STUDY OF LEAKAGES IN SCREW COMPRESSORS

By

ANIKET GANPATBHAI PATEL

17MMET17

DEPARTMENT OF MECHANICAL ENGINEERING
SCHOOL OF ENGINEERING
NIRMA UNIVERSITY
AHMEDABAD-382 481

MAY 2019

Detail Study of Leakages of Screw Compressor

Major Project Report - Part I

Submitted in partial fulfillment of the requirements

For the Degree of

Master of Technology in Mechanical Engineering

(Thermal Engineering)

By

Aniket G. Patel

(17MMET17)

Guided By

Dr. Vikas Lakhera

Mr. Hitesh Patel

DEPARTMENT OF MECHANICAL ENGINEERING
SCHOOL OF ENGINEERING
NIRMA UNIVERSITY
AHMEDABAD-382 481
MAY 2019

Declaration

This is to certify that

1. The thesis comprises of my original work towards the degree of Master of Technology in Mechanical Engineering (Thermal Engineering) at Nirma University and has not been submitted elsewhere for a degree.

2. Due acknowledgment has been made in the text to all other material used.

Aniket Patel

17MMET17

Undertaking for Originality of the Work

I, **Aniket Ganpatbhai Patel**, Roll. No. **17MMET17**, gives undertaking that the Major Project entitled **"Detail Study of Leakages in Screw Compressor"** submitted by me, towards the partial fulfillment of the requirements for the degree of Master of Technology in Mechanical Engineering (Thermal Engineering) of Nirma University, Ahmedabad, is the original work carried out by me and I give assurance that no attempt of plagiarism has been made. I understand that in the event of any similarity found subsequently with any published work or any dissertation work elsewhere; it will result in severe disciplinary action.

Signature of Student

Date:_______________

Place: Nirma University, Ahmedabad

Endorsed by

(Signature of Guide)

Certificate

This is to certify that the Major Project entitled **"Detail Study of Leakages in Screw Compressor"** submitted by **Aniket Ganpatbhai Patel (17MMET17)**, towards the partial fulfillment of the requirements for the award of Degree of **Master of Technology in Mechanical Engineering (Thermal Engineering)** of Institute of Technology, Nirma University, Ahmedabad is the record of work carried out by him under our supervision and guidance. In our opinion, the submitted work has reached a level required for being accepted for examination.The result embodied in this major project, to the best of our knowledge, has not been submitted to any other University or Institution for award of any degree.

Dr. Vikas Lakhera
Head of Depatment and Guide
Department of Mechanical Engineering
Institute of Technology, Nirma University
Ahmedabad.

Mr. Hitesh Patel
Manager, New Product Development,
Contact cooled Rotary,
Compression Technologies and Services,
Ingersoll Rand (India) Limited,
Ahmedabad.

Dr. R. N. Patel
Additional Director,
Institute of Technology,
Nirma University,
Ahmedabad.

Acknowledgments

Though only my name appears on the cover of this dissertation, a great many people have contributed to its production. I owe my gratitude to all those people who have made this dissertation possible and because of whom my post graduate experience has been one that I will cherish forever.

With immense pride and pleasure, I express my sincere gratitude towards **"Ingersoll Rand (India) Limited, Naroda, Ahmedabad"** for providing me opportunity to work on this project. At Ingersoll Rand , I would like to articulate my deep gratitude to my project guide **Mr. Hitesh Patel** who has always been my motivation and always available for help.

I would like to thank my institute project guide **Dr. Vikas Lakhera**, for his advice and support during the work, and especially for his confidence in me. He gave me the freedom to pursue my ideas and work at my own pace, and was always available to discuss various problems on the way.

I am very much thankful to **Dr. Vikas Lakhera** (HOD, Mechanical Engineering, ITNU) and **Dr. Alka Mahajan** (Director, ITNU) and all faculty members of Department of Mechanical Engineering, Nirma University, Ahmedabad. who have directly or indirectly helped me during this dissertation work.

I would like to thanks my parents for their love and support through out my life. Thank you for giving me strength to reach for the stars and chase my dreams. Even to my girl friend to cooperate in my busy time.

To all my friends, thank you for your understanding and encouragement in my many moments of crisis. Your friendship makes my life a wonderful experience. I cannot list all the names here, but you are always on my mind.

Aniket Patel

Abstract

A twin screw compressor is a positive displacement machine. Designed with a pair of inter-meshing rotors with helical grooves (called as screws) separated by very small clearance, to compress the air. In order to achieve higher pressure ratio, oil is inserted in an oil injected twin screw compressor which acts as a cooling, lubricating and sealing agent. At moderate pressure ratio condition, the screw compressor proves itself as the best option for compression among all other conventional compressors. Hence it becomes essential to optimize and enhance the performance of a twin screw compressor. Among various performance parameters, the volumetric efficiency of compressor is the main factor which decides the overall compressor efficiency. In the present study, attempts are made to analyze various leakages related to screw compressors. The clearance provided in between two rotors helps in smooth operation. In the present study, experimental and computational work is presented which are utilized to determine the appropreate values of properties related to twin screw compressor. These properties are further used to determine leakage in twin screw compressor. All the four major leakages associated with the oil injected twin screw compressor are calculated by the simpification of the area from where the leakages are determined. The results obtained by both experimental and numerical study are compared and percentage variation of individual leakage is determined. Present study consider the influence of geometrical parameters like areas and clearances on total leakage rate.

Keywords: Twin screw compressor, clearance, leakages, experiments, Computational fluid dynamics, volumetric efficiency.

Contents

List of Figures

List of Tables

Nomenclature

θ_m	Angle of rotation of the male rotor, *degree*
n_m	Number of lobes on the male rotor
m	Net leakage mass flow rate,kg/s
m_g	Net gas leakage mass flow rate,kg/s
m_l	Net oil leakage mass flow rate,kg/s
C	Flow coefficient
A_c	Clearance leakage area, m^2
P_1	Upstream pressure, N/m^2
P_2	Downstream pressure, N/m^2
T_1	Temperature at the end of the suction process, $Kelvin$
R_m	Modified gas constant for oil-gas mixture
β	Modified adiabatic index
C_p	Specific heat of gas at constant pressure,J/KgK
C_v	Specific heat of gas at constant volume,J/KgK
C_l	Specific heat of oil, J/KgK
ϕ	Mass ratio of oil to gas
S	Sealing line length along the rotor, mm
ρ_l	Density of oil, Kg/m^3
V_t	Rotor tip speed, m/s
a	Clearance between the lobe tip and housing, mm
μ_l	Dynamic viscosity of the oil, Kg/ms
w_t	Lobe tip width, mm
m_{gi}	Gas leakage into the working chamber along all the leakage paths,kg/s
C_{bl}	Flow coefficient for leading blowhole clearance
A_{bl}	Leakage area for blowhole clearance, m^2
C_{dmi}	Flow coefficient of the male rotor at the discharge end plate
A_{dmi}	Leakage area of the male rotor at the discharge end plate, m^2
C_{dfi}	Flow coefficient of the female rotor at the discharge end plate
A_{dfi}	Leakage area of the female rotor at the discharge end plate, m^2
m_{go}	Gas leakage out of the working chamber along all the leakage paths, kg/s
A_{bt}	Leakage area for trailing blowhole clearance, m^2
C_{bt}	Flow coefficient for trailing blowhole clearance
C_{dmt}	Flow coefficient of the male rotor at the discharge end plate
A_{dmt}	Leakage area of the male rotor at the discharge end plate, m^2
C_{dft}	Flow coefficient of the female rotor at the discharge end plate
A_{dft}	Leakage area of the female rotor at the discharge end plate, m^2

