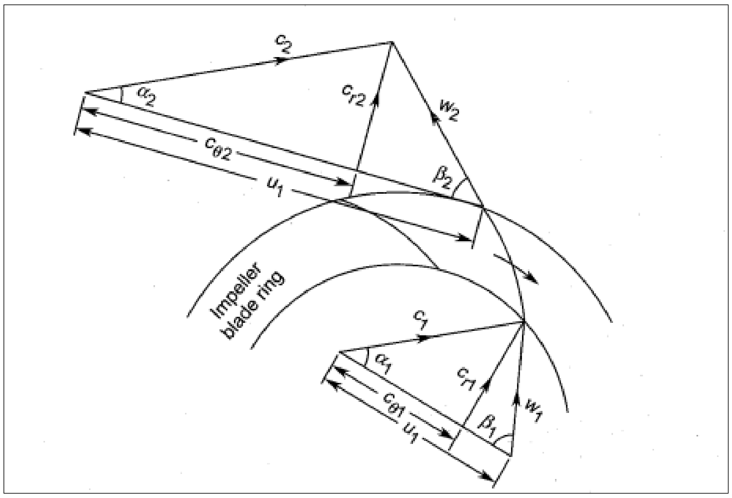


Figure 4: Backward Vane impeller velocity triangles

2.4. Analysis Method

Generally, analysis of centrifugal compressors should be 3D three dimensional. Because during its operation the plane of fluid flow is changed i.e., it has axial inlet and radial outlet. But 1D (one dimensional) techniques are used extensively in literature because. They are less complex and offer reasonable results. So, they are suitable for early-stage design and prediction. This one-dimensional method has been extensively developed throughout the years.

2.5. Prediction methods available

2.5.1. Zero zone

The number of zones means the number of flow paths available. Zero zone method is simplest of all which does not involve geometrical data of the compressor directly. It uses some common non dimensional parameters such as flow coefficient and tip Mach number etc. If given the values of these non-dimensional parameters can give us rough idea/estimation of compressors performance for example if we know that tip Mach number is high, we can easily judge that there will be shock losses at inlet of this compressor which will be affecting its performance. These non-dimensional parameters are applied for extremely early stages with limited input and can also be used to compare two turbomachines when we know all the data about them.

2.5.2. Single zone

Single zone model takes assumptions of uniform flow through the impeller. It is more of a geometry-based model constructed using basic physics. In this approach we divide the overall losses into separate component losses. The components can be thought of as stations. These stations are shown below in figure also. But these single zone models are limited due to their semi-empirical nature. [2] As an example Galvas model was mainly constructed for aircraft engine compressor. Although there are separate selection methods available to choose most suitable loss model for our need but even if we apply it due to unviability of appropriate data the single zone model cannot predict off design characteristics.

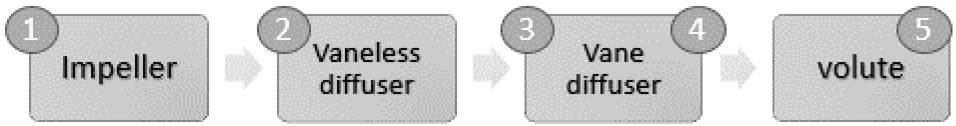


Figure 5: Single zone stations in a centrifugal compressor

2.6. Loss generation mechanisms

The loss generation mechanism is complex in turbomachinery as there are numerous losses involved. As, some components of a single turbomachine are static, and others are rotating. The losses mainly arise due to entropy generation. The losses are mainly categorized in three parts.

- Viscous friction losses (friction between flowing fluid and surface)
- Heat transfer losses (Due to high-speed of rotation in turbomachines high temperature are reached. So, unnecessary heat generated which is to be lost as it is not our desired form of energy in case of compressors)
- Non-equilibrium losses (shock losses which occur at high speed at inlets mostly for Ma>1)

The figure below shows detailed sources of losses which shows that several ways energy can be lost. In our project we have tried our best to include the maximum number of these losses due to get our results as close to real data. Physically, this overall loss of energy is felt in the form of less pressure achieved at output then desired

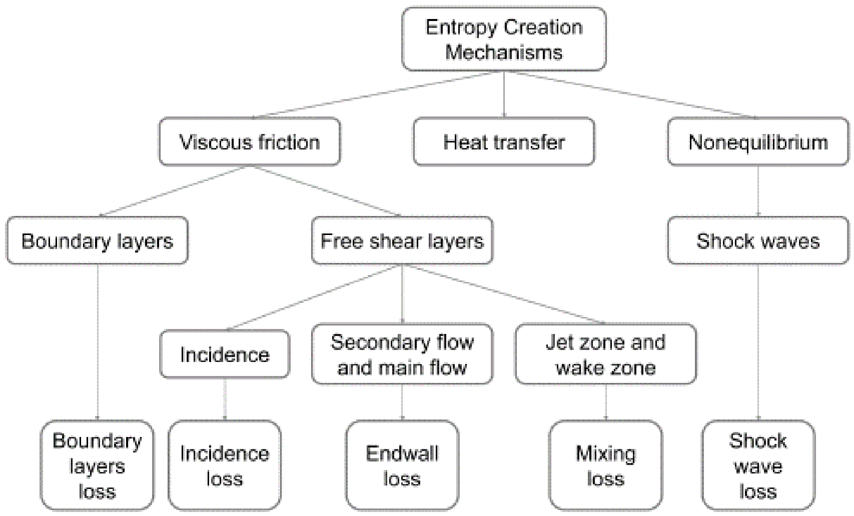


Figure 6: Various Energy Loss Mechanisms

2.7. Optimum set of loss models

During the past few decades, several loss models have been developed and are documented in literature. Several loss models for centrifugal compressors and different correlations might exist for a single case. When a performance prediction is made, there are hundreds of combinations to be selected. Before doing the 1D one dimensional analysis a set of optimum loss models should be chosen. Several researchers, including Galvas, Aungier, and Doust Mohammadi among others, have determined an optimum set of loss models. In the later section of the report, it is mentioned how choice of these optimum models was done using the available literature.

2.8. Off-design characteristics prediction

These are the conditions (Pressure, Temperature, and gas mixture) at which new output is obtained at the compressor inlet. The off-design conditions alternate the conditions at inlet to consider during the design of compressor stage or real conditions at inlet at a particular time of operation of compressor.

From the standpoint of fluid dynamics, strict flow similarity at each performance point is needed. As a result, the flow and head coefficients and Mach number must all be kept constant. The following are the requirements for the proposed method:

- Compressor maps for reference and design. The diagrams (given by the manufacturer) provide an accurate indicator of the capacity of the machine to be used as the initial point. The predicted diagrams are typically issued during the commercial and test performance as well as design stage diagrams are typically issued after the compressor has been shop or field checked at the end of the manufacturing phase. The diagram can provide the input for the method, even "tested" diagrams are more suitable.
- Relevant inlet conditions, such as pressure, temperature, and gas mixture, are connected to reference/design charts. This data set is needed to comprehend off-design calculations. Output conditions that are not what they should be. These conditions (Gas mixture, Temperature and Pressure) at the compressor inlet indicate a new output. Off-design conditions can be utilized to consider substitute channel boundaries during the blower configuration stage, or they can be the genuine delta conditions during blower activity.

Off-plan execution computation necessitates the extraction of invariant data associated with the blower behavior. This information is then used to recreate the exhibition under new circumstances. These calculations are intimately linked to the thermodynamics of gas pressure and genuine gas blends, which means that differences in gas mixture segments, pressing factor, or temperature may have significant consequences. Because of the connections between these various demonstrating territories, the estimation technique may provide a precise forecast of blower execution.

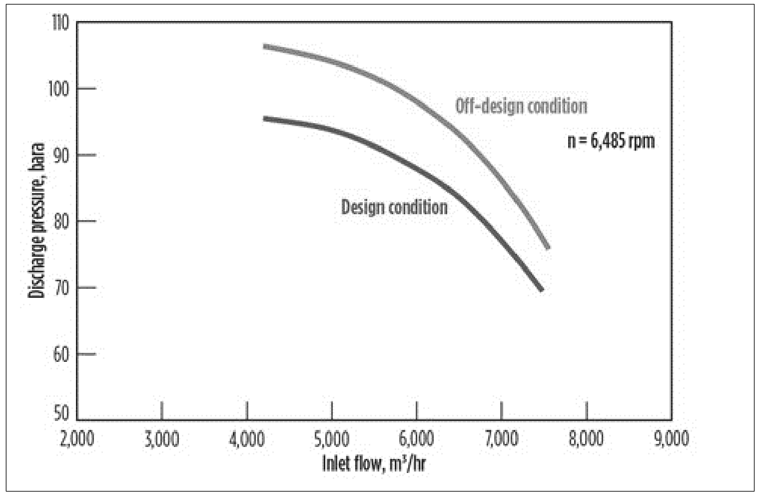


Figure 7: Typical performance chart

Almost all centrifugal compressors operate in an off-design state, meaning they are not operating at their design or guarantee stage. Engineers and operators must be able to understand and forecast how a compressor will work under off-design conditions from both a design and operational standpoint.

2.9. Factors affecting performance

Compressor performance not only depends on its geometry alone. But also, there are numerous other factors which affect compressor performance directly or indirectly. However, some of them have significant effects on compressor performance which are discussed here in this report after careful selection. For example, the properties of gas being compressed i.e., molecular weight and compressibility effects. For ideal gas we can use the relation PV=mRT but it is not true for real gases. In case of real gases, we must include the effects of compressibility also. The compressibility is denoted by Z. Furthermore, this compressibility element includes the amount of energy used to pack a gas. Furthermore, if the gas's subatomic load increases, the blower's pressing factor proportion and mass stream rate increase as well, for the same explicit work

Inlet condition effects performance as input generally dictates output. As, in our case we are using inlet stagnation pressure and temperature at the start of our program. The inlet conditions affect pressure ratio and work done by the compressor. Their effects are to be discussed in the results section.

The presence of Inlet guide vanes IGVs is also an important factor while assessing performance because if IGVs are not present at inlet severe shock losses may occur. Also, depending on the direction of vane angle positive or negative pre-whirl is produced. The disadvantage of positive pre-spin is that, as compared to negative pre-spin, a positive gulf spin speed reduces the energy move and lowers the Mach number. So, positive pre-whirl is more desired because the compressor needs less energy and Mach number advantage is also present.

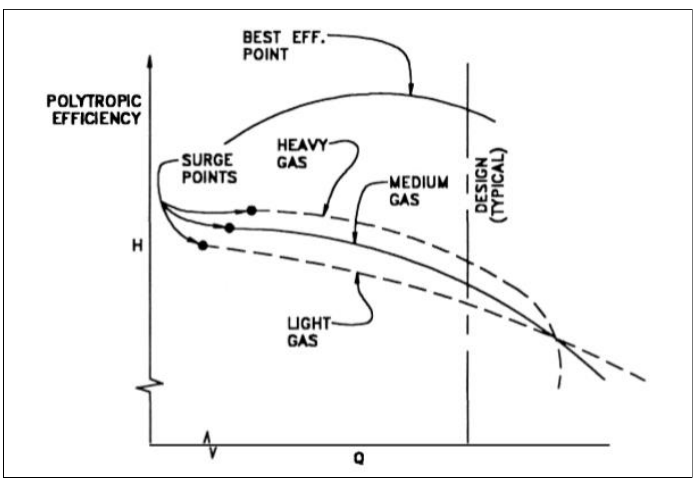


Figure 8: Effect of gas properties on efficiency

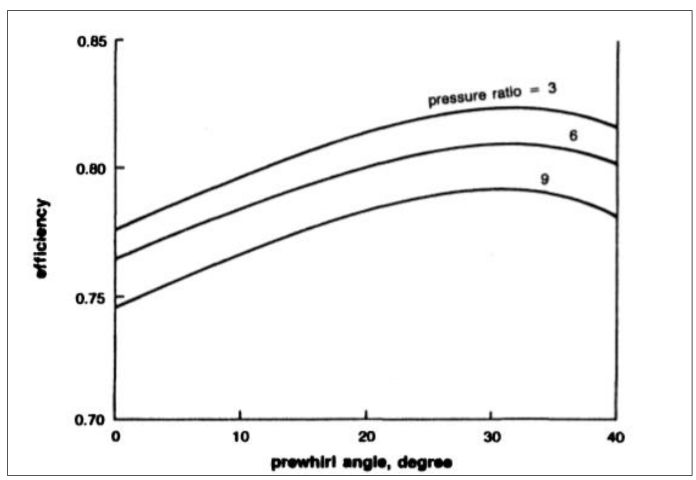


Figure 9: Effect of a pre-whirl angle due to IGV

The blade shapes of centrifugal compressors also play an important role on how much compression we can achieve under given conditions. The blades are categorized on the criterion of exit blade shape as forward curved, backward, and radial blades. For each blade shape we obtain a separate velocity triangle. For same flow rate and the head by changing the volume flow rate (through axial velocity change) for three blade shapes Turbine equation by Euler is used and head achieved was determined. This exercise results in a plot as show [4],

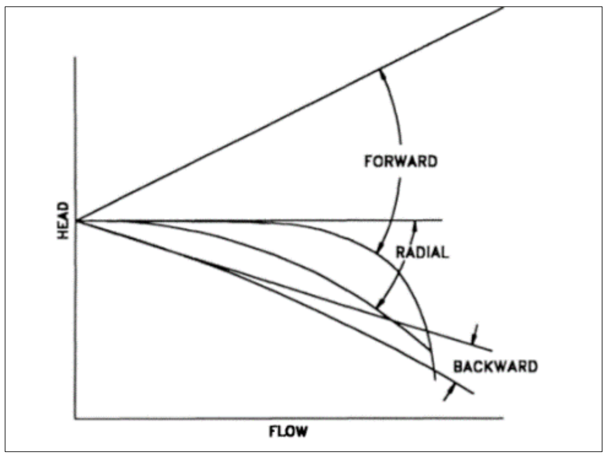


Figure 10: Effect of blade shape on compressor performance

2.10. Performance assessment

The situation becomes a bit more complex with a radial machine. Since the wheel speeds i.e., U1≅U2 are no longer equal, the coefficients can be defined differently. The typical convention is to take the largest radius (exit radius for a compressor or pump and inlet radius for a radial turbine) as the basis for the U value.

2.11. Centrifugal Compressor Stresses

2.11.1. Axial thrust in high pressure centrifugal compressor:

The residual axial thrust acting on the rotor of a centrifugal compressor is caused by the non-uniform pressure distribution on the impeller with the process gas in centrifugal compressors. Differential pressure acting on the impeller faces, as well as the contribution from the process gas's momentum variation. During the design process, the axial load was held below the thrust bearing capacity under all possible operating conditions; this is an important parameter to consider when evaluating thrust load to ensure that thrust bearings last a long time.

The axial impeller force (F_{unbl}) is the difference between the axial forces on the discharge (F_2) and suction (F_1) sides of the impeller shroud $F_{unbl} = F_2 - F_1$. The axial thrust portion (F_1+F_m) of non-hydraulically balanced closed impellers (i.e., with suction side shrouds) is:

$$\mathrm{F_m} = \rho \mathrm{Q} \Delta \mathrm{V_{ax}}$$

$$\mathrm{Q} = \text{Flow rate}$$

$$\rho = \text{Density of the fluid handled}$$

2.11.2. Radial thrust in a centrifugal compressor:

The radial thrust of a centrifugal compressor impeller at its nominal operating point is described as:

$\mathrm{R} = \mathrm{K}\rho\mathrm{gHDB}$

$\mathrm{R} = \text{Radial force}$

$\mathrm{K} = \text{Radial force coefficient}$

$\rho = \text{Density of the fluid handled}$

$\mathrm{g} = \text{Acceleration due to graity}$, $\mathrm{H} = \text{Head}$

D = Outside diameter of the impeller

B = Impeller outlet width

2.12. Stresses Related to the Impeller of the Centrifugal Compressor:

There are three main type of static stresses that are present in the impeller of the centrifugal compressor:

2.12.1. Axial Stresses:

The stresses due to axisymmetric loading on the impeller and acts parallel to the axis. Mainly this type of stress occurs due to the difference of pressure the change in pressure pushes the impeller in the lower pressure side which causes the stress in the impeller known as Axial stress and the formula to evaluate the magnitude is given as:

$$\sigma_{ax}(r) = \frac{3+v}{8} \times \rho \times \omega^2(b^2 - r^2)$$

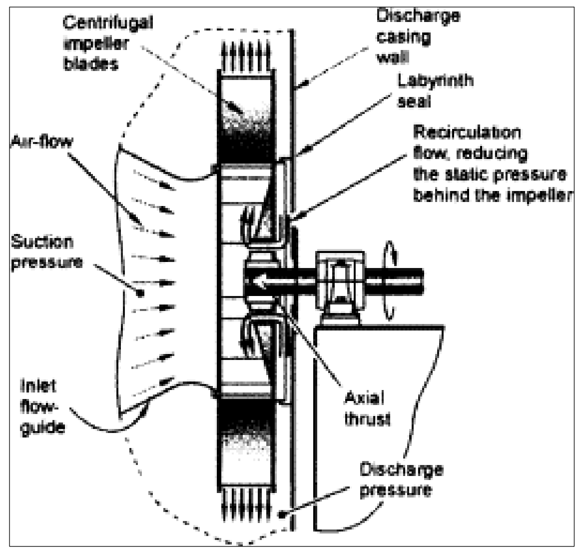


Figure 11: Axial Stress Representation

2.12.2. Radial Stress:

The stresses due to radial loading. This stress usually occurs because of the pressure exerted by the discharge fluid, it is the reaction of the force that the fluid exerts on the impeller casing that is dur to the direction of the discharge fluid, which is tangent to the circumference of the impeller, the formula to calculate the radial stress for the impeller of the centrifugal compressor is given as:

$$\sigma_r = \frac{(3+v)\delta\omega^2}{8g}\left(R^2 + R_o^2 - \frac{R^2 R_o^2}{r^2} - r^2\right)$$

2.12.3. Hoop stress:

It is also known as tangential tensile inertial stress. It a stress that occurs in the tangential direction of the impeller and is perpendicular to the radial stress, this stress occurs due to the Tangential thrust which an impeller exerts on the working fluid the formula to calculate the hoop stress in the impeller blade of the centrifugal compressor is given as:

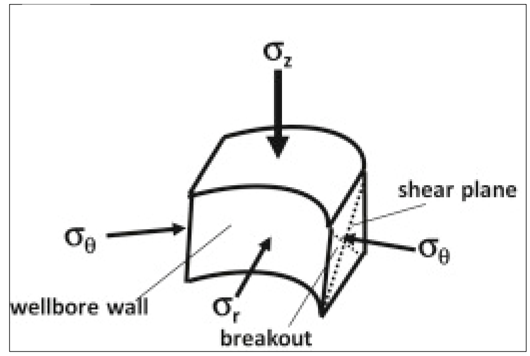


Figure 12: Hoop Stress on wall

$$\boldsymbol{\sigma_h = \frac{\delta \times \omega^2}{8g}\left[(3+v)\left(R^2 + R_o^2 + \frac{R^2 R_o^2}{r^2}\right) - (1+3v)r^2\right]}$$

CHAPTER 3: METHODOLOGY

The methodology opted for this code is primarily the 'Inverse Design Approach'. In this approach the user inputs a set defined criterion i.e., parameters according to his design requirements which are usually the parameters like overall pressure ratio required or any other constraints. Then according to those factors outputs parameters that are mainly geometric parameters defining sizing of whole compressor using various turbomachinery concepts and iterative schemes. Now this inverse design approach is suitable for us because we want to develop a solution that can be used frequently. This approach is faster because in this we know what we desire and calculate remaining parameters based on those which we desire. As, opposed to direct design approach in which we must input new sets of geometric parameters again and again until our desired pressure ratio is achieved.