Nomenclature

A_{il}	Leakage area for interlobe clearance, m^2
m_{gil}	Gas mass leakage flow rate to the suction from interlobe clearance, kg/s
C_{il}	Flow coefficient for interlobe clearance
$m_{litotal}$	Total oil leakage mass flow rate into the working chamber along all the leakage paths,kg/s
m_{li}	Oil leakage into the working chamber along all the leakage paths,kg/s
m_{lrmi}	Oil leakage into the working chamber through the male rotor tip and housing clearance,kg/s
m_{lrfi}	Oil leakage into the working chamber through the female rotor tip and housing clearance,kg/s
S_{ml}	Sealing line length of the male rotor at the rotor tip and housing clearance, mm
S_{fl}	Sealing line length of the female rotor at the rotor tip and housing clearance, mm
$m_{lototal}$	Total oil leakage mass flow rate out of the working chamber along all the leakage paths,kg/s
m_{lrfo}	Oil leakage mass flow rate out of the working chamber through the female rotor tip and housing clearance,kg/s
m_{lrmo}	Oil leakage mass flow rate out of the working chamber through the male rotor tip and housing clearance,kg/s
w_{tm}	Lobe tip width of male rotor, mm
w_{tf}	Lobe tip width of female rotor, mm
h	heat transfer coefficient, W/m^2K
τ	Time required for suction process, $seconds$
T	Total cycle time,$seconds$
N_m	Number of revolutions of male rotor, rpm
M_{total}	Total mass of air and oil present in port, Kg
$M_{Leakage}$	Total leakage mass of air and oil present in port, Kg
Q_{total}	Total volume of air and oil present in port, m^3
$Q_{Leakage}$	Total leakage mass of air and oil present in port, m^3

Abbreviations

CFD	Computational Fluid Dynamics
SCORG	Screw Compressor Rotor Grid Generator
2D	Two-Dimentional
3D	Two-Dimentional

Chapter 1

Introduction

Compressors are used to get pressurised air. Compressors find widescale application in HVAC systems. HVAC industries are doing research for the more efficient and small compressors. Small scale of compressor is only possible when leakages are decreased.

1.1 Compressors

A principle of working of compressors is this device is used for getting high pressure fluid from low pressure fluid. The any type of gas is used as working gas in gas compressors.

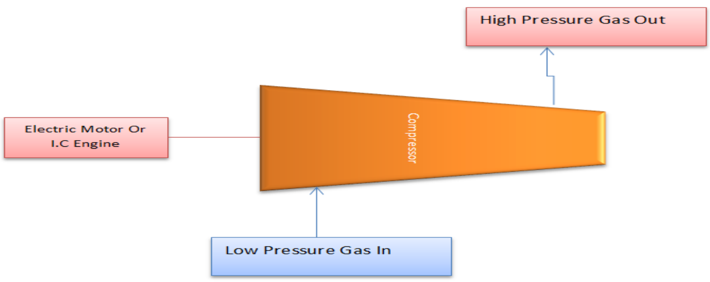

Figure 1.1: principle of compressor[1]

The result as compressed gas, the compressor consumes electric energy for doing mechanical work. Hence the machine consume power so it's device which is consumer of power.

1.2 Categorization of compressors

As output of a compressor, it pressurised gas by two broad way. Where it would be achieve by decreasing the volume and using the high kinetic energy of gas is getting from pressure through diffusers. By that the classification is below:

1. Dynamic type compressor

2. Positive displacement compressors

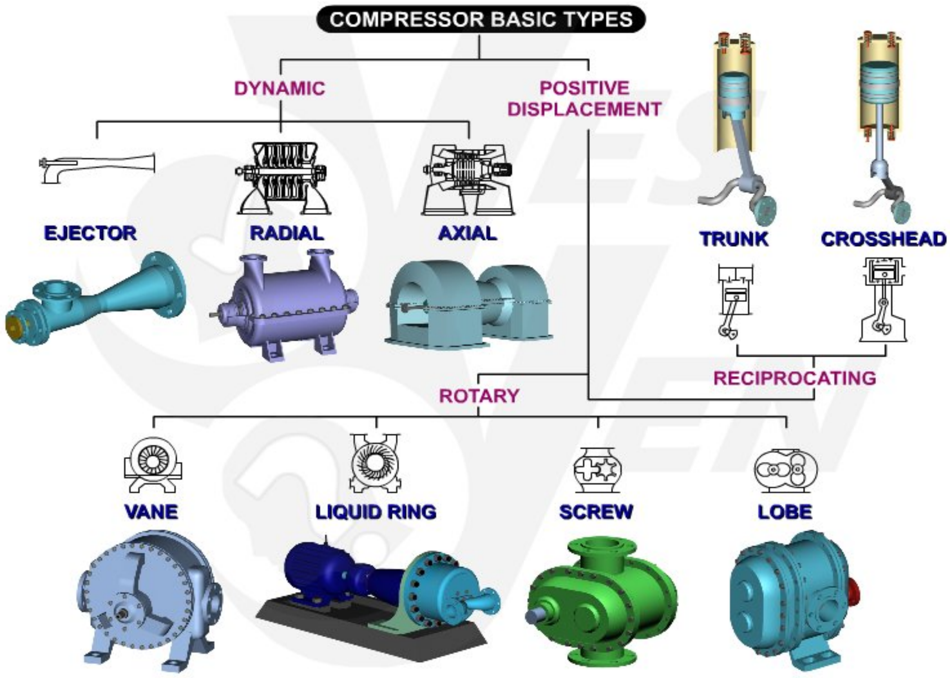

Figure 1.2: Categorization of compressor[1]

In dynamic type compressors, there are two division of classification centrifugal and axial type compressors. Centrifugal force is applied on gas or air and it would be done in centrifugal compressors. In axial compressors supersonic speed or kinetic speed of gas is converted into pressure.

Positive displacement compressors should be reciprocating or be rotary. In reciprocating type piston and cylinder mechanism is used to compress the gas. In rotary compressors there are rotary element like helical screw, vanes and scroll is used for compression.

1.2.1 Reciprocating compressor

In reciprocating compressors first stroke is suction stroke, during this stroke the air is sucked into the chamber as opening of inlet valve. In next stroke it has discharged from discharged stroke. Here all works are done by piston mechanism. There should be single acting and double acting reciprocating compressor. Whenever requirement of the high pressure ratio then it should be used. The compressors have more maintainance compare to others also perform a very noisy operation.

1.2.2 Rotary compressor

Where in rotary compressor, the assemblies are using the elements like helical screws, vanes and scroll in order to get pressurised gas. As gas is entered in the rotary element the gas is continuously passed by revolving action to the less volume as a result of this we are getting compressed gas. Rotary compressors are classify by usage of element the what elements are used to compress the air. There are types are screw compressors, lobe compressors, root blower, vane compressors, scroll compressor and liquid ring compressors.

1.3 Screw compressor

This type of compressors are similar to rotary compressor, which is working on helical rotors for compression. When air is trapped in inlet section of revolving screw lobes then after in final we can get pressurised air at outlet section. As compare to reciprocating part screw compressors have not generated much noise, they have low maintenance cost, light in weight. Screw compressors are classify by single screw compressors and twin or double screw compressors. In single screw compressors there are one screw and twin screw compressors there are 2 male and female screws are there.

1.3.1 Twin screw compressor

This type of compressors as a positive displacement rotary type compressor, which have the inter connected helical two screws. Which both are rotated at constant speed. This rotors have one side which is inlet and from inlet air is sucked and it would passed in the working chamber and then it goes to outlet port. Where we can use output for different application.

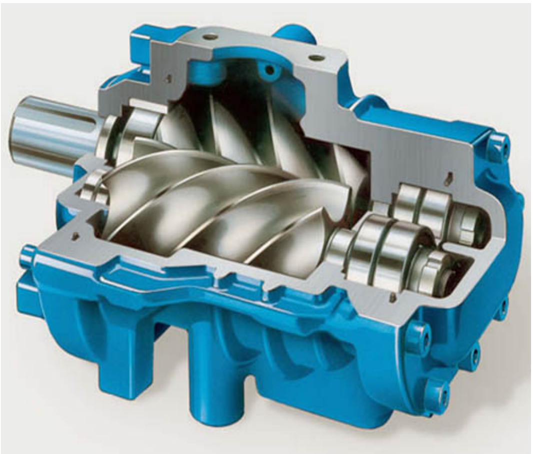

Figure 1.3: Twin Screw Compressor[4]

There are two rotors, one is male rotor and second is female rotor. Male rotors have lobes and female rotors have flutes. Male rotor is power shaft and it is directly connected to motor. Female rotor is driven by male rotor so both are on same speed. Due to lobe and flutes both the rotors are rotating simultaneously. There should be some clearance gap between the rotors and by that both are rotate smoothly. This is mandatory for frictionless rotations. Also it would gives cooling to system.

1.3.2 Operation cycle

Compression cycle of screw compressors are more likely to the compression cycle of the reciprocating cycle. Just difference is it would not go to re-expansion. Hence $P-V$ diagram as shown in Figure 1.4 is similar to reciprocating compressor.

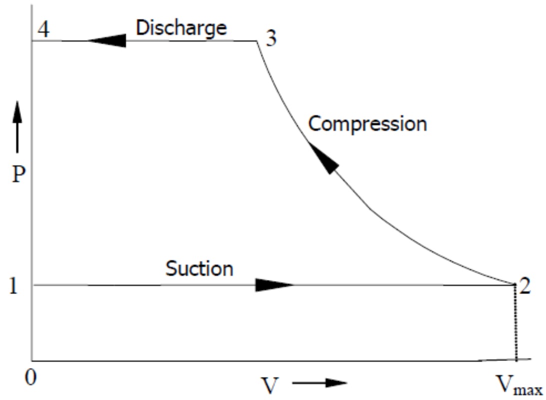

Figure 1.4: Pressure-Volume diagram for screw compressor[2]

In the system of Compressor, there are suction, compression and discharge process. first process is suction in to the process there would be feeling process in this particular phase. Gas would be entered through out the process 1-2. This process is coming with total 360 degree. The process sucking gas from inlet then the compression and discharge should be placed around 2 rotations. By decreasing the volume, there is compression should taking place and both the $P-V$ and $P-\theta$ diagram shows that there should be simultaneously working of compression and discharge process.

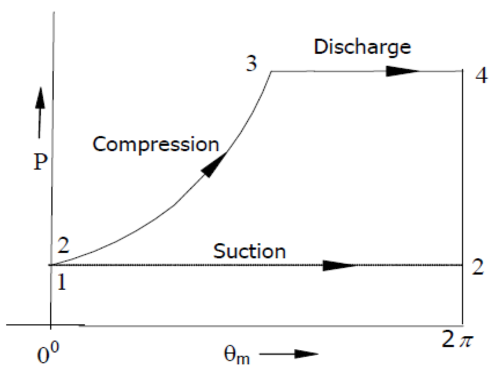

Figure 1.5: $P-\theta$ diagram[2]

There are two types of twin screw compressors with oil and without oil, hence there are two types of compressors oil free and oil injected twin screw compressors.

1.3.3 Oil free twin screw compressor

Oil free term itself shows the definition. Here compression without oil is taking place and total work is free from oil contamination. Working chamber has not oil as working fluid so we can not pass the high pressurised air or gas because it generates heat over cavity and it gives the heat generation rate with large numbers. Hence it is suitable for low pressure output and low speed works only.

1.3.4 Oil injected twin screw compressor

Here we can work behalf of oil free compressors. We have oil and air ratio to enter the working chamber and some particular ratio is constantly entered in working chamber. Oil is working as lubrication and as well as it works as it is absorber of heat. Hence oil injected compressors are using in high pressure ratio can be needed without any external cooling. Oil is providing the protection against the gases.

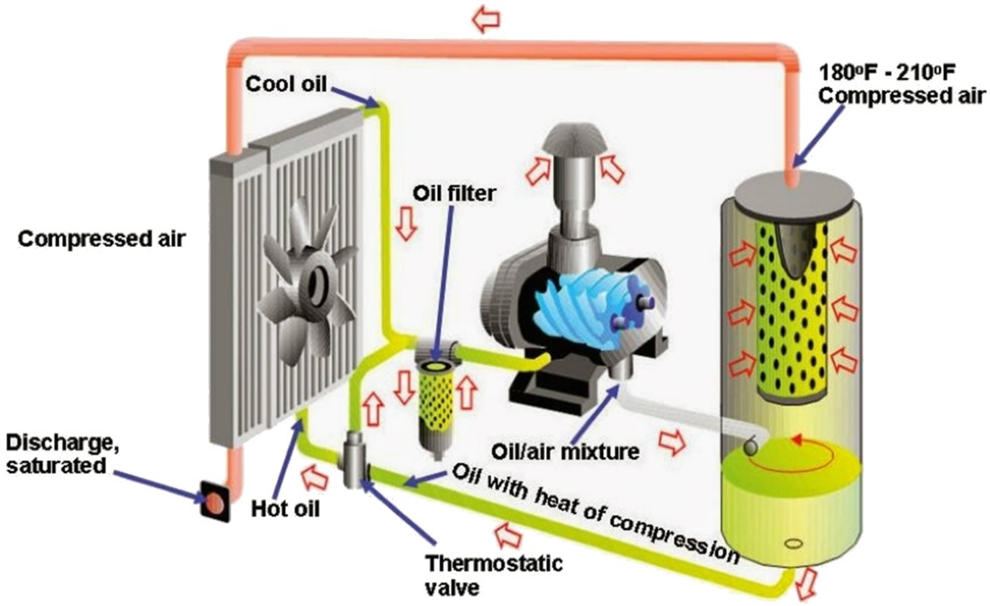

Figure 1.6: Air oil circuit for oil injected twin screw compressor[3]

1.4 Advantages

There are some advantages of screw compressor over conventional compressing machines as

It has less moving pars compared to any machines

Longer life, as there are movements in assembly.

Less noisy operation

It gives good pressure ratio in compact design

Compactness decreases working area

Working with different gases for compression purpose

Because of less moving part it can run on high speed

Compression efficiency is better as high compression ratio and high speed of rotors

Even if inlet pressure is very low, operation is possible

There are various operation should be possible as there are range of flow rates and range of pressure ratio is possible

1.5 Limitations

In this case large quantity of oil is injected in the system to fulfill the purpose of lubrication but as more oil is there lesser the amount of air is there.

Rotor geometry have clearance gap in between. As a result of this leakages should take place.

As large clearance is take place as more leakages is in the system is take placed and, device have low η_v.

1.6 Problem specification

In the industry screw compressors are performing a very large role. Now the screw compressors are device which would be considered as the based product of many assembly. Rotary screw compressors are now used as air compressors, which should be a positive displacement mechanism. This is the alternate option for the reciprocating one. Where the large volume of the compressed air is needed, It should be used in a highly industrial area or where high power air tools are used as a jackhammer. Screw compressors are highly used in food packaging mechanisms, refrigeration systems and pneumatics automated systems. The performance of the screw compressors are very notable factor in all the industry which would be working with the screw compressors.

There are many factors which should affect the volumetric efficiency of the screw compressors. Like the rotational speed of the rotors, pressure difference, the gap between rotor to rotor and case, etc. As a design aspect there would be one area from where volumetric efficiency would be increased and the factor is gap between rotors and housing to rotors. In industry there should be gap used now days is around 100 micron. Now the task is investigation of changes when gap is decreases.

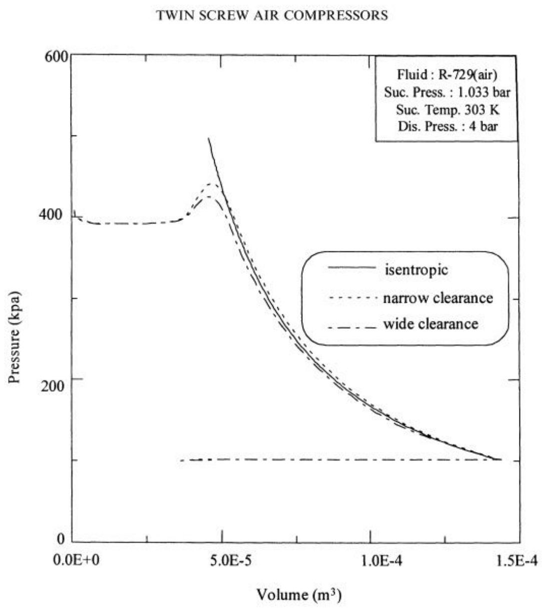

Figure 1.7: Effect of clearance gap on the performance of an air screw air compressor.[21]

In the above figure, the working fluid is air and different screw compressor with a different clearance gap should be monitored and the graph of pressure vs volume is generated. There is some gap between the different conditions. That means the clearance gap is played a vital role in the volumetric efficiency of the screw compressors.