To implement this methodology, we opted for MATLAB as it is much easier to program and can cater for many of our programming expansion requirements. Since, by hands iterations would have been cumbersome so all the calculations ranging from basic to complex loops were done in MATLAB to save time and our effort. Also, the commenting was done to make our code readable for general readers. Now, proceeding with the working of the program. As the program runs, the user is greeted with an interface that implores the user to input certain parameters. These parameters will then dictate the geometry, which will be displayed for the user.

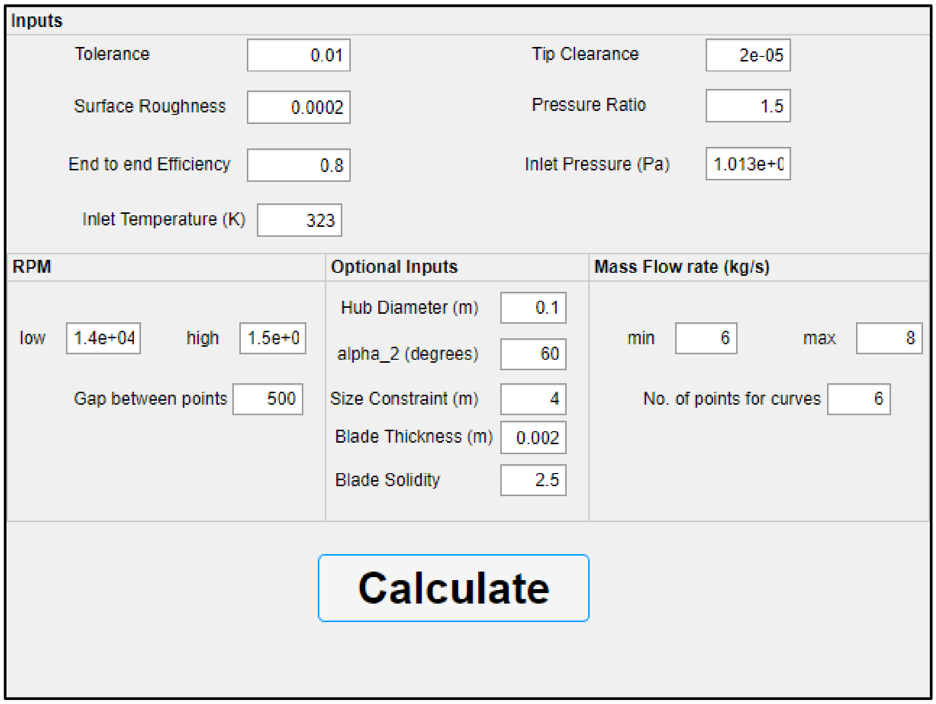


Figure 13: Our code input interface

Now proceeding to what happens inside the code, the code comprises ten sub-files. These files work in unison to form the entirety of the CFC program. Each of which are segregated to ease the perception of the data flow and highlight the different sections of calculations. These files are named as

- main
- inlet_calc
- inlet_loop
- inlet_angle
- outlet calc
- outlet loop
- vanless diffuser
- vaned diffuser
- vaned diffuser diameter
- stresses

The concepts applied and working involved and all refinements done in the program to make it convenient for the end user at the same time not compromising on quality of results written in those subfiles are discussed below in form components / stations of centrifugal compressor.

3.1. Inlet Calculations

As mentioned earlier in inverse design we input the desired pressure along with there are some common parameters which are initially know inlet pressure, temperature mass flow rate and RPMs. First case being discussed here is the case in which we want geometric results for only one fixed input. However, analyzing multiple data sets and their graphing was possible and this option will be discussed at a later stage. Isentropic work was calculated using given pressure ratio and inlet temperature. This work and given volume flow rate were used to find specific speed. Form this speed we needed our specific diameter. For that purpose, we used concept of Cordier diagram. The advantage of Cordier diagram is that it requires minimal compressor knowledge and because of this rudimentary nature it is applicable to every design. The downfall however is predicting off design characteristics from it, thus making optimization improbable.

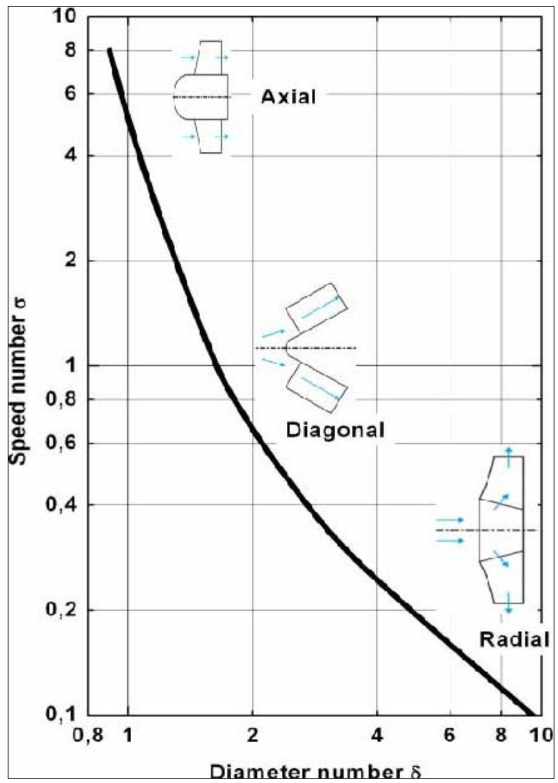


Figure 14: Cordier Diagram

That range is typically 0.5-2.0 for Specific Diameter and 2-4.5 for a Specific Speed. We had two options available first one was that we used complex real time graph scanning and its detection methods in MATLAB. But it was much beyond our programming capabilities

So, instead we made an equation of this curve using logarithmic regression model. Using that we calculated specific diameter from specific speed. This diameter helps us in calculating the outer diameter of impeller D_2. Based on the impeller diameter, a reasonable value of inlet hub diameter can also find. This is then used to calculate the velocity of the impeller at the outlet tip, which is then used to calculate the fluid flow velocity. The first optional which was kept was this inlet hub diameter D_{hub} which according to literature is generally 20% of D_2.

3.1.1. Tip Diameter Concept

Here the main idea was that relative velocity (or relative Ma number) at the inlet of the impeller should be minimum so that shock losses can be reduced. So, here MATLAB's function 'fmincon' is utilized. Previous inputs of mass flow rate, density of fluid, Hub diameter. From the value of Hub Diameter, the program incrementally increases its value onto another variable and proceeds to do so until a value of tip diameter is reached where relative Mach number is minimum. The behind it is to write the relative velocity equation and set it equal to zero however in program we set that equal to action and that function was minimized varying the D_{tip} value until that becomes nearly zero that D_{tip} value is our desired value. Other methods of lowering relative inlet Ma numbers also exist, specifically for high pressure ratios. One of which is through introduction of pre-whirl at the impeller inlet through Inlet guide vanes. But this was out of scope of our current project. However, IGVs can easily be added by modifying a few equations.

3.1.2. Impeller Inlet

After calculating the inlet fluid velocity, we assume a free vortex method and calculate the hub, tip and mean velocity triangle parameters. Using the prescribed value of RPM of impeller, the speed of impeller at hub, tip and mean diameters are calculated. Then using the velocity triangles, relative velocity magnitude and angles are calculated. Then absolute and relative Mach number values are calculated alongside the Static Temperature of fluid at the inlet midspan. After that comes an iterative density calculation loop at the inlet of the impeller. Considering that air is a compressive fluid, this density calculation is crucial and will enable us to calculate other parameters. Initially we do not know density at the very inlet of compressor. So, guessing it initially same as inlet conditions stagnation density and solving an iterative loop, convergence criterion is assessed by recalculating the value of density, based on this difference the loop runs again till the value of density is very close. The parameters calculated in this file are Flow Inlet Area, Absolute Inlet Velocity, Static Temperature, absolute Mach number and Static Pressure.

Now, after using the velocity triangles, we have found the relative velocity angles at the impeller inlet; these angles correspond to the blade inlet angles and not the angles of the fluid. The fluid does not fully follow this path but instead has its own relative flow angle termed as beta $_{flow}$ in the sub. The file proceeds by calculating two different area parameters termed as Inlet area and Optimal Area. The Inlet Area is the circumferential measurement that considers that blade thickness at the circumference must be accounted for. The Optimal Area term ignores this. Using these two circumferential measurements and the geometric angle, we can estimate the fluid inlet angle relative to the impeller.

3.2. Impeller Outlet

We do not know any angle at the outlet. So, the only way we can proceed is by assuming that the outlet fluid angle, α_2 will be 60 degrees. This is because Optimum Impeller tip absolute flow angle lies somewhere between 60 and 70 degrees, which the user has the option of a 10-degree modification window if he desires. Using this angle and the value of Eulerian Work calculated previously, we can calculate for other parameters concerning the velocity triangle for both absolute and relative velocity components. The main input parameters concerning this portion are outlet flow angle (assumption justified above), Eulerian Work and Outer peripheral speed and resultantly gives velocity components and non-isentropic temperature at the outlet.

After solving for the velocity parameters, the program proceeds onto calculating total and Static Values of Temperatures. To make the value as close to the actual value, we have used end-to-end efficiency in this calculation. The program then proceeds to calculate the absolute, peripheral, and relative Mach numbers at the exit. Although we are calculating shock losses at the impeller tip. They are of little concern as it would not achieve sonic conditions because of increase in static pressure of compressed air, resulting in increase in value of speed of sound, making shock losses less improbable.

By calculating outlet blade height and density, which satisfies continuity by performing iterations on it. Once efficiency is converged, it outputs values of relevant thermodynamic properties at the outlet. This efficiency is further corrected by calculating various losses. The inputs are Inlet velocity components and thermodynamic properties and outlet velocity values, Eulerian Work, max iterations limit and tolerance/accuracy. It resultantly gives the fluid properties at the outlet, the blade angles at the inlet and the number of blades.

It proceeds by first assuming an isentropic process for the rotor to begin the iteration process. In this way, the convergence of real pressure at the outlet is achieved. This way efficiency is initialized at a value of 1. The loop begins by initially calculating the Total and Static Temperatures and Absolute Mach numbers at the exit. Then it proceeds to find the Isentropic Outlet Pressures, both static and total. Then it calculates the density and outputs the blade height, while also checking the stability i.e., finds the blade height to outlet diameter ratio. It then finds the average flow deflection at the midspan of the inlet.

To aid in calculating the number of blades, we must set a certain solidity value. Using theory, we find that it would be best to keep the value of solidity at around 2.5. We then calculate the number of blades and to minimize the blade loading, we add another blade to the number of blades and then also calculate the pitch and chord.

3.2.1. Losses in Impeller

The code then calculates the Slip factor and free stream velocity both absolute and relative along with the geometric outlet angle and proceeds to calculate losses. These losses occur due to difference in the geometric and fluid outlet angle. Here, thickness of the blade has been assumed for our ease. Calculating the losses was not itself a challenge rather choosing which loss model to include was quiet a difficult task for us. After extensive research we found relevant research data. Help was taken from research already done on this topic by HwOH and ES Yoon who identified six basic loss mechanisms as shown in the table below. Then out of these six they made 144 possible combinations and tested them by comparing them with experimental results. Finally, they made three sets of 6 loss generation mechanisms, and we chose one of them.

Loss mechanism	Loss model
Incidence loss	Aungier (**5**) Galvas (**9**) Conrad *et al*. (**10**)
Blade loading loss	Aungier (**5**) Coppage *et al*. (**11**)
Skin friction loss	Coppage *et al*. (**11**) Jansen (**12**)
Clearance loss	Aungier (**5**) Jansen (**12**) Krylov and Spunde (**13**)
Mixing loss	Aungier (**5**) Johnston and Dean (**14**)
Vaneless diffuser loss	Coppage *et al*. (**11**) Stanitz (**15**)

Figure 15: Loss models classification by HwOH et al.

To calculate the tip clearance losses, we found out that through Gaetani's work that to aid in calculating tip losses, the tip clearance is to be taken as roughly 2% of the blade height. This clearance value then helps us in calculating the enthalpy loss due to clearance between tip of the impeller and shroud of the compressor.

To calculate blade losses, we define certain parameters such as Hydraulic Length and Average Relative Inlet Velocity.

In calculating Frictional losses, we start by calculating outer perimeter and area. Recall that the outlet of a centrifugal compressor is rectangular. Using these two parameters, we will define a term known as hydraulic diameter which will determine our flow regime via Reynolds number.

To calculate skin friction losses, we first find out the relative roughness of the material that was selected which is then used to find the friction coefficient which then later helps us in finding enthalpy loss in friction.

Next, we calculated the new efficiency keeping in mind the enthalpy losses. If the efficiency value has converged, the iterative process breaks, and outputs new total enthalpy change and a new isentropic/Eulerian Work.

3.3. Vaneless diffuser

In this file we calculate the necessary quantities for a vaneless diffuser. Within this code, subscript 2 refers to the outlet of the impeller/ inlet of the vaneless diffuser and subscript 3 refers to the outlet of the said diffuser. We start by assuming a ratio between the outlet diameter of impeller and the vaneless diffuser diameter. It is important to note that the pressure can be increased if this diameter ratio is increased. However, this can only be done up to a certain limit otherwise efficiency of this entire stage would decrease. In literature it found that the recommended value for this ratio is roughly around 1.05 to 1.20. So, as a safe estimate a value of 1.20 is taken as an assumption. A reasonable design value of this stage significantly reduces losses in the vaneless diffuser segment.

In addition, we assume that the blade height will remain the same, i.e., b3 = b2. Then an iterative loop was defined where we initialized the average density and the average velocity with the compressor outlet conditions. As a result, density and velocity at the inlet and outlet of the diffuser are set as equal respectively to begin the optimization process. Recall that since the diffuser is basically a stator, and no work is done on the fluid here. So, ignoring the losses at this stage, the stagnation temperature before and after this stage will be equal. After this initialization we define a Hydraulic Diameter term and begin the iterations.

The iteration first Calculates Average Quantities of density, velocity, and Reynolds Number, it then finds the Skin Friction Coefficient for the vaneless diffuser. We then calculate the Outlet Velocity of the Vaneless Diffuser in terms of two components, tangential and radial, while also evaluating outlet velocity and magnitude and angle, alongside static temperature, and Mach number values. The program then calculates enthalpy losses occurring at this stage. Static pressure at the outlet of diffuser is also determined using isentropic values of Temperature. Using this pressure, the density at the outlet of diffuser is evaluated and the convergence of the loop is observed. Using this iteration, a good estimate of density is now available at the diffuser outlet.

3.4. Vaned diffuser

In this section we included data calculations for a particular type of diffuser. The diffuser calculations are not purely theoretical, and we must consult design data for tested diffusers. Also, there are various types of vaned diffusers based on geometric configurations. There are strong advocates of each diffuser style and often it. We selected conventional air-foil type diffuser to be included in our program. Since due to time-constraint, we could not include all options to use any of the type of diffuser. The reason for choosing this type is that it more compacts in size. It is the only style for which systematic design approach was available. This design also offers efficiency gains over low solidity vaned diffusers.

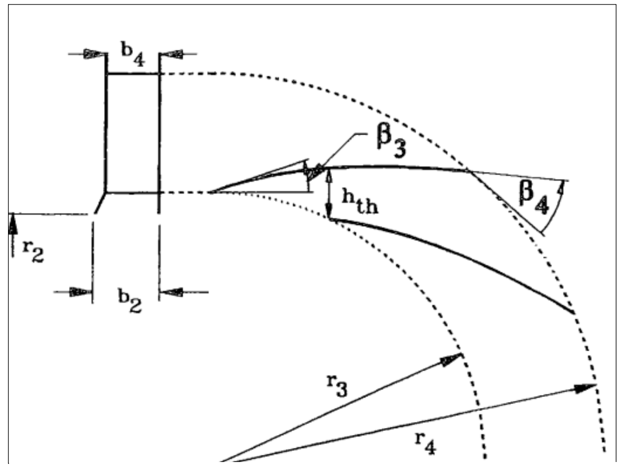


Figure 16: Vaned diffuser geometric parameters

After finalizing our selection looking at the table below, we find that there are different Parameters that need to be taken into consideration while selecting diffusers from different authors. Using the Angular approach, we compute the discharge area blockage that matches the experimental pressure recovery requirements while assuming that the flow isentropic and there is perfect guidance for the flow by the vanes. This discharge blockage parameter donated by B gives us a reasonable parameter for correlation purposes; it is however not wise to merely on blockage alone considering that there is subsequent mixing of momentum viscous flow with the inviscid resulting in losses. Upon further examination of the theory provided by Aungier we observe that two other parameters are of prime importance; these are denoted by θ_c and L the value for L should be less than one third and the value of θ_c should be less than 5.5 as is apparent from the graph attached below where the twice of the angle value is kept roughly 11° degrees for maximum diffuser section efficiency.