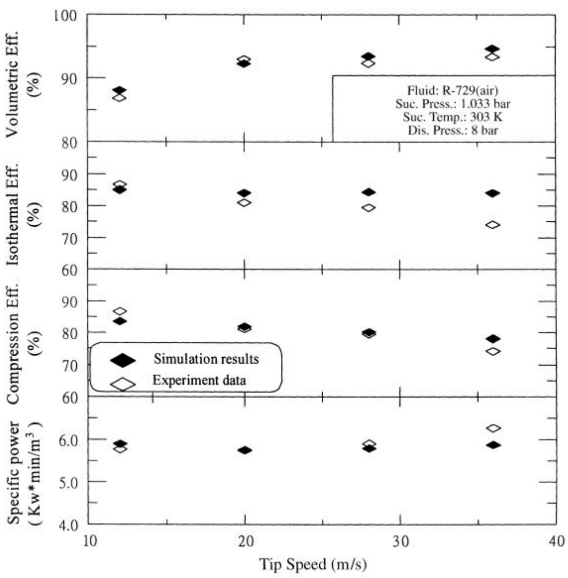

Figure 1.8: Calculated results vs experimental results.[21]

There are in the above graph volumetric efficiency, isothermal efficiency, compression efficiency, and specific power would plot with respect to the tip speed. The gap between the experimental and simulation results are going larger as speed is increased.

Now the question is, how should leakages affect the different leakages of the screw compressors? What is the nature of the leakages in the micro gap between the rotors and what is the amount of the leakages? There are leakages like blow-hole leakage, interlobe leakage, leakage through discharge end plate clearance and leakage through the case and rotor clearance.

The exact problem is how can we find the leakages by a computational tool, experiments, and mathematical modeling. The target is finding out the one simulated method to get all those leakages by those all method. What is the simulated results with new modified designs? What is the aftereffect of that changed parameters?

In the present study, the problem is being simplified with certain assumptions. This study would include mathematical modeling, computational modeling, and experimental setup. This is the engineering where you have some simple solutions to the problems and things would be more simplified. In the present work we simplify the problem and done experiments.

1.7 Objectives

In this investigation these are the subjects as objectives,

To investigate major factors affecting the leakages and how they affect the system.

From the results of a CFD analysis, mass flow rates and power output for different pressure ratios and different designs were obtained.

Experimental set up cost goes very high if we are doing testing with different dimensions so one validated tool for computational work then its a big problem solver.

Here we have enough area for improving the efficiency but the research work is going helpless without CFD. That's why the objective is making system more easy in manner of computational power.

How can we make an equivalent rectangular channel to the nozzle? is simulated.

Study of the blowhole leakages and the interlobe leakages and area from where leakages have been taking placed.

Comparision between the experimental and computational data, and make a appropriate computational tool.

Mathematical modeling by which we can calculate the leakages from different areas and modify the available tools.

Perform experiments with the certain assumptions and compare with the results of computational fluid dynamics.

Chapter 2

Literature review

The main concentration of the study is improvement of the leakages. There are many different ways to find the performance of the compressors. Investigation should be either on analytical modeling or computational modeling or experimental modeling. Experimental model is used to understand the actual case and the actual amount of the leakages associated with the system. By the knowing of the actual case, there should be useful to many other cases of study. Computational modeling is also a best way to find out the leakages and the performance of the compressors. Computational modeling is the best way to simulate the exact overview of temperature, velocity profile and the pressure difference in different contour. Where analytical approach gives the appropriate case of the problem by using the governing equation. In present study computational and experimental data would be compared that would be shown below in this study.

Literature review for the different fundamentals about to design factors, CFD, experiments and analytical study simplifies the problem. In the study, there are some data which were giving an appropriate idea where our study is going on the right path or not.

2.1 Design factors and parameters

C. X. You, Y. Tang and J. S. Fleming had told that the influence of screw or rotor geometrical parameters is explained with center distance between the male and female rotors, outer diameter of the male rotor, mm Length of rotors, lobe numbers of the male and female rotors and respectively male rotor wrap angle[4].

Singh, P.J., Bowman, J.L.[5] have mentioned that rotary compressors have the different features. Those features are based on the working pressure some design constraint and male female rotor specification. Number of the lobe and flutes are also be a considerable factor of the design of screw compressors. Those system would be designed and then after manufacture should been done. This paper show the factors affecting the manufacturing constraints and how much manufacturing cost should? There are lobe design and flutes design which are very costly and tough to manufacture. In this study they told about parameters and the study about that parameters.

V. Boldvig ,V. Villadsen both had done many experiments on screw compressors and told that the parameters like length diameter of rotor and most influence parameter is pressure difference and speed of rotor. By that they told about to selection of the screw geometry. By that only we are get up to the best working design of compressor[6]. In research paper of C.X., Tang, Y., Fleming, J.S.[4] gives research to get the best design constrain about the lobes. They told about the parameters like length to diameter ratio and combination of lobe consideration. In this paper we can find the different lobe combinations also define well.

When design should be modified, we have to clear the perspective through the characteristics. Here a paper by the P.Jenno Xavier, K.Kanthavel, and R.Uma Mythili[22] on the rotor profile design just shows the parameters and the calculation on the rotor profile by the curvature length. Even here in the paper, the author gives the major reason for the leakages is blowhole leakage and they are considering the blowhole and interlobe leakages are the major leakages in the screw compressors. The calculation would be on every segment shows the one instinctive parameter is affecting the profile design on the major bases.

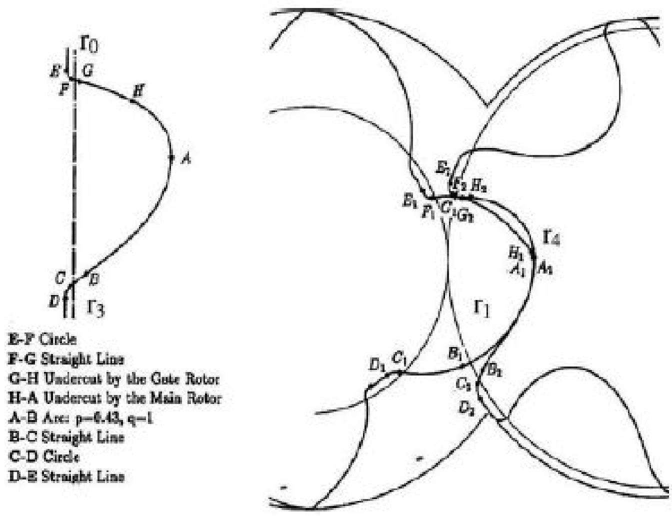

Figure 2.1: Profiling of Rack[22]

In the paper of Nikola Stosic, Ian K Smith, Ahmed Kovacevic and Elvedin Mujic[23], there are some design aspects of the profile of the rotor and they analyze the rotor profile from the history. They define the different coordinates for the design of the rotor profile. In that paper, they define some derivations of the Standard gearing and other similar conditions of the lobes and gear. Last profile shows a hyperbola that seems the almost best suitable replacement, which gives the grate ratio of the sealing line lengths and rotor displacements.