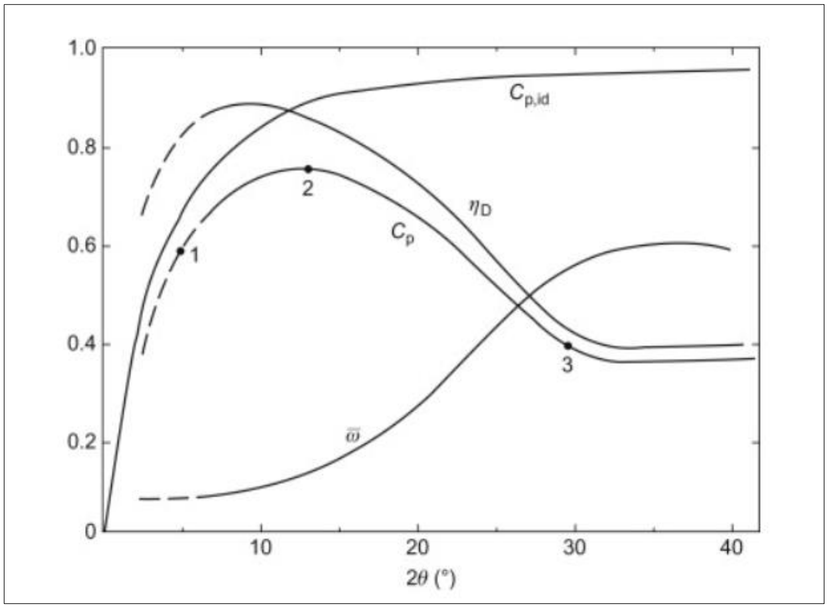


Figure 17: Variation of diffuser efficiency with angle 2θ

From the table below we observe that two diffusers VD-5 and VD-7 are the most effective of the series as they are very close to the design limits but if you look at VD-16 it has a much higher area ratio so in a program we decided to opt for VD-16 whose parameters are also shown in the table above. Using theory, we come to know that maximum efficiency is found at the divergence angle value of somewhere between 8 degrees and 10 degrees. Also, the other reason for choosing this diffuser VD-16 is because the coefficient of pressure recovery is known for this diffuser.

Table 1: Design data for tested diffusers

Diffuser	R	A_R	B_4	L_B/W_3	$2\theta_C$	L	c_p
VD-1	1.350	1.54	0.0722	6.05	5.11	0.1455	0.56
VD-2	1.350	1.77	0.1752	5.43	8.11	0.2483	0.58
VD-3	1.350	1.92	0.1832	6.15	8.56	0.2745	0.64
VD-4	1.350	1.89	0.2665	3.88	13.09	0.4070	0.53
VD-5	1.350	1.89	0.1592	4.79	10.62	0.3297	0.65
VD-6	1.350	1.89	0.1909	5.70	8.93	0.2770	0.62
VD-7	1.350	2.07	0.1993	5.48	11.15	0.3271	0.68
VD-8	1.350	2.17	0.3483	4.31	15.46	0.4394	0.55
VD-9	1.350	2.17	0.3119	5.14	13.00	0.3684	0.60
VD-10	1.420	2.05	0.2630	6.14	9.77	0.2562	0.61
VD-11	1.420	2.18	0.2897	5.16	13.05	0.3326	0.63
VD-12	1.350	1.79	0.1658	5.12	8.82	0.2710	0.60
VD-13	1.450	1.89	0.2099	7.03	7.24	0.1777	0.60
VD-14	1.450	2.17	0.2953	6.43	10.40	0.2503	0.62
VD-15	1.600	2.19	0.3101	8.88	7.67	0.1427	0.61
VD-16	1.600	2.40	0.3628	8.43	9.49	0.1774	0.62
VD-17	1.442	2.70	0.3918	7.17	13.50	0.3258	0.63
VD-18	1.395	2.50	0.3518	6.41	13.35	0.3565	0.62

Then we defined parameters such as Length to Width ratio (LWR), Divergence angle θ_c, Channel Aspect Ratio, Pressure Recovery Coefficient, and the diffuser efficiency. Again since, we are considering the diffuser as a stationary flow passage, there is no work being done to the fluid resulting in same stagnation temperatures. So, after calculating the desired temperatures and pressures, we can calculate the density. To calculate the enthalpy losses, we look at for points from inlet to outlet of the diffuser and notice that the difference in enthalpy is given by the real value minus the isentropic one. After evaluating these losses, we can calculate other thermodynamic properties and resultantly find the velocity and Mach numbers at the exit. Now, we can calculate the total isentropic enthalpy change then divide by Eulerian Work of our compressor resultantly we can get the total efficiency and the total enthalpy.

Finally, to calculate one of the important parameters the vaned diffuser diameter. During this, we use the assumption that the outlet angle of the vaneless diffuser is the angle of suction side of the conventional diffuser. So, using trigonometry and solving for the wedge geometry we can calculate the remaining geometric values such as Diffuser Channel Length, Diffuser channel projected length, total length of the suction side, length suction side to radial direction and finally radius of the vaned diffuser.

3.5. Stress Analysis

The stress analysis was also included as an integral part of our design calculations. Because there is possibility that the compressor might fail due to excessive stress. A detailed stress analysis involves vibration and dynamics stress analysis of impeller, but that thing requires FEM analysis of a particular compressor model and was out of the scope of our project. We kept the analysis as simple as possible yet providing us the correct failure stress criterion. We basically employed basic physics principles here and treated impeller like a rotating disc. Hoop stress was calculated for a rotating hollow disc. After that, a stress safety factor was also calculated by comparing the hoop stress with yield stress of material of impeller. Major parameters needed for stress analysis were density, Tensile Yield Strength, Shear Yield Strength, and the Poisson Ratio. We then proceeded to max impeller diameter possible for that material based on strength of that material. In the end compared that maximum possible diameter with our D_2 to calculate outer diameter safety factor.

In the end to integrate all these files. For the preliminary design this "main" function runs the calculations function for a specified number of iterations or until convergence is reached. Initially it asks to guess the end-to-end efficiency and uses it as the new guess value to calculate the Eulerian work for the centrifugal compressor.

3.6. CFD Analysis

In order to validate our results, we had two approaches one was using actual tabulated experimental data The Other was through computerized fluid dynamics in the realm of fluid dynamics we had two options.

Record to CFD software in her primary reliant on walls ANSYS FLUENT and ANSYS CFX. The first perk regarding ANSYS FLUENT is that it requires a much in depth mesh analysis and can be extended to a wide variety of geometries according to the need however CFX is different it is primarily designed for the analysis of turbomachinery whilst having a much Friendlier user interface primary regarding geometry creation, mesh generation All the while being reliant on a model that is both integrative and flexible with regard to GUI and capabilities for customization comparing that to FLUENT is troublesome and Hectic because here you have to design a new impeller for the whole analysis from scratch using CAD tools The Other Prime aspect of using CFX was it acquired lesser computing power. So, it delivered more faster and accurate results. FLUENT on the other hand requires the presence of much more capable machines which we did not have while being Quarantined at our homes. also, it requires a much in-depth know-how of CFD knowledge which we had not studied as of this point. using CFX was our viable choice.

To properly validate our results, we decided to go with it find pressure ratio for the program which gave us a geometry on which we perform the computation of fluid analysis. For this we opted for ANSYS CFX. The results from the code, i.e., geometry was obtained at Design Point which was used as an input for our CFD analysis, and the values were compared. The CFD methodology is listed as follows:

3.6.1. Preliminary Design Creation

For the Preliminary Design Creation step which is also termed as ‘sizing’, VISTA CCD was used. The reason being is that CCD is a good program when it comes to transitioning from an initial rough design to a full three-dimensional geometry in mere minutes. This also helped us avoid over-sophistication in describing the geometry of the compressor. Once the program is started, we are greeted with the following interface, wherein design point data has already been added beforehand and a clear visual of the blade geometry is also apparent on the right.

The interface visible above consisted of four tabs, namely Duty and Aerodynamic Data, Gas Properties, Geometry and Results. After entering the necessary parameters such as Pressure Ratio, Mass flow rate, RPM etc. we move towards entering the gas properties in the adjacent tab, which will govern the working fluid behavior which in our Design Point was air. Alongside this, the gas model was selected as Ideal gas because the pressure and temperature rise of a single stage is not sufficient enough to make the gas intermolecular forces significant to rule the model as non-ideal. So ideal gas is a good enough approximation for the analysis to take forth.

The next step involves geometry parameters as input such as hub and shroud diameter, number of Intervanes, number of impeller vanes and several other parameters within the geometry tab. Finally, after selecting the relevant units, we pressed 'calculate' and it gave us the initial data in a more tabulated form.

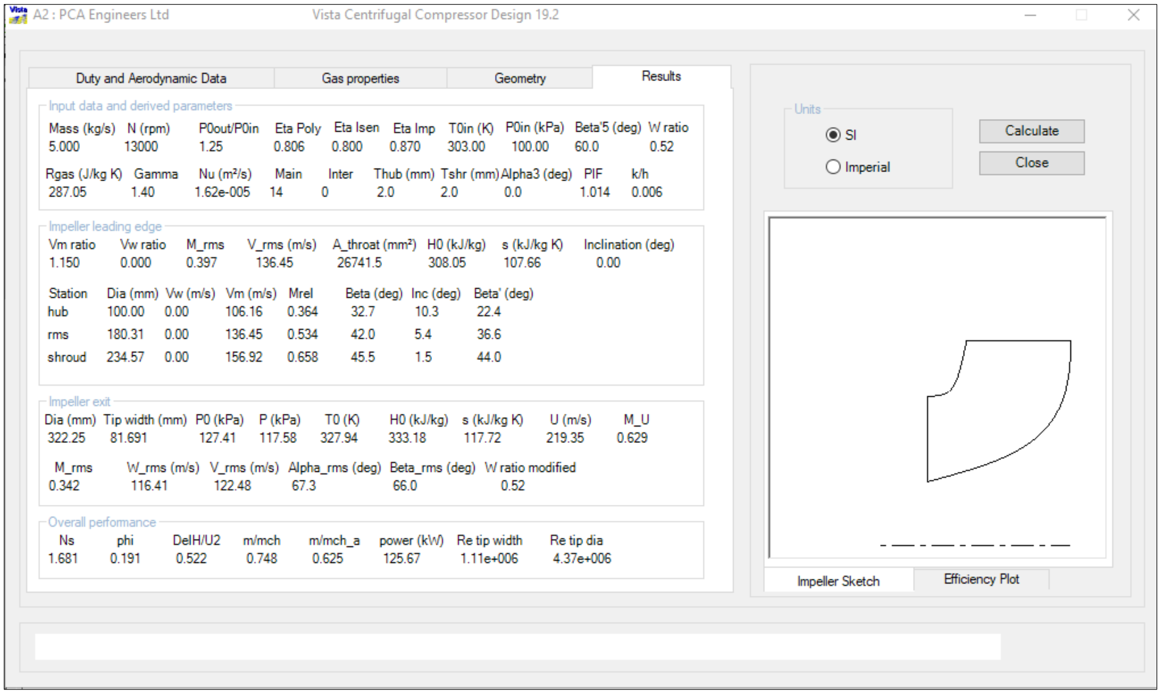


Figure 18: Vista CCD Interface

3.6.2. 3D Geometry and Mesh Generation:

Next, for even more design editing, we can also use BladeGen features to alter the geometric parameters such as normal layer thickness, angle variation along the meridional plane etc. in a spanwise manner, but it is not needed here as we used constant thickness and relatively simple geometric parameters within our program. So, in the event a more complex geometry needs modelling or spanwise variation is needed then this interface will be the way to go. But for now, we only need this to finalize the blade geometry and import it for meshing.

To make the geometry region ready for meshing we must also specify the impeller inlet and outlet flow regions which can also be altered using BladeGen. Using the inlet hub data and the inlet diffuser flow we can estimate an inlet and outlet flow profile that which will guide the flow of the working fluid.

Now we can create a grid for the generated blade model using TurboMesh, where once the program loads, we are greeted with a single blade view of the impeller while also highlighting the Inlet and Outlet regions alongside the Shroud and Hub regions for proper flow visualization. To have proper grid generation, a modest y+ value was added for emphasis on near wall parameters calculation considering if the flow reached a high Reynolds number value, the emphasis on the interaction between the working fluid and the blade must not be overlooked.

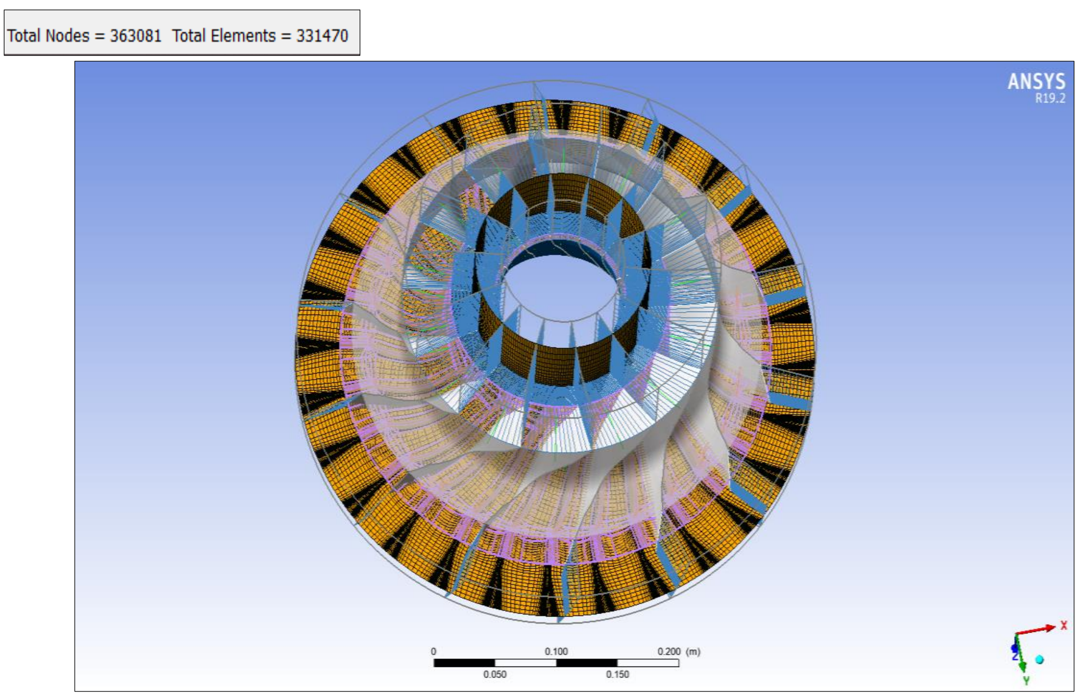


Figure 19: TurboGrid Mesh generated

Next, we finally generate the mesh, a normal mesh with a reasonable mesh quality is generated for validation purposes. Once the mesh is generated, we can alter the element size if more accuracy is to be desired, but the mesh used thus far is suitable enough. After this mesh generation we can also trim our Inlet and Outlet regions or the Topology in a more three-dimensional manner to guide the mesh according to the flow requirements.

3.6.3. Pre-Processing

After a mesh has been generated, we can export it to ANSYS CFX for providing Boundary Conditions and solving for our model. Here, we specify the Turbomachine type for our solver which is a Centrifugal Compressor here. An rpm value as well as the direction of rotation for the impeller is added. Then in Physics definition we can again, get an overview of our working fluid conditions. Here we selected an Ideal Gas Model for Air and chose a suitable Turbulence Model.

The turbulence model chosen here was Shear Stress Transport (SST) model which is a hybrid of k-ω and k-ε turbulence models. In this model, the eddy viscosity is limited so that in wake regions improved performance can be obtained. Using this model we use the k-ε model in the fully turbulent region and k-ω in the near wall region. This was a good simulation that was achieved using the strengths of both models. The appropriate boundaries are then rechecked for ensuring they do not overreach or are improperly defined.

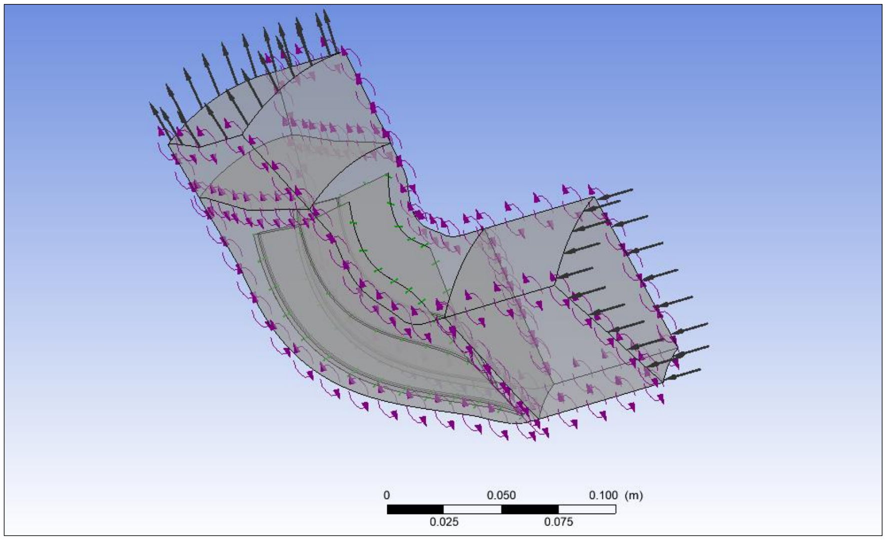


Figure 20: CFX – Pre blade domain

Regarding the solver controls, we chose High resolution advection scheme and Turbulence Numerics, with maximum number of iterations of 300. The convergence criteria were chosen of a target of 1e-6 with RMS Residual type. Now we proceeded to solve it in CFX Solver and defined the Run Definition. The solution was set to run and converged at about 80 iterations. After the iterations we proceeded onto CFD-Post where relevant contours and plots were generated which will be elaborated in the results section.

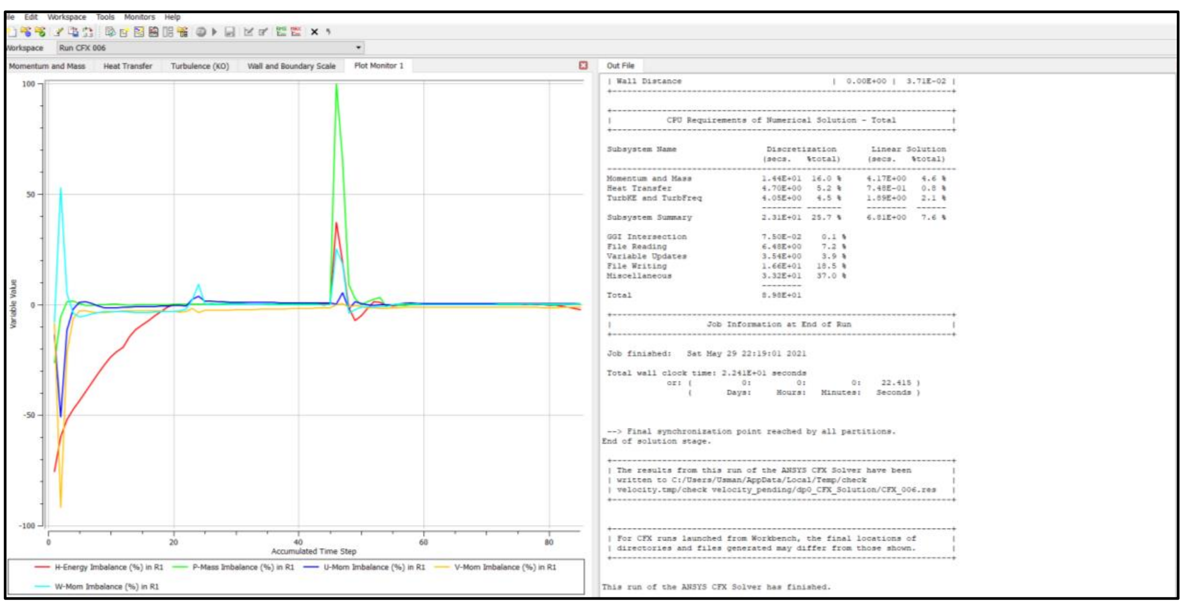


Figure 21: Normal Force Convergence Chart

3.7. Drawback of Cordier Diagram:

The Cordier-diagram is approximated by a simple equation giving the best possible specific speed for a given specific diameter as shown in above in this report. The best achievable efficiency is then given by a second equation, thus determining all three parameters, and the major advantage of this model is that it requires very limited knowledge of the compressor and is thus applicable to any design, as rudimentary as it could be. The drawback is a very limited design space, which gives no room for optimization. And it is of course impossible to estimate off-design performance with this equation. That is why when calculating off-design results. Our code removes the constraint applied on specific speed and diameter.