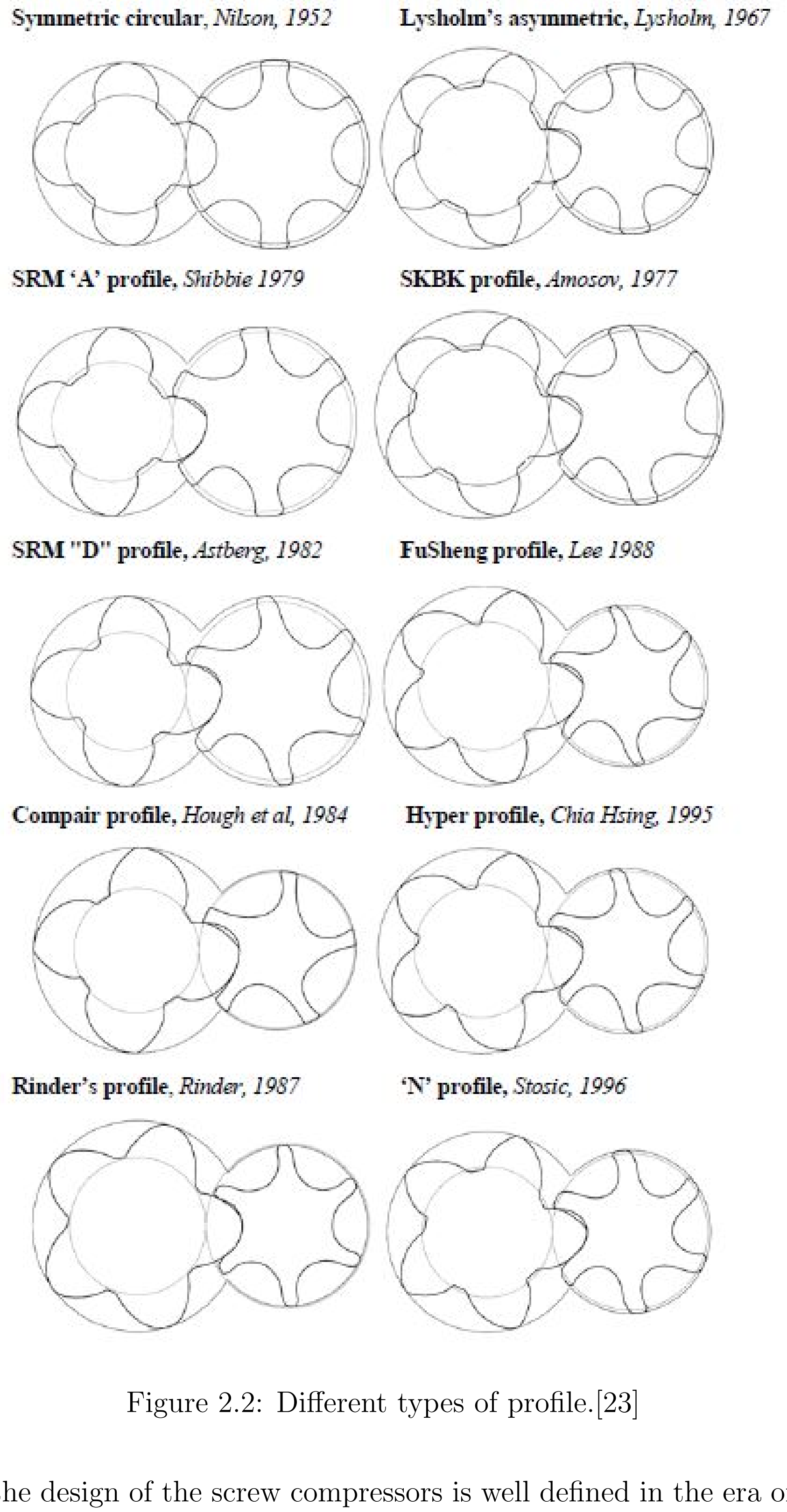

Figure 2.2: Different types of profile.[23]

Although the design of the screw compressors is well defined in the era of engineering development. Still, design gives the more refinement in the efficiency and reduction in the size of the rotor and screw compressors and also the cost of these parts and compressors. Although there is wide development in the area of designing of the screw compressors, there is the scope of the further improvement on strength and weight of the rotor with the high displacement and very fewer contact stresses.

2.2 Flow and leakage analysis

For getting a smooth working and noise free working some design level tolerance would be given to the final design. There are some clearance gap would be given between the male and female rotors and the different parts and rotor gap. Gap is creating the leakage from it. By that there some fluid leakage is taking place and in last the volumetric efficiency has been decreased. the gap of 0.01mm gives the decrement of 1% efficiency of compressor.[7]. The main problem in the industry is decrease the leakage amount from the clearance gap. J S Fleming and Y Tang [8] have studied the different six leakage from the system and they simplify the leakages path in geometry. Following figure 2.3 shows the path and figure 2.4 shows the leakages direction and the area from it should passed. Here in following figure 2.3 paths are inroduced by numbers as,

1. Path 1: contact area of both rotors
2. Path 2: gap in between rotor and housing
3. Path 3: leakage from blow-hole
4. Path 4: starting of blow-hole
5. Path 5: gap between end of suction and rotor
6. Path 6: gap between end of discharge and rotor

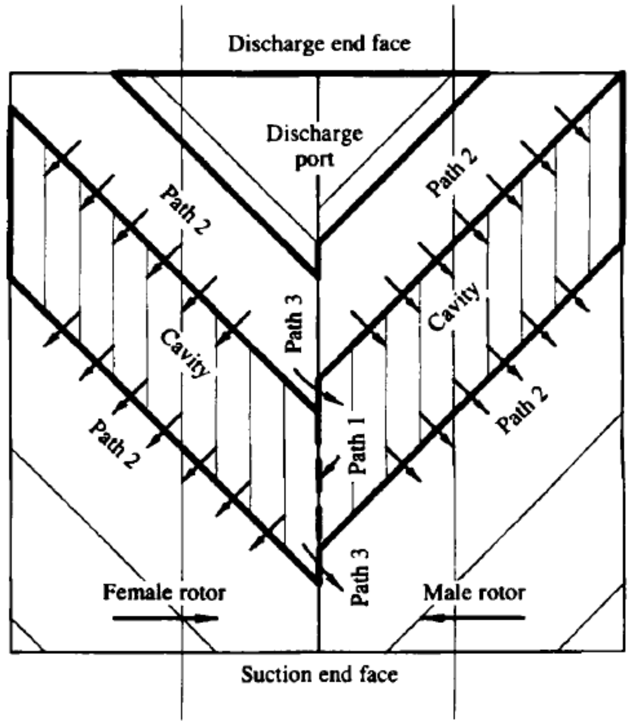

Figure 2.3: Leakage paths in oil injected twin screw compressor [8]

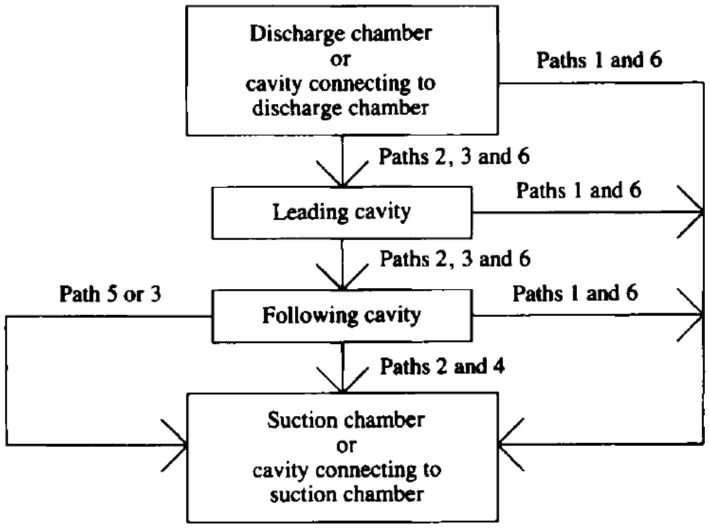

Figure 2.4: Leakage distribution in oil injected twin screw compressor [8]

In that paper, the author is presenting the mathematical program to analyse the data about the leakages from different gaps and that would be presented by the computer program. This program is able to give the leading flow and the lagging flow leakages in the system. The authors are also put the study about the how speed affect the leakages and in which manner leakages are taking placed between the system.

As there are some leakages and that would be found out by the researchers by the experiments and simulations. There are one lagging point is there that we can't find the leakage in a particular zone in given situations. There are many researchers are working on that but they didn't define the appropriate way to define.