CHAPTER 4: RESULTS AND DISCUSSIONS

Since we are using an inverse design approach it will give us various geometric and flow parameters keeping in view our specified output. As such direct comparison cannot be made with data available at hand for example through data collected from real time compressors. However, a comparison can be made to cross check by using direct design software available online by inputting the geometry we obtained from our code and see whether we can get the desired pressure ratio or not.

4.1. Parameter at App output

Below here are shown the results which we got at the output of our app.

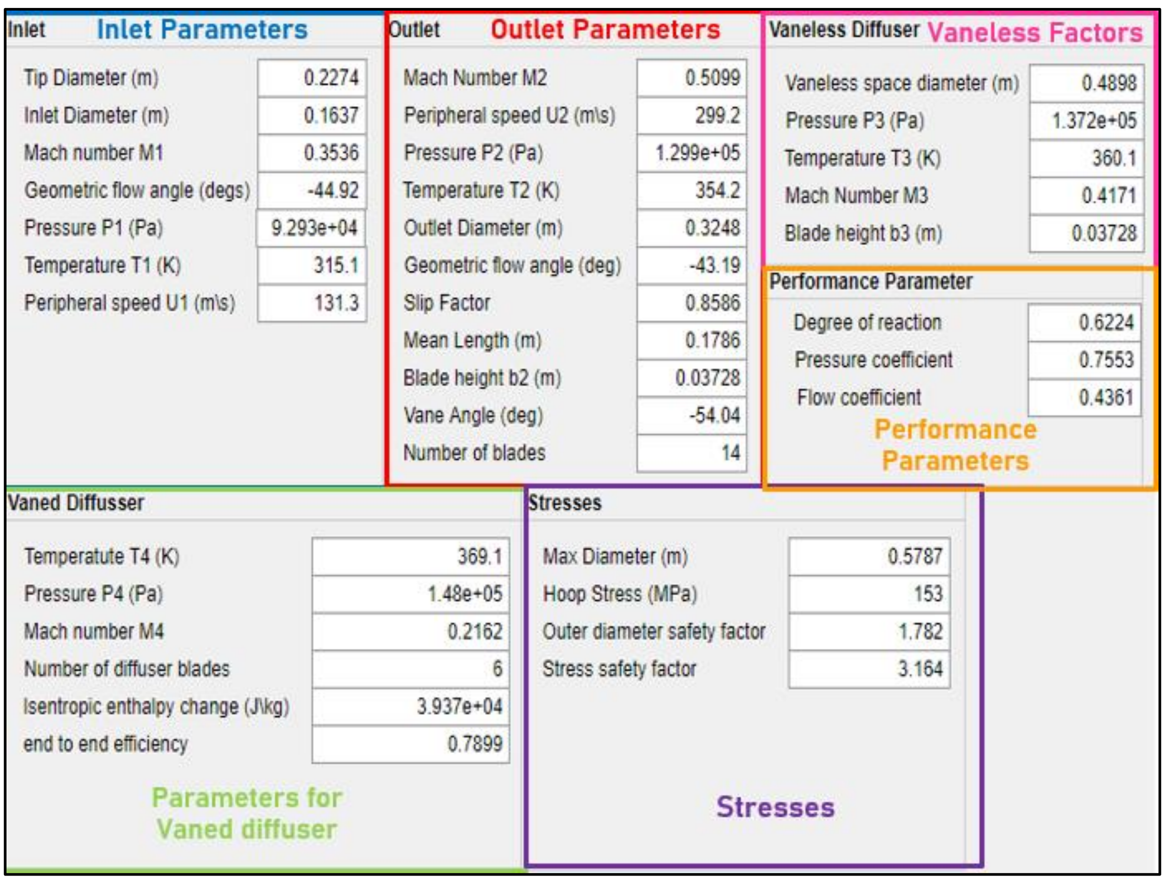


Figure 22: Results from App

The major parameters included almost all the geometric parameters except from those which we asked at input. And apart from geometry information we get thermodynamic parameters ranging from temperatures to Mach numbers at every station of the centrifugal compressor. And other important factors like safety factors and performance parameters were calculated.

4.2. Performance Parameters

Our code produced results of various performance parameters including degree of reaction, flow coefficient, pressure coefficient. In literature we have some well-defined relations among these performance parameters. Below are those well-defined relations graphically shown and with the values from our design point calculations.

4.3. Degree of Reaction and Flow coefficient

The graph below shows the relationship between flow coefficient and degree of reaction for various relative flow angles at outlet. The following graph shows us the relation for backward swept blades. In our case the beta 2 angle was approximately 25 degree and flow coefficient were 0.25 this leads to a degree of reaction of 0.79 which is in line with the value we get from our code i.e., 0.7830. It can also be seen and verified from code that degree of reaction increases with flow coefficient for backward curved blades and vice versa for forward curved blades.

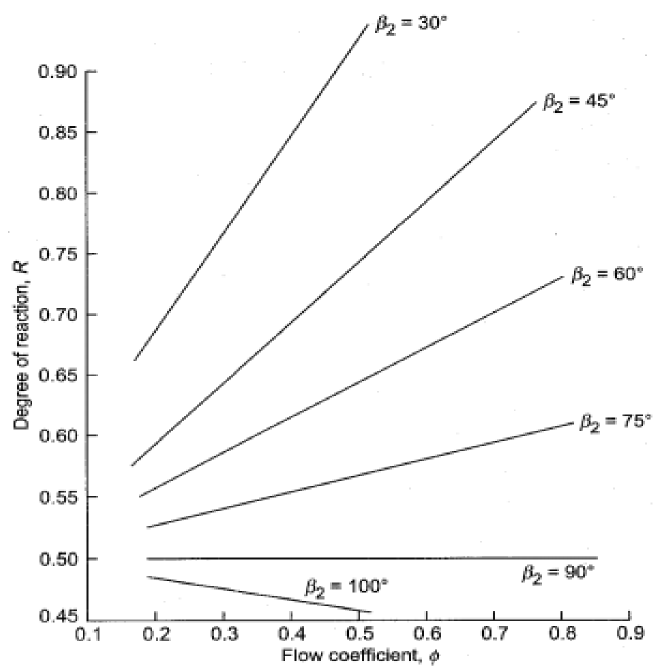


Figure 23: DOR vs ϕ variation

4.4. Degree of Reaction and Pressure Coefficient

The graph below shows the relationship between pressure coefficient and degree of reaction for various relative flow angles at outlet along a slanted line. The following graph shows us the relation for all kinds of blade configurations. In our case the beta 2 angle was approximately 25 degree and pressure coefficient were 0.4340 this leads to a degree of reaction of 0.78 which is in line with the value we get from our code i.e., 0.7830. And it can also be seen and verified from code that degree of reaction increases with pressure coefficient and degree of reaction are inversely related and backward curved blades give a higher degree of reaction at low pressure coefficient.

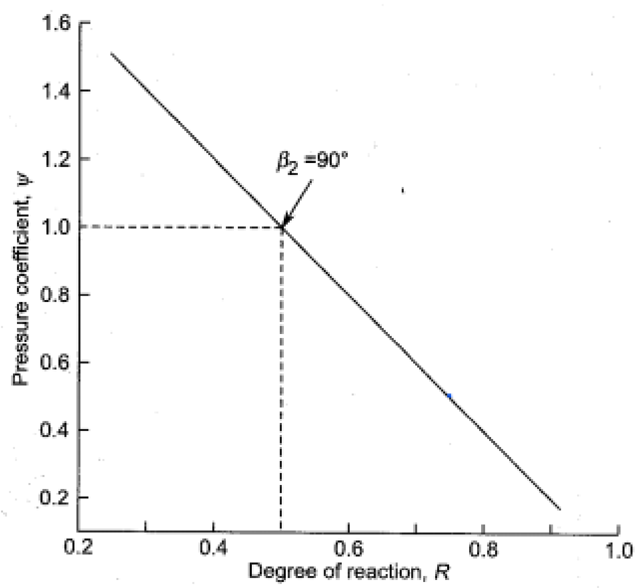


Figure 24: DOR vs Ψ variation

4.5. Performance Charts

Radiating blowers can handle a wide range of stream speeds and pressures. The choice of individual part size and working conditions, such as channel pressure and delta temperature, influence the working exhibition bend. The performance of any centrifugal compressor can be fully understood by looking at these charts. These charts as mentioned earlier are used to assess off design performance of a compressor as well. They are also used to define surge limits of any compressor hence play a role in defining safe operating limits. OEMs include these charts in their catalogue as well. Below I have discussed the results obtained from our code with charts available from the manufacturer.

4.6. Discharge temperature trend

Observing the performance chart obtained from our code we see the following trends.

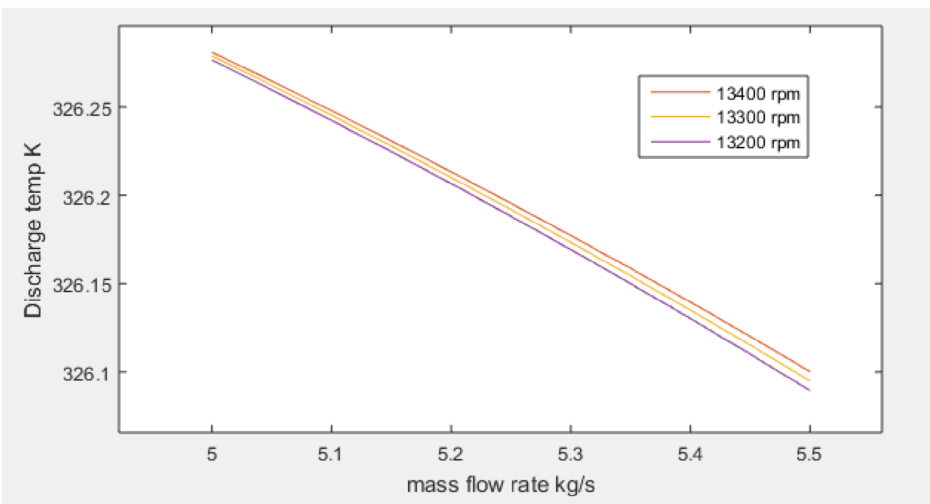


Figure 25: Discharge temperature vs flow rate chart

- The mass flow rate increased at the same RPM the discharge temperature is decreased.

- As RPM is increased for a given flow rate discharge temperature will increase.

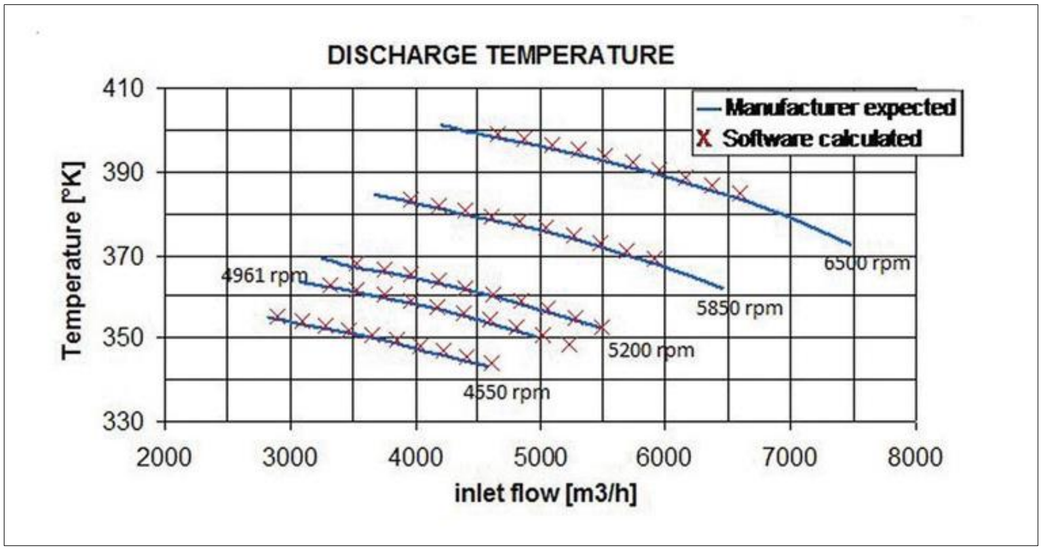
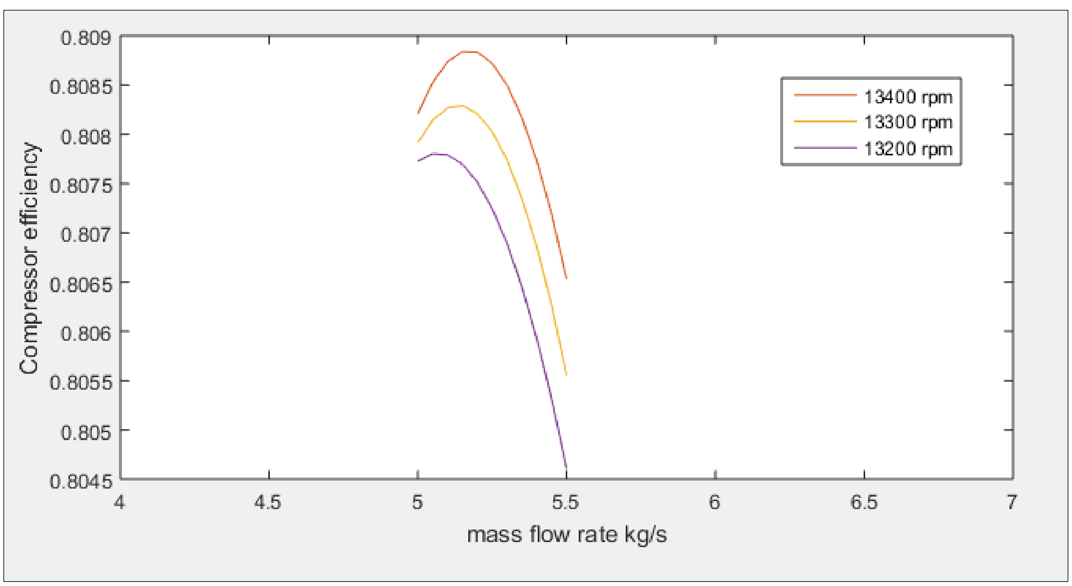


Figure 26: Sample temperature vs flow rate chart

4.7. Polytropic efficiency trend

Observing the performance chart obtained from our code we observe the following trends.



- The mass flowrate is increased at the same RPM the polytropic efficiency is decreased.

- As RPM is increased for a given flow rate the efficiency seems to get increased.

Below is a sample chart from a manufacturer to compare the trend.

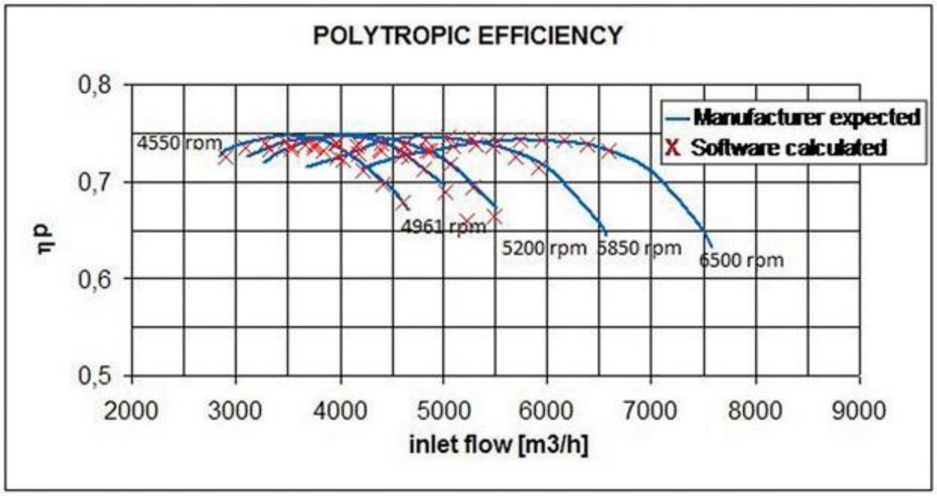


Figure 28: Sample compressor efficiency vs flow rate chart

4.8. Discharge pressure trend

Observing the performance chart obtained from our code we see the following trends.

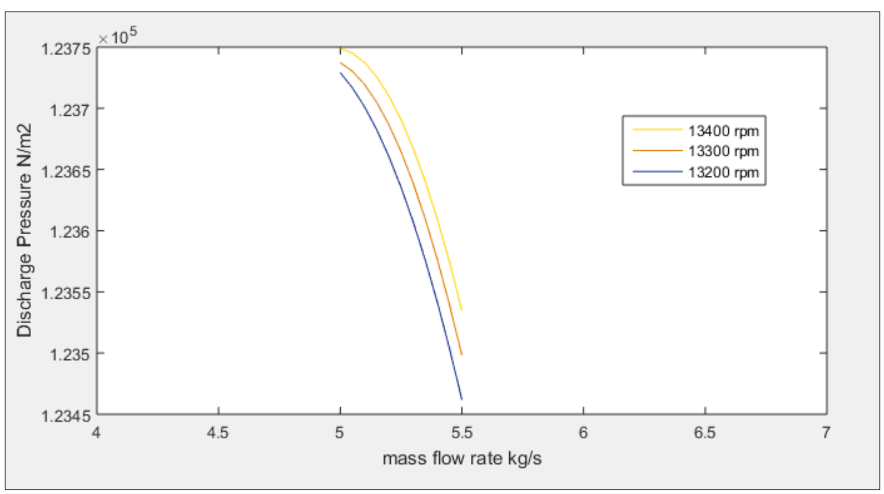


Figure 29: Discharge pressure vs flow rate chart

- The mass flow rate is increased at the same RPM the pressure is decreased.

- As RPM is increased for a given flow rate the pressure seems to get increased.

Basically, increasing the discharge pressure causes the effect of raising the weight of the compressed air stream throughout the stage which will result in less flow of air often considering the same power input. On the other hand, lowering the pressure will often allow more flow.

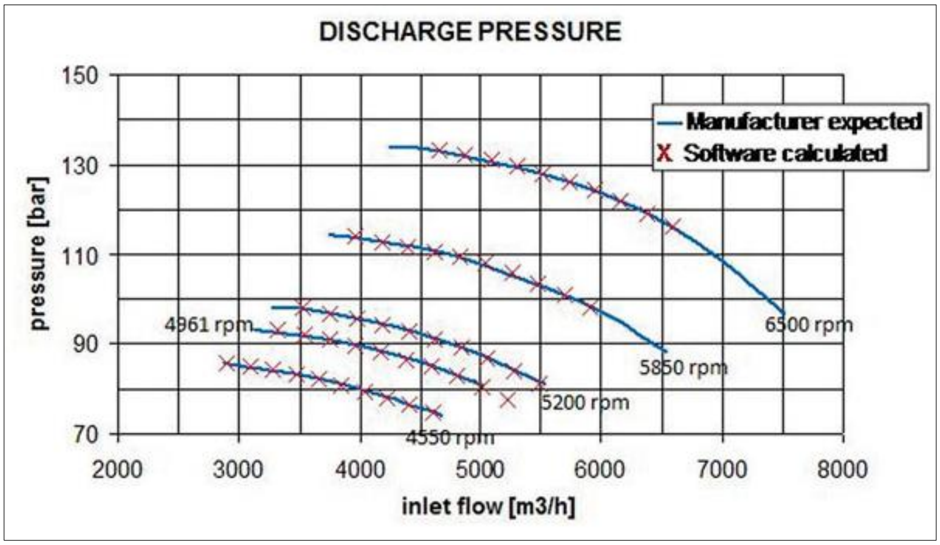


Figure 30: Sample pressure vs flow rate chart

4.9. Effect of other parameters

After analyzing compressor performance through performance charts and parameters we analyzed the effect of various other parameters and below are some of the results which were obtained:

- Changing the angle α_2 we observe that the best value of efficiency is obtained within the range we had prescribed i.e., from 60 to 70 degrees. Deviating from this range the value of efficiency dropped.

- Increasing the inlet temperature and reducing the inlet pressure results in reducing air density. As a result, less air is to be compressed hence power requirements were found to be decreased.

4.10. Validation Techniques

Since we are using an inverse design approach with certain parameters depending on the choice of the best possible outcome. Hence direct comparison with the real-time results available in the literature was not possible. One way to validate the results obtained was through various relations available in literature among parameters or various graphs or charts as mentioned during performance parameters section results. One better approach was to make use of all the known parameters and verify the results in the form of performance parameters or curves using a credible source/software. Since direct designing was beyond the scope of our project and would take additional time. We made use of credible software available online. We had few options like CMAP, CTrend, NASA code but it was only available in Fortran written back in the 1970s. Also, it was limited to specific design calculations. We choose CTrend for our validations of results as it has a user-friendly interface.

The validation of results was done using CTrend software used for performance analysis of centrifugal compressors ranging from monitoring and detection of performance. It employs artificial neural networks to predict performance.

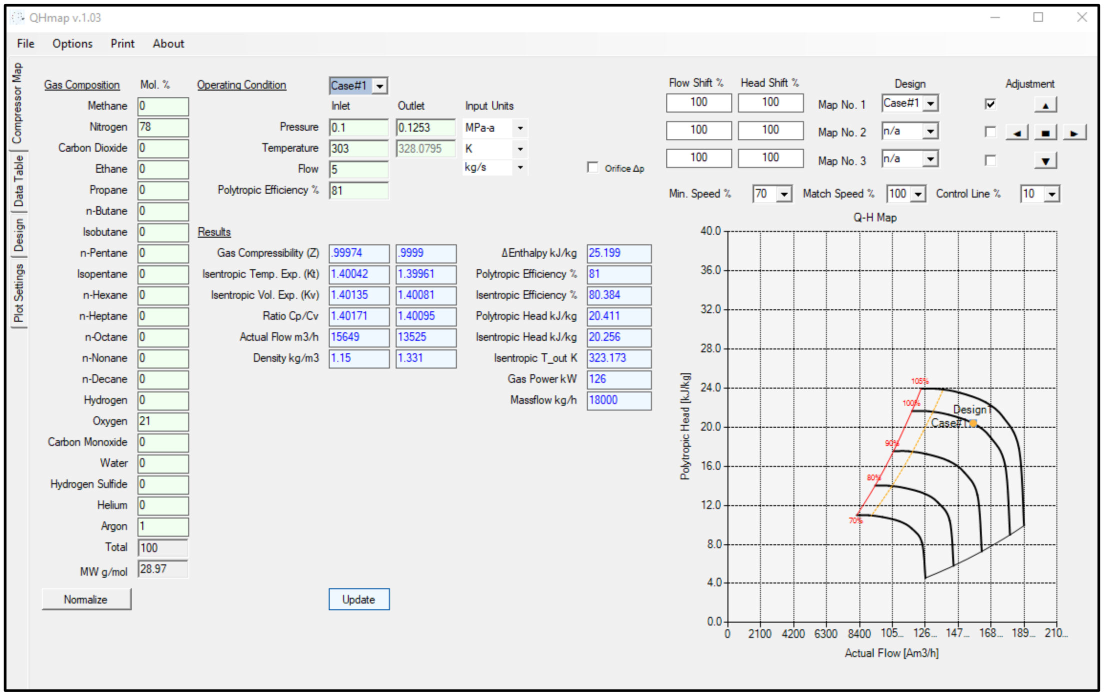


Figure 31: CTrend software and results

Below here is just the comparison of our design point condition using both our code and software and we can see that the trend is the same along with the value. Only the graph obtained from MATLAB is magnified a bit more than that obtained by software. The magnitude of the values obtained are discussed below in the table:

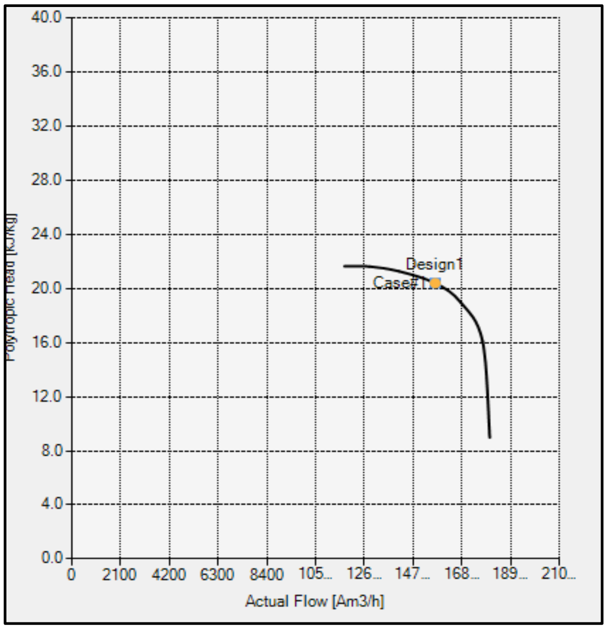


Figure 33: Isentropic work vs volume flow rate graph – Ctrend

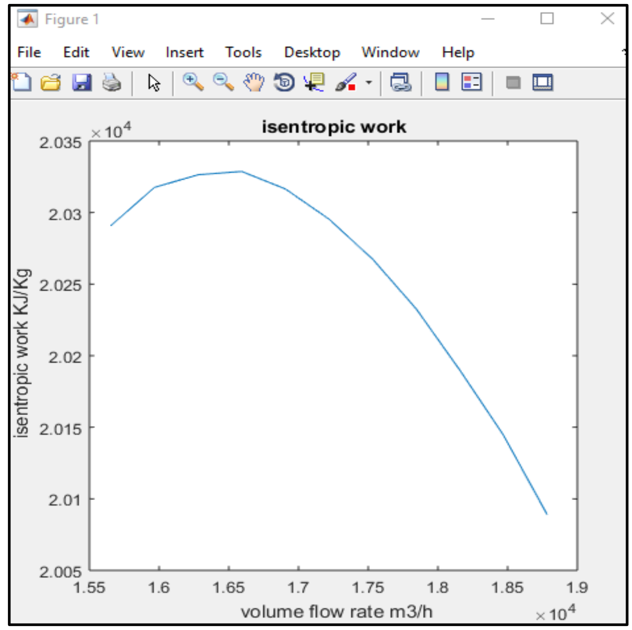


Figure 32: Isentropic work vs volume flow rate graph – our code

The other parameters calculated via our code and this Ctrend software are listed below for comparison in tabulated form. The reader can easily observe the accuracy of our code. The minor variations are due to certain assumptions taken for example the selection of loss models.

Table 2: Comparison of Ctrend and our code results

Parameters	**CTrend**	**Our code**
isentropic head	20.411 (kJ/kg)	20.291(KJ/kg)
outlet temperature	324.173(k)	326.53(k)
overall enthalpy change	25.199 (kJ/kg)	25.058(kJ/kg)

4.11. CFD Results:

After running the CFD setup at the settings described in the methodology section. We obtained some results which when compared with our software results show that they are proximity of each other, and the error is minimal. The first parameter is a significant one which is pressure. Considering that the geometry we input on our CFD analysis was able to achieve closer value of pressure as to our code generated value. It shows that the geometry that was given out by our code according to the RPM and the working fluid conditions, was suitable.

In context, the pressure ratio given by the user in the program was 1.25, giving an output pressure of 125kPa, the maximum value of output pressure obtained through our CFD analysis was 1.257 which again shows that the program outputs the appropriate geometry.

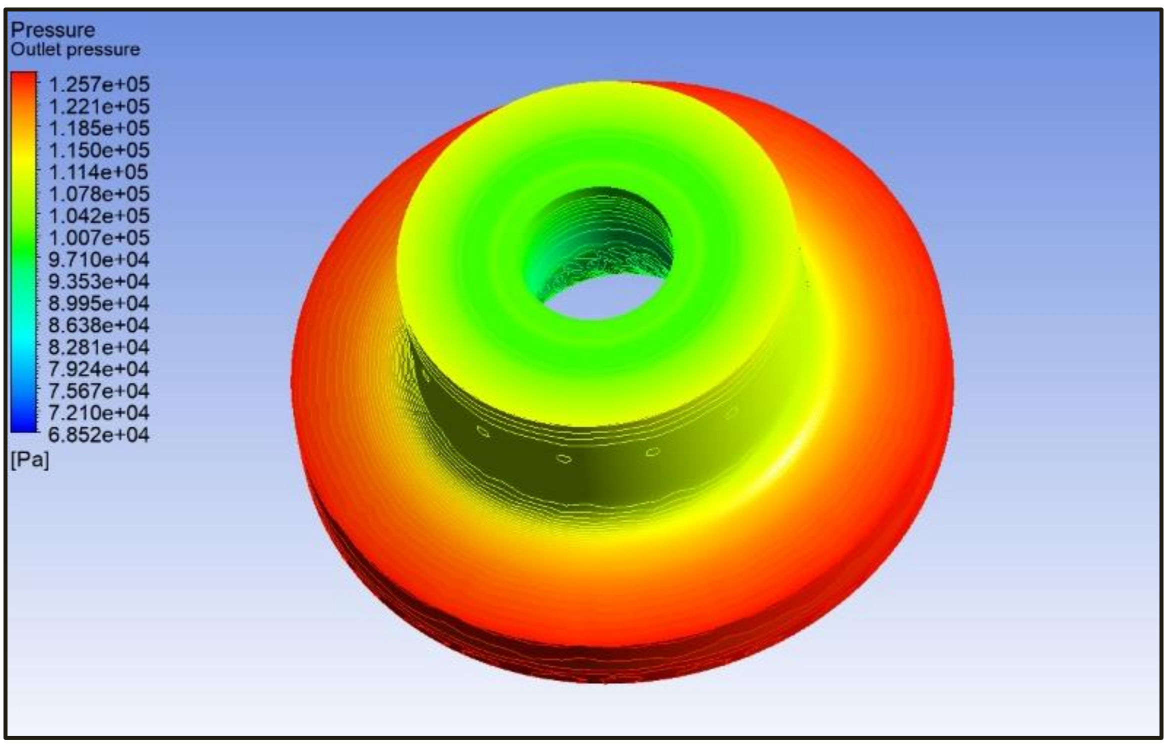


Figure 34: Static Pressure Profile of Impeller

Moving onto other parameters, we also compared static temperature values and the fluid velocity at the outlet. Here on comparison with the static temperature values we observe that the contour generated has a maximum value for 345.8K amounting to mean value of 342.2K, whereas the code generated value was 326K, giving us a difference of the code value from the CFD results of about 4.73%

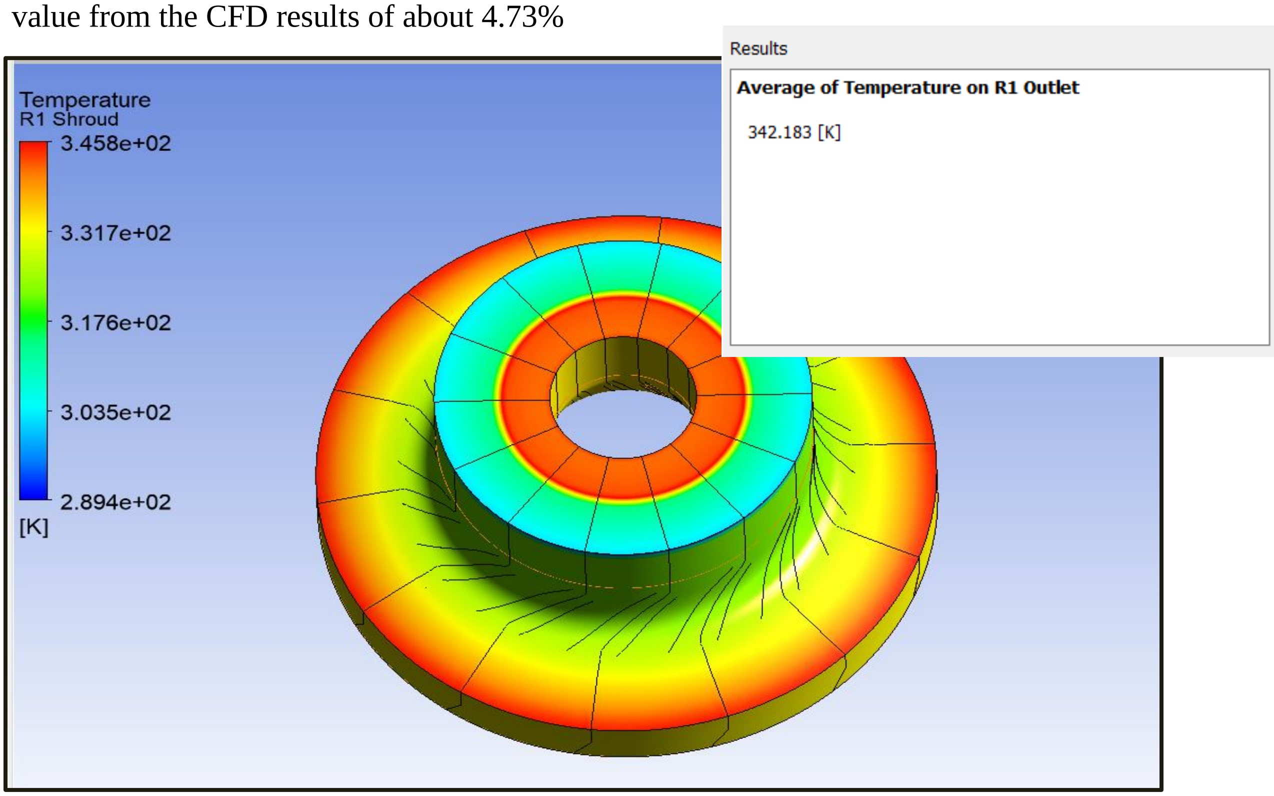


Figure 35: Static Temperature Profile of Impeller

Now comparing the velocity results contours were obtained, which showed a velocity profile we see that average value obtained at the outlet was 113.78 ms^{-1} whereas the code generated fluid velocity was 115 ms^{-1}, giving us a difference of about 1.08% from the CFD results.

From all these contours we observe that our code generated values are validated from our CFD plots and contours.

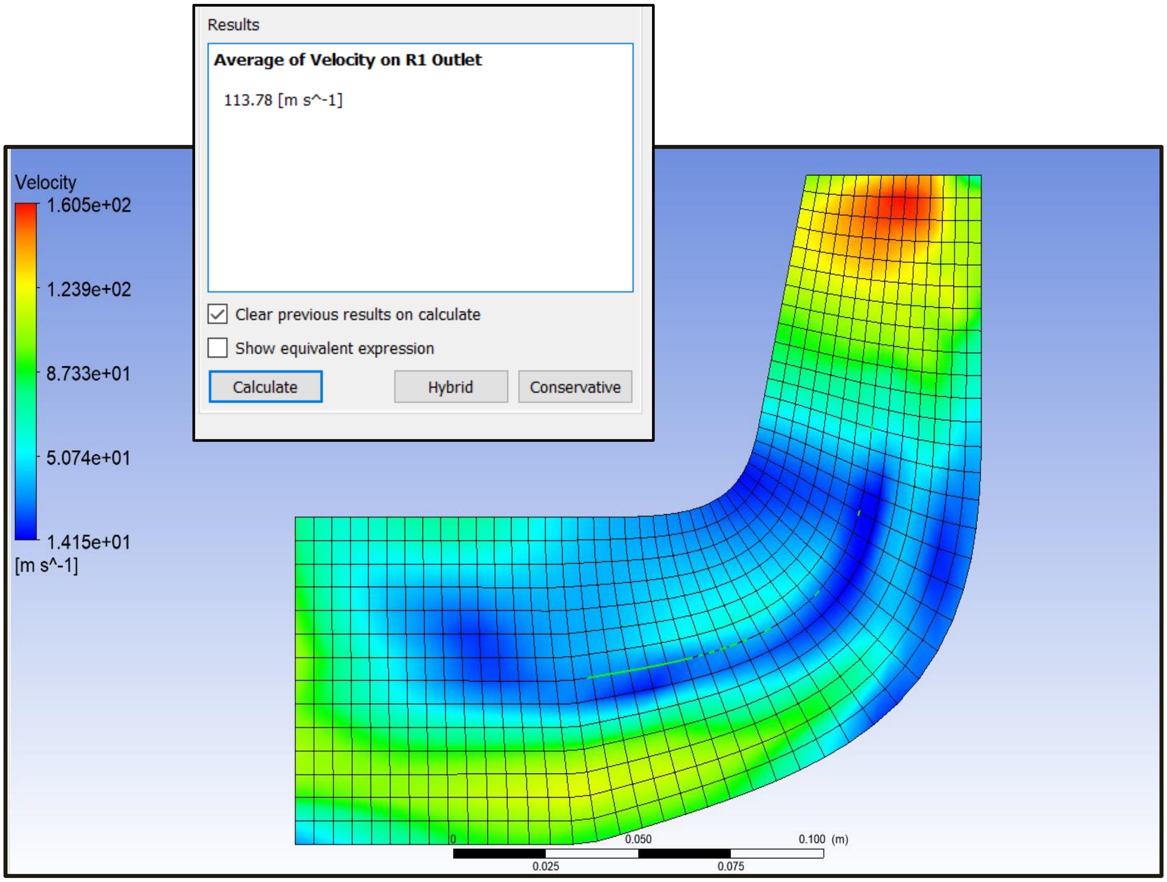


Figure 36: Fluid Outlet Velocity

CHAPTER 5: CONCLUSION AND RECOMMENDATION

In the initial section of this report, we stressed on importance of having such a program for predicting compressor performance because compression energy for up to 60% of entire plant operational costs. So, ensuring that the designed compressor works as per our desire is important. We made an effort towards finding solution of it and we were successfully able to integrate various concepts and develop a universally applicable solution for best possible performance prediction of a centrifugal compressor given a few non-geometric parameters of a centrifugal compressor. The previous chapters of this report have explained the design of the centrifugal compressor in detail along with calculations and analysis. From Methodology Chapter it is evident that our code of centrifugal compressor prediction code is fulfilling the design and functionality requirements that were expected from it. Whereas, in the result section, various outputs whether values or graphs were presented and finally the correctness of our final code was proved by the CFD analysis we performed and the comparison which we drew between our code and CTrend.