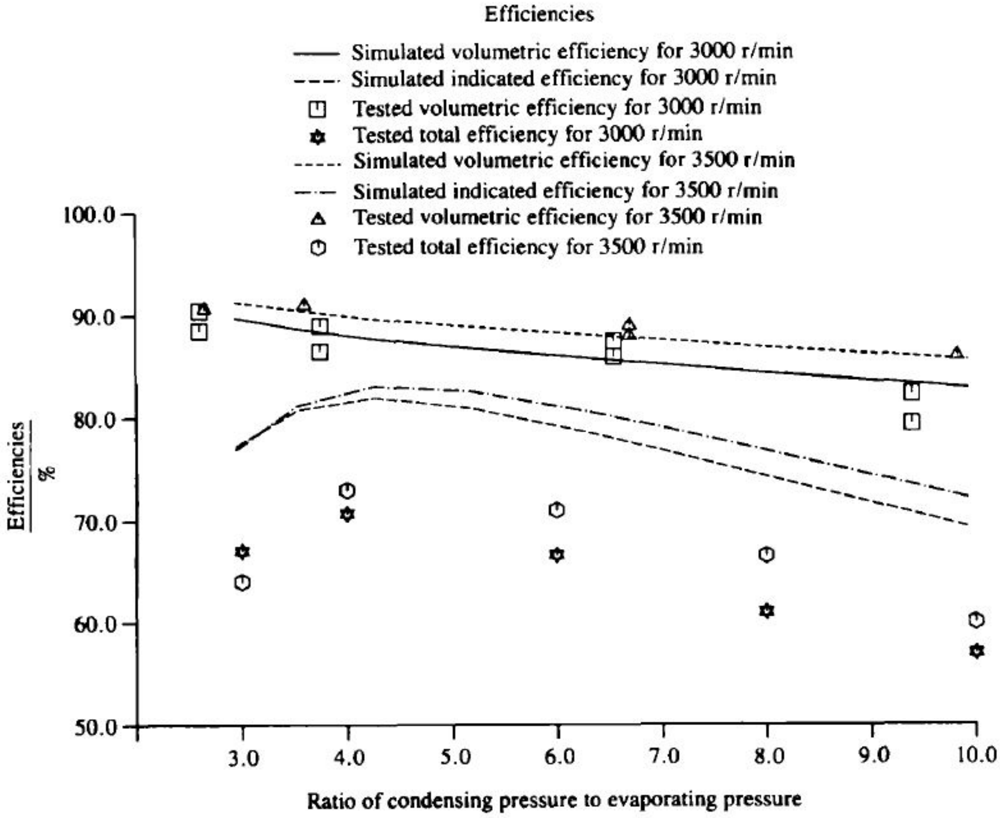

Figure 2.5: Tested efficiencies.[24]

In this figure, we have the data of simulations and tests are directly compared. The results are more likely similar in between the indicated efficiency and the total tested efficiency. In this analysis, there were two different speeds are there and they both indicate that the difference in efficiency by changing the RPM of the rotors. By controlling the leakages and rpm of the rotor is directly gives the parallel change in the efficiency. Now having this final conclusion by some kind of literature review that testing of leakages is important for future reference.

2.3 Mathematical modeling

There are various methods used to determine the performance of the compressors includes experimental analysis, analytical analysis, mathematical modeling and CFD or computational model. Experimental set is prepared on basis of the actual case then upcoming results are very similar to the actual case. It will give the perfect idea about the actual condition too but the limitation is preparing of experimental set is very expensive. If we are fail to achieve the appropriate result then we cant do more research by this way of working. That's why we have analytical approach and computational modeling.

Fujiwara and Osada[9] had given the paper about the analytical model and the program to simulate the heat transfer in the system and that will give the performance of the compressors. They did work on the oil flooded screw compressors. In the Study of this paper the observation of the experimental data told that the decrement of the volumetric efficiency. This study also tells about the efficiency in now days are much better then conventional type compressors.

As more use of the twin screw compressor in the industry of waste heat recovery. There is mathematical model for the calculation of the performance of the twin screw compressors. The mass and energy conservation law of the thermodynamics are applied to the system. There are many flows occur in the system and by that there are pressure drop in the working area. This model state the mathematical problem for the leakages and the flows which should decrease the efficiency[10].

$$m = m_g + m_l = \frac{CA_cP_1}{\sqrt{T_1}}\sqrt{\frac{2\beta(r^{2/\beta}r^{\frac{\beta+1}{\beta}})}{(\beta{*}l)R_m}}$$

for $r > (\frac{2}{\beta+1})^{\frac{\beta}{\beta-1}}$

$$m = \frac{CA_cP_1}{\sqrt{T_1}}\sqrt{\frac{\beta(2/\beta-1)^{\frac{\beta+1}{\beta-1}}}{R_m}}$$

for $r \leq (\frac{2}{\beta+1})^{\frac{\beta}{\beta-1}}$

Numerical modeling of the screw compressors has been developed calculating both discharge and suction for finding the performance. Even this study tells that the flow coefficient would be calculated by the experimental observation of compressors and by the volumetric efficiency of the screw compressors.

Mathematical modeling by using the principle equation of convergent and divergent nozzle gives that all modified equations. In the publication of the Nagam Seshaiah, it should clear that nozzle equations are the basic equations to get leakages from blow hole and the interlobe leakages. Even Nagam Seshaiah published another paper on the Screw compressors leakages shows the all basic equations are nozzle equations.

Mathematical modeling is helping us to get up to the actual case but there are certain assumptions. Sometime assumption makes some errors. To get out from these errors, there are computational modeling and Experiments are there. Considering the above topic, next Literature review would be on computational works and experimental works.

2.4 Computational analysis

As we know, there are certain analytical methods available in the present day. Computational fluid dynamic is also one of them. As the same working experimental set up or prototype manufacturing needs huge amount of investment so CFD is the most appropriate method among all. Now computational methods are totally dependent on the computational power. There for some time we have some complex or large area should be analysed by simplification of the system. By that we can get similar results as actual case.

Iva Papes, Joris Degroote and Jan Vierendeels had discussed the CFD in their research work that in the filling process there should be some pressure drops occur in the system. When the pressure ratio is been placed in the system there are some over and under expansion can occur. With the increase in the pressure ratio or decrease in rotational speed the influence of the leakage flows is more significant. The study presented in this paper will be used to improve the performance of the expander and optimize the design[13].

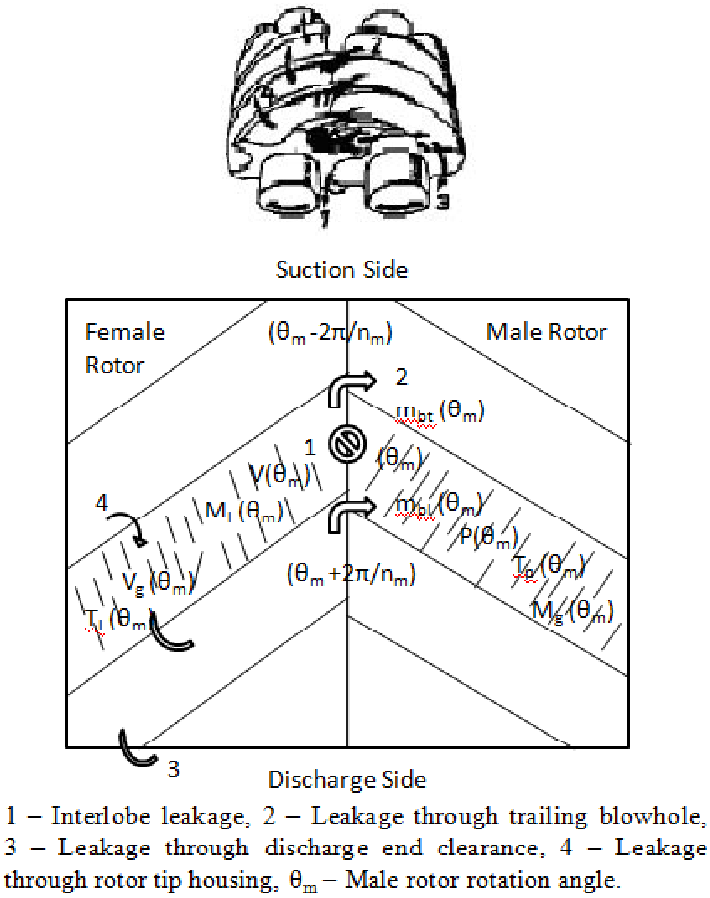

Figure 2.6: Different leakages in compressors[20]

A kovacevic and S.Rane[14] have put in one of the research paper with modeling of the positive displacement instrument in SCORG. In that paper, they duscribe how the performance of the air screw expander with 3/5 lobe specification. Here author is investigate the whole model with the use of the CFD. The same process is applied to the compressors and the leakages are investigated. In present study SCORG is used as working mesh generator in this system.

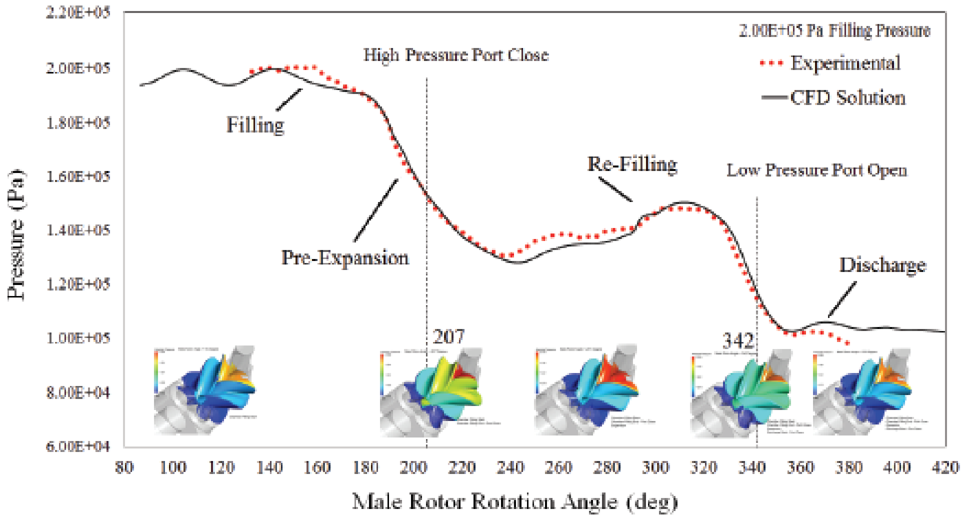

Figure 2.7: Diagram of indicated pressure in twin screw compressor[14]

Andreas spille-kohoff and Jan hesse presented the TwinMesh with the screw design and they would do the simulation based on the detailed mesh made by TwinMesh. They occupied the gas compressibility and the turbulence effect, heat transfer through parts and leakages from clearance gaps. In the paper, they show the time-resolved results for pressure, velocity, and temperature[15].

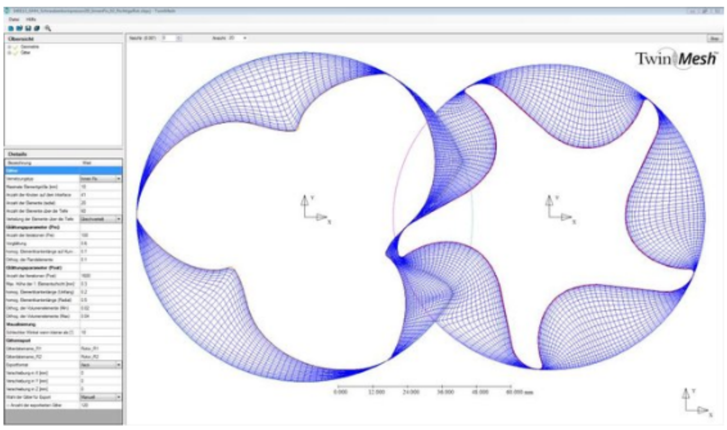

Figure 2.8: TwinMesh GUI for the mesh generation[15]

Jan Vimmer did the study of the cross section of a narrow channel formed by a tooth of male rotor and housing in a compressor. They just pick up the small area where the leakages occurred and simulate the system. The profile gives the flow rate, velocities, and pressure at different sections. This whole study he did in 2 dimensions. The influence of turbulence models on to the results is investigated. That's the simplification of the large area which should be tough to simulate in once.

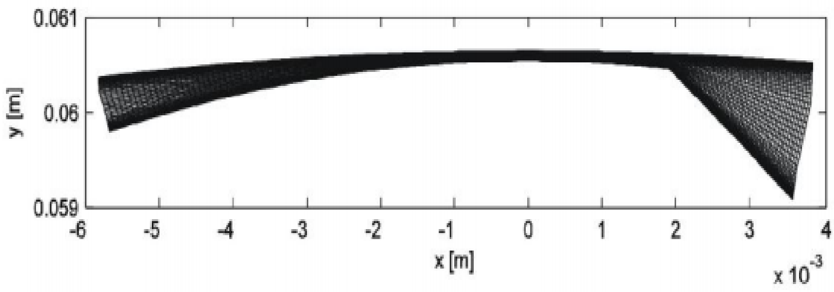

Figure 2.9: Computational domain and computational grid with 17868 quadrilateral cells[18]

The contour of Mach number and the static pressure distribution in the channel shows that no shock is available in the flow. By the velocity profile, we can see the behavior of the fluid in the channel.

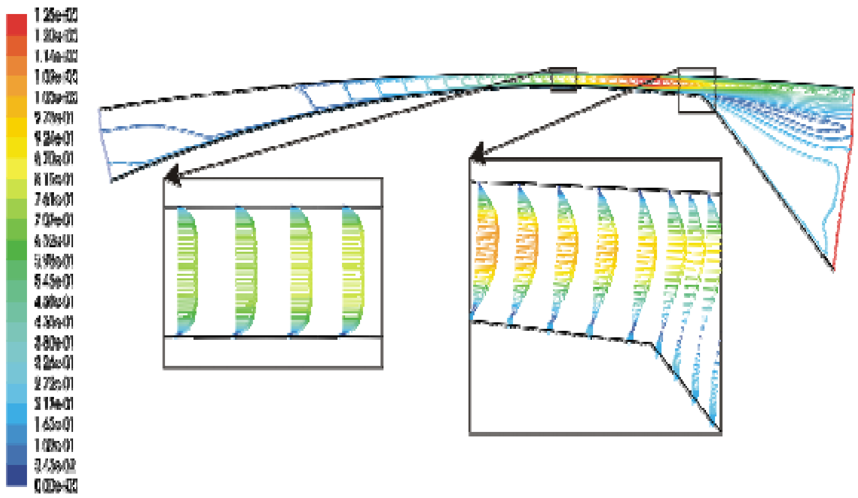

Figure 2.10: Contours of Mach number and velocity profiles in the male rotor-housing gap[18]

Here we can see velocity profile that would be in one pattern that shows flow doesn't have a shock in the system.

Wang, Zhigang, Zhang, Sun and Lili[17] put the studies of the positive displacement machine. They have studied whole system with the consideration of the experimental parameters like oil gas leakages, oil injection, gas oil heat transfer and refrigerants properties. The possibilities of prediction of the author was demonstrated by experiments. Where their measurements was compared with their corresponding computer simulation for the same outlet and inlet conditions, rotor speed and back pressure.

In this paper, the Author did work on a de Laval nozzle. He has a Design of the geometry and gave the details of this. Even shared the model of the nozzle and dimensions of this nozzle. As we are planning to do simplify the problem. Obviously, we have to work with the nozzle, microchannel, and orifice. In this paper, the author is giving the details of working tool of CFD. Also they are mentioned details about the meshing, pre and post processing.

General	Solver type: Density-based 2D Space: Axi-symmetric [3]
Models	Energy equation : On Viscous model : Standard k-ε model, realizable, enhanced wall treatment.
Materials	Density: ideal gas Cp = 1880J/kg K γ = 1.19 Viscosity = 8.983x10^{-5}Pa. s Thermal conductivity = 0.0142 W/mK Mean molecular mass = 27.7 g/mol
Boundary conditions	Inlet Pressure = 100bar Inlet Temperature = 3300K Outlet Temperature = 1700K (For initialization purpose only)

Figure 2.11: Problem Set-up

In the study conducted by Nikhil D Deshpande[22], simulation using ANSYS was done for different types of nozzles. Meanwhile they are doing calculations for the velocity, and pressure. by having both the results they are comparing the data and giving appropriate CFD tool for the calculations. Eventually, they have done calculations based on the one-dimensional nozzle flow.