5.1. Major findings

Since we designed a general-purpose app for performance prediction and not a single product. So, we had a number of parameters to analyze and discuss their effects on compressor performance. Few important such parameters are discussed here. Keeping mass flow rate constant as rpm’s is increased end to end efficiency and pressure ratio are increased. However, the work done by the compressor increases too. This increase is up till a certain limit set by Cordier criteria going out the limits of specific speed and diameter will impact performance of compressor. Similar results were observed for mass flow rate.

The issue of supersonic flow never occurred while testing out our code because we have found such value D_{tip} at which relative inlet speed hence Mach number is minimized. The issue of supersonic flow is only important to be considered here at the inlet because after that due to compression the temperature of fluid increases. As, flow angle α_2 is increased we see a similar trend as we saw for mass flow rate and rpm’s. However, there is a limit to which we can increase it based on the geometry and manufacturing constraints. Degree of reaction is decreased as mass flow rate and rpm are increased. As mass flow rate is increased the discharge temperature is decreased and discharge pressure is increased. As rpm are increased discharge temperature is increased and discharge pressure

5.2. Application and Future prospects

The initial target audience of our code is mechanical engineering student community studying compressors in their fluid or gas turbine course. The program will augment their learning and they will be able to analyze the effects of varying various parameters while keeping others as constants. The plotting feature will also be helpful in catching their attention. The future target are compressor manufacturers. Once we can further test it and improve it after getting feedback from various users of our code.

5.3. Recommendation

Including two zone analysis is further recommended. There is a hybrid modelling technique which employs one-dimensional analysis as well as some features of two-dimensional i.e., jet and wake zone analysis. The results are more accurate. This hybrid technique was proposed only recently. The two-dimensional analysis helps to cover end wall losses which remain unexplained during one-dimensional 1D technique.

Stress analysis in software to validate. It is recommended to perform more in-depth stress analysis. In our project we have just performed a rudimentary stress analysis using some basic physics equations to just get an idea about stresses generated. To accurately predict stress, it is advised to include vibrational stress and complex dynamic stresses relations which are not possible to solve directly as a result finite element analysis is performed on proposed geometry. So, it is advised to perform such detailed stress analysis on one of the design points which our software gives. The program can be made more flexible to include other components of a centrifugal compressor for example catering for inducer and IGVs presence instead of just assuming that flow at the inlet is axial.

To improve the applicability of the program it is recommended to provide users with multiple working fluid and compressor material options with a fully verified database and give users an option to input their own values. Also, options for other diffuser types can be added as well to improve code applicability for example options like wedge diffusers, low solidity diffusers can be added. Additionally, although results of our code/software are verified from CFD and Ctrend software It is still suggested to do further testing of this app prototype before launching the software for commercial use.

REFERENCES

[1] S.L Dixon, C.A Hall (2010), Fluid Mechanics, Thermodynamics of Turbomachinery, 6th edition

[2] Donald H. Aungier, (2000), Centrifugal Compressors: A Strategy for Aerodynamic Design and Analysis, ASME Press

[3] S.M Yahya, (2011), Turbine’s compressors and fans, 4th edition

[4] Johannes Schiff, (2013), A Preliminary Design Tool for Radial Compressors, Lund University

[5] A Whitfield, (2020), Preliminary design and performance prediction techniques for centrifugal compressors

[6] En Sup Yoon and Chung, (1993), An optimal set of loss models for performance prediction of centrifugal compressors

[7] Sandburg Mark R, (2016), Centrifugal Compressor Configuration, Selection and Arrangement: A User's Perspective

[8] Hopkinson, E. and Chorlton, A.E.L., 1912: The Evolution and Present Development of the Turbine Pump, Engineer, 93,112, 1912

APPENDIX I: DESIGN POINT

Table 3: Design Point

Parameter	**Value**
Mass Flow Rate (kg/s)	5
RPM (rev/min)	15000
Tip Clearance (m)	2e-05
Surface roughness (m)	0.0002
Efficiency	80%
Pressure ratio	1.25

APPENDIX II: MATLAB CODE

Main file

```
clear all

%% constants
R=287;                          %universal gas constant
cp=1005;                        %specific heat
gamma=1.4;                      %specific heat ratio
k=((gamma-1)/(gamma));          %for convenience
ki=1/k;                         %inverse of K
itrmx=5;                        %max iterations
tol=0.01;                       %tolerance
eps=0.000500;                   %tip clearance
rgh=0.000025 ;                  %surface roughness

%% inputs
p_r=1.5;                       % pressure ratio needed
eta_tt=0.8;                    % end to end efficiecny
rpm=0;                         % rpm
m_dot=0;                       % mass flow rate
pt_1=100000;                   % inlet stagnation pressure
Tt_1=303;                      % inlet stagnation temperature
rho0=(pt_1/(R*Tt_1));          % inlet stagnation density
D1_hub=0.1;                    % hub diameter
mat = 'Steel.txt';             % Compressor material
th=0.002;                      % blade thickness

%% inital settings
e=[]
x=[]                           % storage variables
y=[]
z=[]
a=[]
b=[]
c=[]
d=[]
v_empt=[]
P_P=[]
num_point=1;                  % number of points between max and min mass
flow for plotting
m_max=6                        % max mass flow rate
m_min=5
m_gap=1;
```

```
m_dot=m_min:m_gap:m_max;
numb=numel(m_dot)           % number of elements in m_dot
rpm_max=16000;
rpm_min=15000;
rpm_gap=1000;
rpm=rpm_min:rpm_gap:rpm_max;       % rpm range
numbo=numel(rpm)

 %% loops
for j=1:1:numbo             % rpm loop
   w_isn  = cp * Tt_1 * ((p_r ^ k) - 1);        %isentropic or max work
   w=rpm(j)*((2*pi)/60);
   for i=1:1:numb           % mass flow rate loop
   V_dot=m_dot(i)/(rho0);
   Ns=((w)*((V_dot).^1/2))/(w_isn.^(3/4));  % specifc speed
   if Ns==[0.5,2]

 m_dot=m_dot;
 rpm=rpm;
   else

       if Ns<0.5 || Ns>2
  sprintf('the values of Ns is out of range')
  return;
end
   end

   Ds=3.222708798 -(2.144495533*log(Ns));   % specifc diameter
   D2  = (Ds*sqrt(V_dot))/(((w_isn))^(1/4));  % Impeller outer tip dia

   % now check this with the given size constraint

   U2=w*D2/2;                 % velocity
   w_non_isn=w_isn/eta_tt;    % non_isn_work

   % if hub diamter was not in input


      if D1_hub ~= 0
        D1_hub = D1_hub;
    else
        [Dmax,sf] = max_diameter(mat,w,Dhub,D2);    % [m] "Max" tip
diameter
        D1.hub    = 0.2*Dmax;
      end

 %% outer efficiency loop

    for itr = 1:itrmx
```

```
    D1_tip = tip_diameter(m_dot(i), rho0, D1_hub, D2, w);

    D1_mid=((D1_hub+D1_tip)/2);

    result(i,j) = inlet__loop(rho0, D1_hub, D1_tip, itrmx,
tol,m_dot(i), cp, R, k, pt_1, Tt_1,D2,w,gamma);
    if  result(i,j).T1>273   % T1 constraint otherwise impossible
results
    r=[ result(i,j).M1];      % Storing Ma at inlet
    x=[x,m_dot(i),r]          % storing r in an array
    elseif result(i,j).T1<273
    break;
    end




    inlet(i,j) = inlet__calc(result(i,j), w,pt_1, Tt_1, cp, R, k,
D1_tip, D1_mid, D1_hub,gamma);
    t=[inlet(i,j).T_mid];    %-do-

    alpha_2=60;             %input or constant outlet flow angle

    outlet(i,j) = outlet__calc(alpha_2, w_non_isn, U2, R, cp, Tt_1,
k,gamma);
    u=[outlet(i,j).T2];      % -do-

    outlet2(i,j) = outlet__loop(inlet(i,j), outlet(i,j), w_non_isn,
itrmx, tol, eps,
Tt_1,cp,R,ki,m_dot(i),D2,D1_mid,D1_hub,D1_tip,rgh,gamma,pt_1);


    beta_geo(i,j) = inlet__angle(outlet2(i,j).Nb,th,inlet(i,j),
D1_hub,D1_mid,D1_tip);

    vaneless(i,j) =
vaneless_diffuser_calcs(itrmx,tol,outlet2(i,j).rho2,outlet(i,j).V2,outl
et2(i,j).b2,D2,outlet(i,j).Tt_2,outlet(i,j).T2,outlet2(i,j).P2,outlet2(
i,j).muw,m_dot(i),cp,R,gamma);

    vaned(i,j) =
vaned_diffuser_calcs(vaneless(i,j).b3,vaneless(i,j).TT3,vaneless(i,j).T
3,vaneless(i,j).P3,vaneless(i,j).PT3,R,k,cp,gamma,pt_1,Tt_1,w_non_isn);

    v=[vaned(i,j).P4];  % -do-
    l=[vaned(i,j).eta_tt];
    o=[vaned(i,j).T4];
    y=[y,v]
    z=[z,l]
    e=[w,o]
```

```
    D4 = diff_diameter(vaneless(i,j).alp, vaneless(i,j).D3,
vaned(i,j).theta, vaned(i,j).W,vaned(i,j).LWR)
    S=Stress(mat,w,D1_hub,D2) % stress related file

     %% [G]: Performance parameters
        p_c = ((w_non_isn)/(U2^(2)));
%pressure coefficient denoted by si
        %h_c=  ((2)*(w_non_isn)/(U2^(2)));      engineering notes   mei
ye deine p_c mei isn w but normelly same                         % si r
head coefficient graphs wagera basically iske hi
        fprintf('\n pressure or head coefficient is \n',w_isn)
        %f_c= (V_dot)/(U2*((D2/2)^(2)))
%flow coefficient
        %fprintf('\n flow coefficient is \n',f_c)
        f_cc=(outlet(i,j).V2.rad)/(U2);
% another way to calculate flow coefficient
          fprintf('\n flow coefficient is \n',f_cc)
        D_R_1= 0.5 + 0.5*f_cc*cotd((90+outlet(i,j).W2.ang));
%flow coefficient
        D_R_2= 1-(0.5*((outlet(i,j).V2.tan)/U2))
        fprintf('\n degree of reaction \n',D_R_2)

%degree of reaction

        p_c_2=(outlet(i,j).V2.tan)/(U2);
%work coefficient same as head coeffcient verified should be same as
pressure
        p_c_3=1-(f_cc*cotd((90+outlet(i,j).W2.ang)))


     %%  Main interation

     fprintf('Main Iteration: %d\n', itr)

    %[E]:Calculate Residual & Check Convergence
     RES = abs(eta_tt - vaned(i,j).eta_tt) / eta_tt;
     fprintf('Main Residual: %f\n\n', RES)

     if RES < tol
        fprintf('\nMain converged in %d iterations\n', itr)
        break
        elseif itr == itrmx
        fprintf('\nMax iterations reached\n')
     end
     %[F]:Reset Efficiency & Iterate
        eta_tt = vaned(i,j).eta_tt;
    end


  B_i(j,i)= v;            % -do-
```

```
  C_i(j,i)=l;
  D_i(j,i)=o;
  m_i(i)= m_dot(i);

   end

  % if j~=1
  figure(1)               %pressure plot
  x_xx=m_i
  y_yy=B_i(j,:)
  p=polyfit(x_xx,y_yy,2) % introducing curves using iterpolation
  y_1=polyval(p,x_xx)
  plot(x_xx,y_1)
  xlabel('mass flow rate (kg/s)')
  ylabel('discharge pressure (Pa)')
  xlim([m_min-4 m_max+4])
  hold on


  figure(2)                %efficiency plot
  x_xx=m_i
  y_yy=C_i(j,:)
  p=polyfit(x_xx,y_yy,2)
  y_1=polyval(p,x_xx)
  plot(x_xx,y_1)
  xlabel('mass flow rate (kg/s)')
  ylabel('polytropic efficiency')
  xlim([m_min-4 m_max+4])
  hold on
   %end

  figure(3)                %temperature plot
  x_xx=m_i
  y_yy=D_i(j,:)
  p=polyfit(x_xx,y_yy,2)
  y_1=polyval(p,x_xx)
  plot(x_xx,y_1)
    xlabel('mass flow rate (kg/s) ')
  ylabel(' discharge temperature (K)')
  xlim([m_min-4 m_max+4])
  hold on
  %end

end
```

<u>Inlet-calc</u>

```
function inlet = inlet__calc(inlet, w,pt_1, Tt_1, cp, R, k, D1_tip,
D1_mid, D1_hub,gamma)
    %% []: Extract Structures for Ease
    V1 = inlet.V1;

    %% [E]:Remaining Inlet Quantites
    %% [E.2]:Velocity Components, Magnitude, & Angle
    % We have calculated V1 @ the tip. Then assuming a free vortex
    %   method, the hub and tip velocity triangles can be
    %   caluclated.
    %% [E.2.1]:Translational Velocity
    U1.tip = w * D1_tip/2;                          % [m/s]
    U1.mid = w * D1_mid/2;                          % [m/s]
    U1.hub = w * D1_hub/2;                          % [m/s]

    % Relative Velocity Components & Magnitude
    W1.hub.tan = inlet.V1.mid.tan - U1.hub; W1.hub.axl =
inlet.V1.mid.axl;    % [m/s]
    W1.hub.mag = sqrt(W1.hub.tan^2 + W1.hub.axl^2);    % [m/s]

    W1.mid.tan = inlet.V1.mid.tan - U1.mid; W1.mid.axl =
inlet.V1.mid.axl;    % [m/s]
    W1.mid.mag = sqrt(W1.mid.tan^2 + W1.mid.axl^2);    % [m/s]

    W1.tip.tan = inlet.V1.mid.tan - U1.tip; W1.tip.axl =
inlet.V1.mid.axl;    % [m/s]
    W1.tip.mag = sqrt(W1.tip.tan^2 + W1.tip.axl^2);    % [m/s]

    % Relative Velocity Angle
    %  The relative velocity angle is measured from the axial
direction. Simply
    %  use MATLAB's function atand().
    W1.hub.ang = atand(W1.hub.tan/W1.hub.axl);    % [deg]
    W1.mid.ang = atand(W1.mid.tan/W1.mid.axl);    % [deg]
    W1.tip.ang = atand(W1.tip.tan/W1.tip.axl);    % [deg]

    %% []:Mach Numbers
    a1 = sqrt(gamma * R * inlet.T1); % [m/s]: Speed of sound

    % Absolute Mach Number
    M1.mid = V1.mid.mag / a1;

    % Relative Mach Number
    M1w.hub = W1.hub.mag / a1;
    M1w.mid = W1.mid.mag / a1;
    M1w.tip = W1.tip.mag / a1;

    %% []:Temperature at Midspan
    T_mid = Tt_1 - V1.mid.mag ^ 2 / (2*cp);

    %% [I]:Output
```

```
    inlet.a1   = a1;        % [m/s] Speed of sound
    inlet.V1   = V1;        % [m/s] Absolute velocity
    inlet.M1   = M1;        % []    Absolute Mach number
    inlet.W1   = W1;        % [m/s] Relative velocity
    inlet.M1w  = M1w;       % []    Relative Mach number
    inlet.U1   = U1;        % []    Peripheral speed

    inlet.T_mid  = T_mid;        % [K]   Midspan temperature
    inlet.Tt_1 = Tt_1;        % [K]   Total temperature
    inlet.Pt_1 = pt_1;        % [Pa]  Total pressure


end
```

Tip Diameter

```
%% safi ullah
%{
 % This function takes initial design parameters and calculates the
inlet
 %  tip diameter minimizing the relative velocity function. This comes
 %  from the fact that for a centrifugal compressor we want to minimize
 %  the relative Mach number. MATLAB's built in optimization function
 %  fmincon is utilized.
 %
 %      The following are inputs:
 %
 %          mdot  : The inlet mass flow rate
 %          rho   : Inlet density
 %          Dhub  : Hub diameter
 %          w     : rotational speed
 %          Dlimit: Max diameter limit
 %
 %      The following are outputs:
 %
 %          Dtip: Inlet tip diameter
%}

function D1_tip = tip_diameter(m_dot, rho, D1_hub,D2, w)



    %% [A]:Define Function to be Minimized
    func = @(D1_tip) w^2 * D1_tip^2/4 + (m_dot / (rho * pi/4 *
(D1_tip^2 - D1_hub^2)))^2;

    %% [B]:Initial guess
    D0 = D1_hub + 0.001;
```

```
    %% [C]:Inequality Constraints
    A = [];
    b = [];

    %% [D]:Equality Constraints
    Aeq = [];
    beq = [];

    %% [E]:Upper & Lower bounds
    lb = 0.5 * D2;
    ub = 0.7 * D2;

    %% [F]:Solve for Minimum
    options = optimset('Display','off');
    D1_tip = fmincon(func, D0, A, b, Aeq, beq, lb, ub,[],options);

end
```

Inlet-Loop

```
% inlet loop function finding properties to be used in inlet calc
function result = inlet__loop(rho0, D1_hub, D1_tip, itrmx, tol,m_dot,
cp, R, k, pt_1, Tt_1,D2,w,gamma)
    %% []:Optimization Loop
    for itr = 1:itrmx
        if rho0>1
         D1_tip = tip_diameter(m_dot, rho0, D1_hub, D2, w);
        else
            break
        end
        %%  Dtip = tip_diameter(mdot, rho0, Dhub, D2, w);    % [m]
        %% [B]:Calculate Flow Inlet Area & Absolute Velocity
     sprintf('value of D1tip = %f',D1_tip)
        S1     = pi/4 * (D1_tip^2 - D1_hub^2)      % [m^2]
        V1.mag = m_dot / (rho0 * S1)         % [m/s]
        V1.ang = 0
        V1.axl = V1.mag
        V1.tan = 0

        %% [C]:Static Temperature
        T1 = Tt_1 - V1.mag^2 / (2 * cp) % [K]

        %% [D]:Mach number
```

```
        M1 = V1.mag/(sqrt(gamma*R*T1))      % []

        %% [E]:Static Pressure
        p1 = pt_1 / (1 + (gamma-1)/2 * M1.^2).^(gamma/(gamma-1)) % [Pa]
P1 = PT1 / (1 + (y-1)/2 * M1^2)^(y/(y-1));

        %% [F]:Recalculate Density
        rho = p1 / (R * T1)            %[kg/m^3]

        %% [G]:Check for Convergence
        RES = abs(rho - rho0) / rho0

        % if strcmp(monitor_residual,'yes') == 1
        %    fprintf('Iteration: %d | Residual %0.6f\n', itr, RES)
        % end

        if RES < tol
            fprintf('Minimization problem converged in %d iterations w/
residual: %0.6f\n', itr,RES)
            break
        elseif RES > 1e6
            fprintf('WARNING solution diverging')
            break
        end

        %% [H]:Reset/recalculated Density
        rho0 = rho

    end

    rho1 = p1 / (R * T1)


    %% [I]:Output
    result.Dtip    = D1_tip; % [m]   Tip diameter
    result.T1      = T1;    % [K]    Static Temperature
    result.M1      = M1;    % []     Absolute Mach number
    result.P1      = p1;    % [Pa]   Static Pressure
    result.V1.mid  = V1;    % [m/s]  Absolute velocity
    result.S1      = S1;    % [m^2]  Inlet flow area
    result.rho1    = rho1;

    end
```

Inlet-Angle

```
% number of blades in outlet loop
% blade thickness

function beta_geo = inlet__angle(Nb,th,inlet, D1_hub,D1_mid,D1_tip)

    D1 = [D1_hub,D1_mid,D1_tip];
    beta_flow = [inlet.W1.hub.ang,inlet.W1.mid.ang,inlet.W1.tip.ang];

    %% [A]:Inlet Area
    SAC = pi * D1 - Nb * th; % [m]

    %% [B]:Optimal Area
    S = pi * D1;  % [m]

    %% [C]:Calculate Geometric Inlet Angle
    beta_geo = atand( SAC/S * tand(beta_flow) );    % [deg]
    beta_geo = table2struct(array2table(beta_geo));
    beta_geo = cell2struct( struct2cell(beta_geo), {'hub', 'mid',
'tip'});


end
```

Outlet-Calc

```
%% safi ullah
%{
 % Recall that we do not know any of our flow angles at the exit of the
 %  impeller. So the outlet absolute flow angle is assumed
 %  to fix the velocity triangle.
 %
 % The following are inputs:
 %
 %          alpha2: Absolute outlet flow angle
 %          l_eul: Eulerian work
 %          U2: Outer peripheral speed
 %
 % The following are outputs
 %
 %          outlet: Structure containing velocity components and Non-
```

```
 %                   isentropic temperature
%}

function outlet = outlet__calc(alpha2, w_non_isn, U2, R, cp, Tt_1,
k,gamma)

    %% []:Velocity Triangles
    % Absolute Velocity
    V2.ang = alpha2;                    % [deg] Angle
    V2.tan = (w_non_isn) / U2;          % [m/s] Tangential component
uctheeta2 -1
    V2.mag = V2.tan / sind(alpha2);     % [m/s] Magnitude
    V2.rad = V2.mag * cosd(alpha2);     % [m/s] Radial component

    % Relative Velocity
    W2.tan = V2.tan - U2;               % [m/s] Tangential component
    W2.rad = V2.rad;                    % [m/s] Radial component
    W2.ang = atand(W2.tan / W2.rad);      % [deg] Angle
    W2.mag = sqrt(W2.tan^2 + W2.rad^2); % [m/s] Magnitude

    % Total & Static Temperature
    % These are the real values of the temperature considering a non-
    %   isentropic process. The irreversibility is included in the
    %   end to end efficiency assumed at the beginning of the
    %   calculations
    Tt_2 = Tt_1 + (w_non_isn) / cp;             % [K] Total temperature
    T2   = Tt_2 - V2.mag^2 / (2*cp);      % [K] Static temperature

    % Mach Numbers
    outlet.M2  = V2.mag / sqrt(gamma * R * T2);     % [] Absolute Mach
number
    outlet.M2u = U2 / sqrt(gamma * R * T2);         % [] Peripheral Mach
number
    outlet.M2w = W2.mag / sqrt(gamma * R * T2);     % [] Relative Mach
number

    %% []:Output
    outlet.U2   = U2;
    outlet.V2   = V2;
    outlet.W2   = W2;
    outlet.Tt_2 = Tt_2;
    outlet.T2   = T2;


end
```

Outlet-Loop

```
%% safi ullah
%{
 % This function calculates an (((((((((((((::outlet blade height and
density::))))))))))) that
 %  satisfies continuity by ((((((((((((((((((((((((::iterating on the
efficiency::)))))))))))))))))))))). Once the
 %  efficiency has converged, the function stops and returns the
 %  aforementioned values along with the thermodynamic properties at
the
 %  outlet. The ((((((((((((((((((::efficiency is corrected by
calculating various losses::)))))))))))))))))))))
 %
 % The following are inputs:
 %
 %          inlet : Structure containing inlet quantities
 %          outlet: Structure containing outlet quantities
 %          l_eul : Eulerian work (((((((((((((((((based on baseline
 efficiency))))))))))))))))))))))))))))))))))))
 %          itrmx : Max iterations
 %          tol   : Tolerance
 %
 % The following are outputs:
 %
 %          outlet: Structure containing all the thermofluid properties
at
 %                  the outlet
 %          beta1 : The geometic flow angles at the inlet
 %              Nb: Number of blades
%}

function outlet = outlet__loop(inlet, outlet, w_non_isn, itrmx, tol,
eps, Tt_1,cp,R,ki,m_dot,D2,D1_mid,D1_hub,D1_tip,rgh,gamma,pt_1)

    %% []:Initalize
    % Assume an isentropic process for the rotor to begin the iteration
    %   process. This process is to converge to the real pressure at
the
    %   outlet of the compressor------
    eta_0 = 1;


    U2    = outlet.U2;
    V2    = outlet.V2;
    W2    = outlet.W2;

    for itr = 1:itrmx

        %% [A]:Total & Static Temperature
```

```
        Tt_2is = Tt_1 + (w_non_isn) * eta_0 / cp;       % [K] Total
temperature
        T2is  = Tt_2is - V2.mag^2 / (2*cp);   % [K] Static temperature
        disp(T2is)

        % Mach Numbers
        M2is  = V2.mag / sqrt(gamma * R * T2is);   % [] Absolute Mach
number

        %% [B]:Isentropic Outlet Pressure
        P2  = inlet.P1 * (T2is / inlet.T1) ^ (ki);  % [Pa] Static
pressure
        PT2 = P2 * (1 + (gamma-1)/2 * M2is^2) ^ (ki); % [Pa] Total
pressure
        Bc    = PT2 / pt_1;


        %% [C]:Density & Blade Height
        rho2 = P2 / (R * T2is);                    % [kg/m^3]
             sprintf('the value of rho2 is :%f',rho2)
                  sprintf('the value of D2 is :%f',D2)
        b2   = m_dot / (rho2 * pi * D2 * V2.rad);% [m]
        sprintf('the value of b2 is :%f',b2)


        %% [D]:Check Stability
        rbD = b2 / (D2/2);                          % [] Blade to outlet
diameter ratio || should be less than 1


        %% [E]:Average Beta
        % To calculate the average flow deflection use the midspan
inlet
        %  relative angle
            sprintf('the value of W2.ang is :%f',W2.ang)
        sprintf('the value of W1.mid.ang is :%f',inlet.W1.mid.ang)
        beta_avg = (W2.ang + inlet.W1.mid.ang) / 2; % [deg]


        %% [F]:Number of Blades
        % Before continuing we define the solidity. Theory and practice
tell
        %  us to keep 1/solidity = 0.4. For the number of blades we
round up
        %  and add 1 blade to minimize the blade loading. The pitch and
chord
        %  are also calculated. its range - manufacturability - proven
        oi = 0.4;                                                % []
Inverse of solidity
        Nb = 2 * pi * cosd(beta_avg) / (oi * log(D2/D1_mid));   % []
Number of blades
```

```
        Nb = ceil(Nb) + 1; % []  No. of blades rounded up
        sprintf('the value of number of blades is :%f',Nb)
        s  = pi * D2 / Nb;                                       % [m]
Pitch
        % ch = s / oi;                                           % [m]
Chord


        %% [G]:Slip Factor & Freestream Velocity
        muslip = 1 - 0.63 * pi / Nb;             % Assumtion that it is
approximatley equal from stanitz original formula and proved also i.e.
1-phi2 cot beta2 ~~ 1
        V2.inf = (1 - muslip) * U2 + V2.tan;     % [m/s] Absolute
        W2.inf = V2.inf - U2;                    % [m/s] Relative


% slip factor check ???? confirm ??

        %% [H]:Geometric Outlet Angle
        beta2_geo = atand(W2.inf / W2.rad);


        %% [I]:Calculate Losses
        %% [I.1]:Geometric Inlet Angle
        % We first analyze the losses due to having a difference in the
        %  geometrical outlet angle and the fluid outlet angle. Here we
        %   need the thickness of our blade that is assumed for now.
        th    = 0.002;  % [m] Thickness of our blade
        beta1 = inlet__angle(Nb,th,inlet, D1_hub,D1_mid,D1_tip);
        in    = (beta1.hub - inlet.W1.hub.ang);
        dhin  = (inlet.W1.hub.mag * sind(in)) ^ 2 / 2;

         sprintf('the value of dhin is :%f',dhin)


        %% [I.2]:Tip Clearance Losses
        % From paper provided by Gaetani we found the following
relation to
        %   calculate tip losses. The tip clearance was also found via
        %   other papers to be roughly 2% of the exit blade height
        if eps == 0
            eps = 0.02 * b2;
        end
        dhcl = 0.6 * eps / b2 * V2.tan * sqrt( 4*pi/(b2*Nb) *
ceil((D1_tip^2/4 - D1_hub^2/4)/((D2/2 - D1_tip/2)*(1 + rho2 /
inlet.rho1))) * ...
            V2.tan * inlet.V1.mid.axl ...
        );

     sprintf('the value of dhcl is :%f',dhcl)
```

```
        %% [I.3]:Blade Losses
        sprintf('the value of cosbavg is :%f',cosd(beta_avg))
        L      = (((D2/2) - (D1_mid/2))) / cosd(beta_avg);            %
[m]     Hydraulic Length
        sprintf('the value of L is :%f',L)
        W1.avg = (inlet.W1.hub.mag + inlet.W1.mid.mag ...
               + inlet.W1.tip.mag)/3;                           % [m/s]
Average relative inlet velocity
       sprintf('the value of W1.avg is :%f',W1.avg)
           D      = 1 - W2.mag/W1.avg + (pi * D2 * V2.tan) / (2 * Nb *
L * W1.avg)+ 0.1 * (D1_tip/2 - D1_hub/2 + b2) / (D2/2 - D1_tip/2) * (1
+ W2.mag/W1.avg);  % formula ????????[]      Diffusion Factor
         sprintf('the value of D is :%f',D)
        dhdiff = 0.05 * D^2 * U2^2;                             %
[J/kgK] Change in enthalpy
         sprintf('the value of dhdiff is :%f',dhdiff)



        %% [I.4]:Frictional Losses
        % We first calculate our outlet perimeter and area. Remember
that the
        %   outlet of a centrifugal machine is rectangular. With these
two
        %   quantities we define our hydraulic diameter which will be
used to
        %   determine our flow regime via Reynolds number.
        S2    = pi * D2 * b2;                    % [m^2] Outlet flow
area
        Per2  = Nb * (2*b2 + 2*s);               % [m]   Outlet flow
perimeter
        D_hyd = 4 * S2 / Per2; % [m]   Hydraulic diameter
        sprintf('the value of Dhyd is :%f',D_hyd)
        sprintf('the value of P2 is :%f',P2)
        mu = thermodynamic_data(1,(T2is-273));
        muw=mu*(10^-(6));
        sprintf('the value of W2mag is :%f',W2.mag)
        Re    = rho2 * W2.mag * D_hyd / muw;     % []    Reynolds
number
        disp(Re)

        % Enthalpy increase
        rel_e  = rgh / D_hyd;                    % [] Relative roughness
        f = colebrook(Re,D_hyd,rgh)
        cfm=f;                                               %cfm    =
moody(Re,rel_e);               % [] Call moody diagram
        cf  = cfm + 0.0015;                   % [] Adjusted friction
coefficient
        dhfric = 4 * (cf * L * W2.mag^2) / (2*D_hyd);
         sprintf('the value of dhfric is :%f',dhfric)
```

```
        %% [J]:Calculate New Efficiency
        sumdh = dhdiff + dhfric + dhcl + dhin;  % [J/kgK] Sum of
enthalpy losses
             sprintf('the value of w non isn is :%f',w_non_isn)
                  sprintf('the value of sum dh is :%f',sumdh)
        eta_n = (w_non_isn - sumdh) / w_non_isn;        % []      New
efficiency
        RES   = abs(eta_n - eta_0)/eta_0;       % []      Residual

        % fprintf('Iteration: %d | Residual: %0.4f\n', itr, RES)

        if RES < tol
            fprintf('Outlet calculations converged in %d iterations w/
residual %0.6f\n', itr,RES)
            break
        end

        %% [K]:New Total Enthalpy Change & Eulerian Work
        his_n = w_non_isn * eta_n;          % [J/kg] New isentropic
work
        eta_0 = eta_n;                    % []     Set new efficiency to
old

    end

    %% [L]:Output
    outlet.beta2_geo = beta2_geo;   % [deg]     Geometrical flow angle
    outlet.T2is    = T2is;          % [K]       Static isentropic
temperature
    outlet.TT2is   = Tt_2is;         % [K]       Total isentropic
temperature
    outlet.P2      = P2;            % [Pa]      Static real pressure
    outlet.PT2     = PT2;           % [Pa]      Total real pressure
    outlet.eta     = eta_n;         % []        Final efficiency
    outlet.M2is    = M2is;          % []        Absolute isentropic
Mach number
    outlet.rbD     = rbD;           % []        Blade to diameter ratio
    outlet.b2      = b2;            % []        Blade height
    outlet.rho2    = rho2;          % [kg/m^3]  Density
    outlet.L       = L;             % [m]       Mean length
    outlet.his_n   = his_n;         % [J/kg]    New isentropic work
    outlet.dhdiff  = dhdiff;        % [J/kg]    Diffusion losses
    outlet.dhin    = dhin;          % [J/kg]    Incidence losses
    outlet.D       = D;             % van[]        Gaetani diffusion
factor
    outlet.S2      = S2;            % [m^2]     Exit area
    outlet.in      = in;            % [deg]     Incidence angle
    outlet.muslip  = muslip;        % []        Slip factor
    outlet.V2      = V2;
```

```
    outlet.W2      = W2;
    outlet.Nb=Nb;
     outlet.muw=muw;
     outlet.Bc=Bc;
    Re=Re;

end
```

Vaneless Diffuser

```
function result = vaneless_diffuser_calcs(itrmx,
tol,rho2,V2,b2,D2,Tt_2,T2,P2,muw,m_dot,cp,R,gamma)

    %% []:Grab Required Values From Design
    %rho2  = initial.outlet2.rho2;     % [kg/m^3] Outlet density
    %V2    = initial.outlet.V2;       % [m/s]    Outlet velocity
    %b2    = initial.outlet2.b2;         % [m]      Outlet blade height
    %D2    = initial.comp.D2;         % [m]      Outlet diameter
    %TT2   = initial.outlet.TT2;      % [K]      Outlet real total
temperature
    %T2    = initial.outlet.T2;       % [K]      Outlet real static
temperature
    %P2    = initial.outlet.P2;       % [Pa]     Outlet real pressure

    %% []:Calculate Vanless Diffuser Diameter
    % We assume a ratio between the outlet diameter and the vaneless
    %   diffuser. In addition, we assume that the blade height will
remain
    %   the same, i.e. b3 = b2. the range of ratio from 1.05-1.2
    D3D2 = 1.2;
%D3/D2
    D3   = D3D2 * D2;
    b3   = b2;



    %% []:Setup Density Loop
    % In this iterative loop, we initialize the average density and the
    %   average velocity with the compressor outlet conditions. As a
    %   result, set rho3 and V3 to rho2 and V2 respectively to begin
the
    %   optimization process. Note that since the diffuser is basically
a
    %   stator and no work is done. TT3 will equal TT2.
    TT3  = Tt_2;                                        % [K]
Total Temperature
    rho3 = rho2;                                        % [kg/m^3]
Density
```

```
    V3   = V2;                                               % [m/s]
Velocity
    Dhyd = (4 * pi * D3 * b3) / (2 * (pi * D3 + b3));    % [m]
Hydraulic Diameter         (4A/P) %?????????????

    for itr = 1:itrmx

        %% []:Calculate Average Quantities
        rho_avg = (rho2 + rho3) / 2            % [kg/m^3] Average
density
        V_avg   = (V3.mag + V2.mag) / 2        % [m/s]    Average
velocity
        Re_avg  = rho_avg * Dhyd * V_avg / muw  % []       Average
Reynolds number
        sprintf('the value of Reavg is :%f',Re_avg)


        %% []:Calculate Friction Coefficient
        k_f  = 0.02;                                % [] Experimental
constant
          sprintf('the value of kf is :%f',k_f)
        cf = k_f * (1.8 * 10^5 / Re_avg)         % [] Friction
coefficient

         sprintf('the value of cf vaneless is :%9.8f',cf)


        %% []:Vanless Diffuser Outlet Velocity


        den = (D3D2 + cf/2 * pi * rho2 * V2.tan * D3 * (D3-D2)/m_dot)
%denominator
        disp(rho3)
        disp(D3)
        disp(b3)
        V3.tan = V2.tan / den                      % [m/s] Tangential
component
        V3.rad = m_dot / (pi * D3 * b3 * rho3)   % [m/s] Radial
component
        V3.mag = sqrt(V3.tan^2 + V3.rad^2)       % [m/s] Outlet velocity
magnitude

        V3.ang = atand(V3.tan / V3.rad)          % [m/s] Outlet angle
        alp=V3.ang


        %% []:Thermodynamic Values
        T3 = TT3 - V3.mag^2 / (2*cp)
        M3 = V3.mag / sqrt(gamma*R*T3)
```

```
        %% []:Calculate Losses
        num = cf * D2/2 * (1 - (1/D3D2)^1.5) * V2.mag^2              %
numerator

        den = 1.5 * b2 * cosd(V2.ang)                               %
denominator

        dh  = num / den                                    % enthalpy
loss


        %% []:Calculate Isentropic Values
        TT3is = TT3 - dh/cp
        T3is  = TT3is - V3.mag^2 / (2*cp)
        PT3 = P2 * (TT3is/T2)^(gamma/(gamma-1))
        P3  = PT3 / (1 + (gamma-1)/2 * M3^2)^(gamma/(gamma-1))


        %% []:Calculate Outlet Density
        rho3_n = P3 / (R*T3)


        %% []:Calculate Residual
        RES = abs(rho3 - rho3_n) / rho3;
        % fprintf('Iteration: %d | Residual: %0.3f\n', itr, RES)

        if RES < tol
            fprintf('Vanless Diffuser calcs converged: Iterations: %d |
Final residual: %0.6f\n', itr, RES)
            break
        elseif itr == itrmx
            fprintf('Max iterations reached\n')
        end
        rho3 = rho3_n;

    end

    %% []:Output
    vaneless.D3    = D3;       % [m]      Outlet diameter
    vaneless.V3    = V3;       % [m/s]    Outlet velocity
    vaneless.PT3 = PT3;     % [Pa]     Isentropic total pressure
    vaneless.P3  = P3;      % [Pa]     Isentropic static pressure
    vaneless.TT3is = TT3is;    % [K]      Isentropic total temperature
    vaneless.TT3   = TT3;      % [K]      Real total temperature
    vaneless.T3    = T3;       % [K]      Real static temperature
    vaneless.T3is  = T3is;       % [K]      Real static temperature
    vaneless.cf    = cf;       % []       Friction coefficent
    vaneless.rho3  = rho3;     % [kg/m^3] Density
    vaneless.M3    = M3;       % []       Real absoulte Mach number
```

```
    vaneless.dh     = dh;        % [J/kg]   Enthalpy losses
    vaneless.b3     = b3;        % [m]      "Blade height"
    vaneless.alp     = alp;      %outlet angle ka dosra name

    result=vaneless;
end
```

Vaned Diffuser

```
%{
 % This function calculates the thermodynamic quantities of the wedge
 %  diffuser. Theory tells us that max efficiency occurs at a
divergence
 %  angle (2 times theta) between 8 and 10 degree ---- in book also
 %
 % The following is the input:
 %
 %      design: Current design with vanless diffuser
 %
 % The following is the output
 %
 %      result: Final design with wedge diffuser
%}

function result =
vaned_diffuser_calcs(b3,TT3,T3,P3,PT3,R,k,cp,gamma,pt_1,Tt_1,w_non_isn)


    %% []:Grab Required Values From Design
    %b3     = design.vldiff.b3;       % [m]      Outlet blade height
    %TT3    = design.vldiff.TT3;      % [K]      Outlet real total
temperature
    %T3     = design.vldiff.T3;       % [K]      Outlet real static
temperature
    %P3     = design.vldiff.P3;     % [Pa]     Outlet real pressure
    %PT3    = design.vldiff.PT3;    % [Pa]     Outlet total pressure


    %% []:Choose Values
    % We start the diffuser design process by choosing certain
geometric
    %   parameters. Aungier, in his book, has a table of various
diffuser
    %   specifications. For packaging's sake, the smallest length to
width
```

```
    %   ratio was chosen with a divergence angle between 8 and 10
degrees.
    %   In addition, the coefficient of pressure recovery is given for
    %   this diffuser. The aspect ratio was taken to be 1 for
simplicity.
    LWR   = 8.43;       % []    Length to width ratio for the vaned
diffuser
    theta = 9.49 / 2;   % [deg] Half total divergence angle
    AS    = 1;          % []    Channel aspect ratio
    prc   = 0.62;       % []    Pressure recovery coefficient
    eta_d = 0.87;       % []    Diffuser efficiency corresponding to a
2theta = 8 [deg]
    W = AS * b3;        % diffuser channel width


    %% []:Thermodynamic Values
    % Because the diffuser is a stationary flow passage there is no
work
    %   done to the fluid. Thus, the total temperature remains constant
    %   resulting in:
    %
    %       h03 = h04
    %       TT3 = TT4
    TT4  = TT3;                         % [K]  Total temperature
    P4   = prc * (PT3 - P3) + P3;       % [Pa] Static pressure
    T4is = T3 * (P4 / P3) ^ k;          % [K]  Isentropic static
temperature
    T4   = T3 + (T4is - T3) / eta_d;    % [K]  Real static temperature
    rho4 = P4 / (R * T4);              % [kg/m^3] Density


    %% [.1]:Losses
    % Looking at a Mollier diagram for points 3 to 4, we notice that
the
    %   difference in enthalpy is given by the real value minus the
    %   isentropic one. Thus, our enthalpy loss is given by:
    %
    %       dhloss = h4 - h4is
    % -OR-
    %       dhloss = cp(T4 - T4is)
    dhloss = cp * (T4 - T4is);          % [J/kg] Enthaply drop


    %% [.2]:Continue with remaining thermodynamic values
    TT4is = TT4 - dhloss/cp;            % [K]  Isentropic total
temperature
    PT4   = P4 * (TT4is/T4)^(gamma/(gamma-1));  % [Pa] Total pressure


    %% []:Velocity
```

```
    % Velocity at the outlet can be derived from the total temperature.
The
    %   total temperature is the sum of the static temperature plus
    %   V^2/(2cp)
    V4 = sqrt(2 * cp * (TT4 - T4));
    M4 = V4 / sqrt(gamma * R * T4);


    %% []:Final End to End Efficiency
    % Calculate total isentropic enthalpy change [J/kg], then divide by
the
    %   Eulerian work of our compressor
    Be     = PT4 / pt_1;
    htis   = cp * Tt_1 * (Be ^ k - 1);
    eta_tt = htis / w_non_isn;

    fprintf('End to end pressure ratio: %0.3f | End to end efficiency:
%0.3f\n', Be, eta_tt)

    %% []:Output
    vaned.T4      = T4;              % [K]      Real static temperature
    vaned.T4is    = T4is;            % [K]      Isentropic static
temperature
    vaned.TT4     = TT4;             % [K]      Real total temperature
    vaned.TT4is   = TT4is;           % [K]      Isentropic total
tempertature
    vaned.P4      = P4;              % [Pa]     Static pressure
    vaned.PT4     = PT4;             % [Pa]     Total pressure
    vaned.V4      = V4;              % [m/s]    Outlet velocity
    vaned.M4      = M4;              % []       Absolute Mach number
    vaned.rho4    = rho4;            % [kg/m^3] Density
    vaned.Nb      = 6;               % []       Number of diffuser
channels
    vaned.dhloss = dhloss;           % [J/kg]   Enthalpy loss
    vaned.htis    = htis;            % [J/kg]   End to end enthalpy change
    vaned.eta_tt = eta_tt;           % []       End to end efficiency
    vaned.LWR     = LWR;             % []       Diffuser channel length to
width ratio
    vaned.dvang  = 2 * theta;        % [deg]    Total divergence angle
    vaned.W       = AS * b3;         % [m]      Diffuser channel width
    vaned.Be      = Be;              % []       End to end pressure ratio
    vaned.eta_tt = eta_tt;               % []   End to end pressure ratio
    vaned.theta=theta;


    result = vaned;


end
```

Diffuser diameter

```
%{
 % This function calculates the vaned diffuser diameter. This function
is
 %  based on the assumption that the outlet angle of the vanless
diffuser
 %  is the angle of the suction side of the diffuser wedge.
%}

function result = diff_diameter(alp, D3, theta, W,LWR)

    %global alp r th w

    %% [A]:Givens
    % Grab the necessary values from the design
    %alp = design.vldiff.V3.ang;         % [deg] Vanless diffuser
outlet angle
    %r   = design.vldiff.D3 / 2;         % [m]   Vanless diffuser
radius
    %th  = design.diff.dvang / 2;        % [deg] Half the divergence
angle
    %w   = design.vldiff.b3;             % [m]   Diffuser channel width
    %LWR = design.diff.LWR;              % []    Diffuser channel
length to width ratio

    %% [B]:Solve Geometry
    % The known values provide a fully constrained problem.
    %       Equations                        Variables
    % -----------------------       -------------------
    % (1) Law of cosines                 x(1) = x
    % (2) Law of sines                   x(2) = y
    % (3) Law of sines                   x(3) = gamma
    % (4) Angles of triangle             x(4) = delta
    % (5) Angles of triangle             x(5) = beta
    % (6) Angles of line                 x(6) = epsilon

    o  = 90 - theta;
    f1 = @(x) x(1)^2 + x(2)^2 - 2*x(1)*x(2)*cosd(x(4)) - W^2;
    f2 = @(x) W * sind(x(3)) / sind(x(4)) - x(1);
    f3 = @(x) sind(x(5)) / sind(x(6)) * (D3/2) - x(2);
    f4 = @(x) x(3) + x(4) + o - 180;
    f5 = @(x) x(4) + x(6) + alp - 180;
    f6 = @(x) 2*x(6) + x(5) - 180;

    % System of equaitons
    fsys = @(x) [f1(x), f2(x), f3(x), f4(x), f5(x), f6(x)];
    x0 = ones(1,6);
```

```
    options = optimset('Display','off');
    x = fsolve(fsys,x0,options);

    %% [C]:Calculate Remaining Geometric Values
    d  = LWR * W;                   % [m] Diffuser channel length
    h  = d / cosd(theta);              % [m] Diffuser channel length
projected
    L  = x(1) + h;                  % [m] Total length of suction side
    cp = L * sind(alp);             % [m] Length suction side to radial
direction
    e  = cosd(alp) * L;             % [m] Length from cp to radial
direction

    R  = sqrt(cp^2 + ((D3/2)+e)^2);  % [m] Radius of vaned diffuser

    %% [D]:Output
    design.diff.D4 = 2*R;
    %design.diff.D2tD4 = design.diff.D4 / design.comp.D2;
    result = design;

end
```

Stress Analysis

```
function stresses = Stress(material,w,Dhub,D2)

    %% [A]:Import Material Properties
    props = read_file(material)
    rho    = props(1);                  % [kg/m^3] Density
    YS     = props(2);                  % [Pa]     Tensile yield strength
    % G      = props(3);                  % [Pa]     Shear yield strength
    nu     = props(4);                  % []       Poissons ratio


    %% []:Max diameter
    ri    = Dhub / 4;                                  % Shaft diameter
    A     = 8 * YS / ((3 + nu) * rho * w.^2);                %rotation
Xdlmao rblgcc
    B     = ri.^2 * (1 - (1 + 3*nu) / (3 + nu));
    Dmax = 2 * sqrt(0.5 * (A - B));
        fprintf('value of max diameter is  \n',Dmax)

    %% []:Disk Stress
    r2     = D2 / 2;      % Exit radius
```

```
    sig_h = (3 + nu) / 8 * rho * w ^ 2 * (2 * r2 ^ 2 + ...
            ri ^ 2 *(1 - (1 + 3*nu) / (3 + nu)));
    o_h   = sig_h / 1000 / 1000;                          % [MPa]
    fprintf('Hoop stress: %0.1f[MPa]\n', o_h)

    %% Safety Factors
    sf   = Dmax / D2;
    sf_o = YS / sig_h;
    fprintf('Outer diameter safety factor: %0.2f\n', sf)
    fprintf('Stress safety factor: %0.2f[]\n', sf_o)


  stresses.Dmax=Dmax;
     stresses.ri=ri;
      stresses.Dmax=Dmax;
         stresses.sig_h=sig_h;
         stresses.rho=rho;
          stresses.YS=YS;

end
```

APPENDIX III: APP PACKAGE

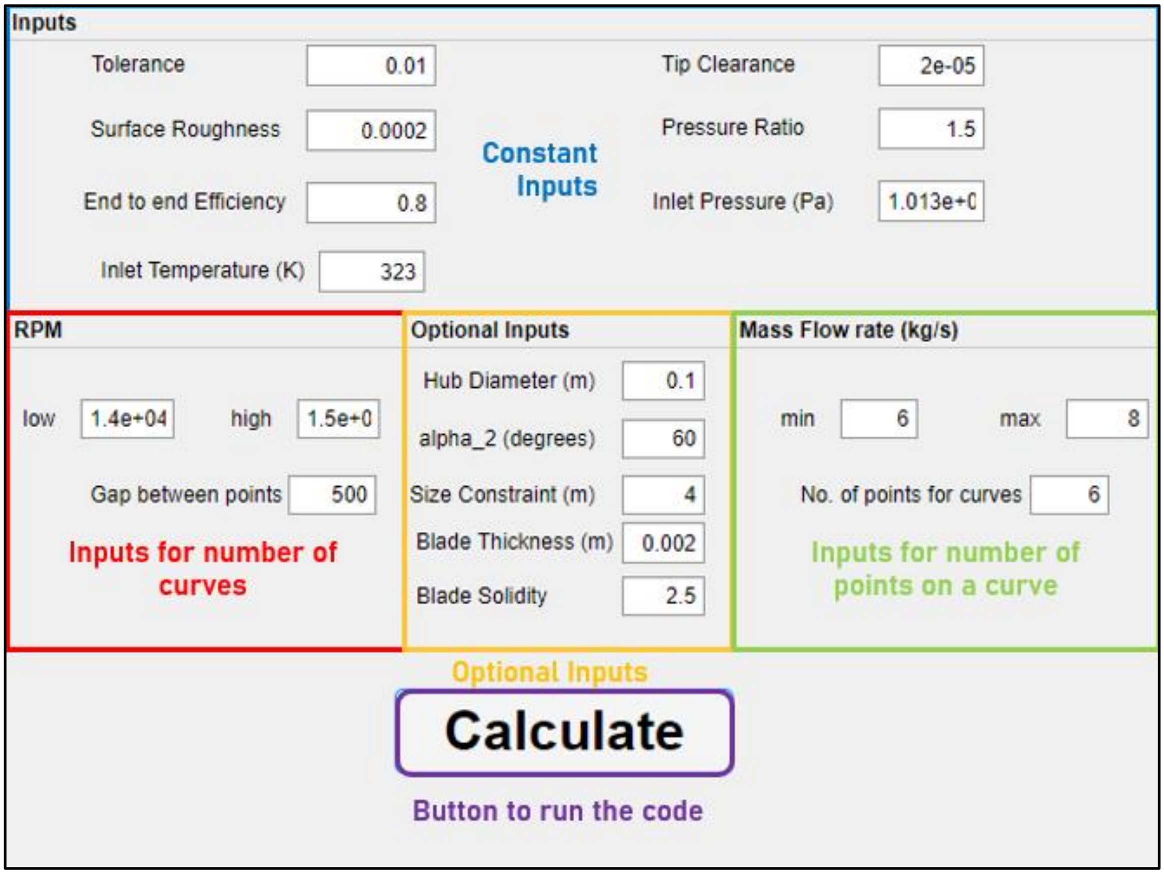


Figure 37: App interface Input

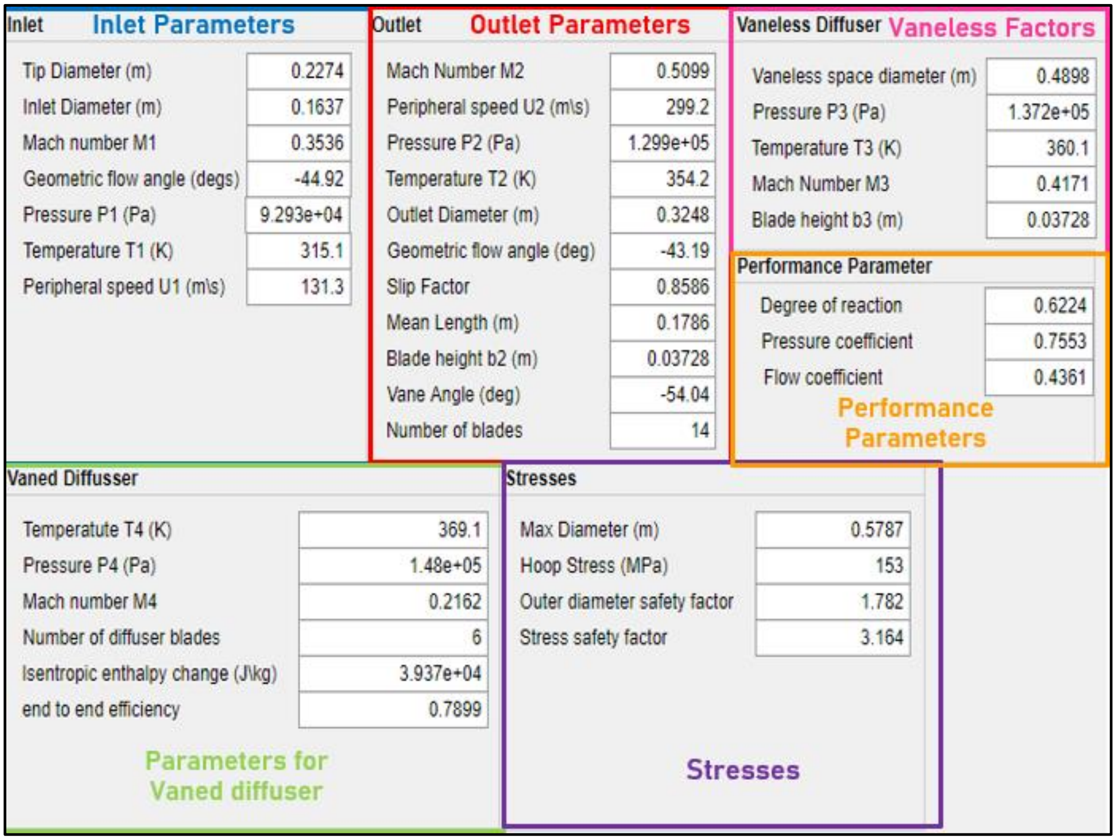


Figure 38: App interface Output

DER KOMPRESSOR

PERFORMANCE PREDICTION