Dr. BASUDEB MUNSHI did work on the computational fluid dynamics for microchannel. He did work with the rectangular microchannels. There is flow in between channel gives different behavior. There is a high Nusselt number which shows that there is more distance between atoms and so physics of compressibility is changed. The nussult number and Reynolds number are important to analyze the flow of the system. Whenever its microchannel then its most important to observe it.

2.5 Thermal analysis

In the paper of P. J. Singh, J. L. Bowman we can see mathematical modeling of thermodynamic analysis of a oil flooded screw compressor. In study, they were assuming the oil injected in the form of non interacting spherical droplet. When droplet hit the boundaries of rotor, after that in a free flight time it would done heat transfer with rotor. That will give us a very improved compressor performance.

$$h = \frac{(k * P_0 * V_0)}{(k-1) * At_s} \frac{(d\eta_v)}{(dT_0)} \tag{2.1}$$

This equation is used to find the heat transfer coefficient, here is the equation for finding the relative heat transfer are as $A = V_{th}^{2/3}$. The authors are giving the statement about the wrap angle and the statement is as increasing the wrap angle as it is good to compressor's leakages. Leakages are decreased as the wrap angle increase.

Chapter 3

Analytical modeling of Leakages in Screw Compressors

In the present study, there were some leakages and factors which would be affected by the leakages are discussed. Flow path and flow diagram of the screw compressors are different rather than the other compressors so this chapter itself would be helpful to understand the flow characteristics and factors affecting the leakages. This study provides the couple of the equations which would be used for the determine the flow rate from different gaps of the screw compressors. In this study, there would find the set of the different nozzle and slit equations which have been modified as per the gap, area and flow coefficient is subjected to the problem.

3.1 Recapitulation of Leakages

The volumetric efficiency is directly affected by the total leakages of the screw compressors, and hence the overall efficiency also been affected by the leakages. Leakage should the return or backflow which is generated in the working cavities. The pressure difference between two working cavities drives the fluid into the opposite direction through the applied gap in the screw compressors. This backflow is the main reason to decrease the efficiency of the screw compressor.

There are mainly two types of leakages, like gain leakages and loss leakages. Those all are categorized on the basis of directions and the flow of the back-flow. Throughout the whole compressor or any working chamber, there are two possibilities either it is in-flow or it would be outflow. When the flow direction is from higher pressure or temperature cavity to the lower pressure or temperature, is called gain flow to the working chamber. If the flow is within the opposite direction of the gain flow then it should be loss leakage. Different types of leakages are associated with the Screw compressors and those are defined by the J S Fleming and Y tang.

Many factors have been affected the amount of the leakages of the compressors, like pressure difference between the leading and present cavity, temperature of the successive chamber, rotational speed of the rotors, etc.

There are leakages in the screw compressors from the gap placed between the rotors and housing. From all gap provided in the screw compressor, following leakages are in consideration as the main leakage in screw compressors.

1. Leakage through inter-lobe clearance 2. Leakage through blowhole area 3. Leakage through discharge end clearance 4. Leakage through rotor tip-casing clearance

Shows the different path of the leakages and the different types of leakages are associated with the screw compressors. Detail explanation of each leakage is given below and modified equations also given by which mass flow rate from a particular gap is calculated.

3.2 calculation of the thermodynamic cycle-time

Cycle time of the thermodynamic cycle is defined on the basis of the rotation angle of the rotor either male or female. Time of one revolution of male rotor and suction time is almost the same in the manner of suction of air. Hence the compression process time is directly proportional to the wrap angle of the rotor. For N_m is the speed of male rotor in revolution per second (rps) then cycle time for suction process in second is given as

$$\tau = \frac{1}{N_m} \tag{3.1}$$

The entire cycle is depend on the summation of the suction time, and compression time. The cycle time is dependent on the rotor speed, wrap angle and the lobes on the rotors.

$$T = \frac{1}{N_m}\left(1 + \frac{\varphi}{360} + \frac{1}{n_m}\right) \tag{3.2}$$

Here n_m shows number of lobes on male rotor and φ is for male rotor wrap angle.

3.3 Volumetric efficiency

Volumetric efficiency is ratio to the mass which would go inside the compressor and the mass which would be out from the discharge. Discharge mass flow rate is directly proportional to the leakages or the backflow in the compressors. Volumetric efficiency is calculated in terms of the mass flow rate or the volume flow rate. Reduction in volumetric efficiency is nothing but its reduction in the overall efficiency.

η_{vol} = (Amount of mass to be delivered from discharge end) / (Amount of mass which is Contained for same volume from suction)

$$\eta_{vol} = \frac{M_{total} - M_{Leakage}}{M_{total}} \tag{3.3}$$

where
M_{total} is the total mass of fluid passing from suction port in Kg and
$M_{Leakage}$ is the total leakage in Kg

Volumetric efficiency in terms of volume flow rate can be given as,

η_{vol} = (Actual amount of volume to be delivered) / (Theoretical volume of fluid contain for same volume at suction condition)

i.e.

$$\eta_{vol} = \frac{Q_{total} - Q_{Leakage}}{Q_{total}} \tag{3.4}$$

where
Q_{total} is the total volume of air and oil present in port in $m^3$and
$Q_{Leakage}$ is the total leakage for air and oil in the compressor in m^3

3.4 Modified equations of the convergent nozzle.

As above work is related to nozzle and we do experiments regarding nozzle so analytical approach must be cleared by concept. By knowing the analytical approach we should know that what para meters are correspond to the system and which factor is affecting the orifice flow.

Nozzle have the design as there are sudden contraction from large diameter inlet to small one. There are some factors are affecting the flow through orifice like pressure ratio, density, co efficient of discharge and many more. We have equation for the mass flow rate is written below.

$$q_m = \frac{C_d}{\sqrt{1-\beta^2}} \epsilon \frac{\Pi}{4} d^2 \sqrt{2\rho_1 \Delta p} \tag{3.5}$$

As defined equation we have the diameter of inlet and outlet so that we have areas of different sections. Also we have property of gas and material, also for system we know the data of the pressure inlet and pressure outlet. By that we have the delta P and by all data we have discharge.

For the nature of discharge from different diameter outlet we applied here this equation for the different small gap diameter changes and we have the data as define.

Also we have the flow rate in terms of pressure and it would be as given.

$$Q = C_d A_0 Y \sqrt{\frac{2\Delta p}{\rho\left(1-\beta^4\right)}} \tag{3.6}$$

As we know that $\Delta p = \rho g \Delta h$, the equation in terms of h is,

$$Q = C_d A_0 Y \sqrt{\frac{2g\Delta h}{\rho\left(1-\beta^4\right)}} \tag{3.7}$$

Also here we have pressure loss and by that we can calculate the flow rate.

$$\Delta p = \frac{1}{2}\rho\left(\frac{Q}{C_d A_0 Y}\right)^2 \tag{3.8}$$

Here in present study we just check the behavior of the flow rate as the diameter changes. As by that we have data and we analyse that the flow rate would be increased exponentially and it would be low as diameter decreases.

Chapter 4

Computational investigation related to Experimental setup validate the boundy conditions.

In this project, finding out the leakages of the screw compressor is functionally more important to get the perfect scenario of the actual case and improved case. There for we are doing computational model on this problem. CFD is relatively easy, cheap and more reliable method for getting verify the actual case.

As getting many literature and research study. In part of this study the most important objective is make a tool with the help of CFD. To make the tool we have to understand the actual case and the actual working system. That should done by some researcher.

In the present study, the narrow gap between a rotors and the casing would be analysed. That gap should be in shape of either orifice or nozzle or slit. In while the meaning of this is the cross section between the rotors and the hub is those only. Simplification of the system is the easiest way to get appropriate results. Those cross sections is simplification of the whole area where leakages are in picture. Computational model for that large area is quite difficult and not reasonable task. In this study we can do first validation of the modelling. For validation we have refer that one paper which would be presented by W Zhao and the nanochannels should be analysed in this study. In this study we had done the simulation for the different gaps. The study itself a simplification of the appropriate problem. That would be shown by what geometry they had done experiments and done the simulation.

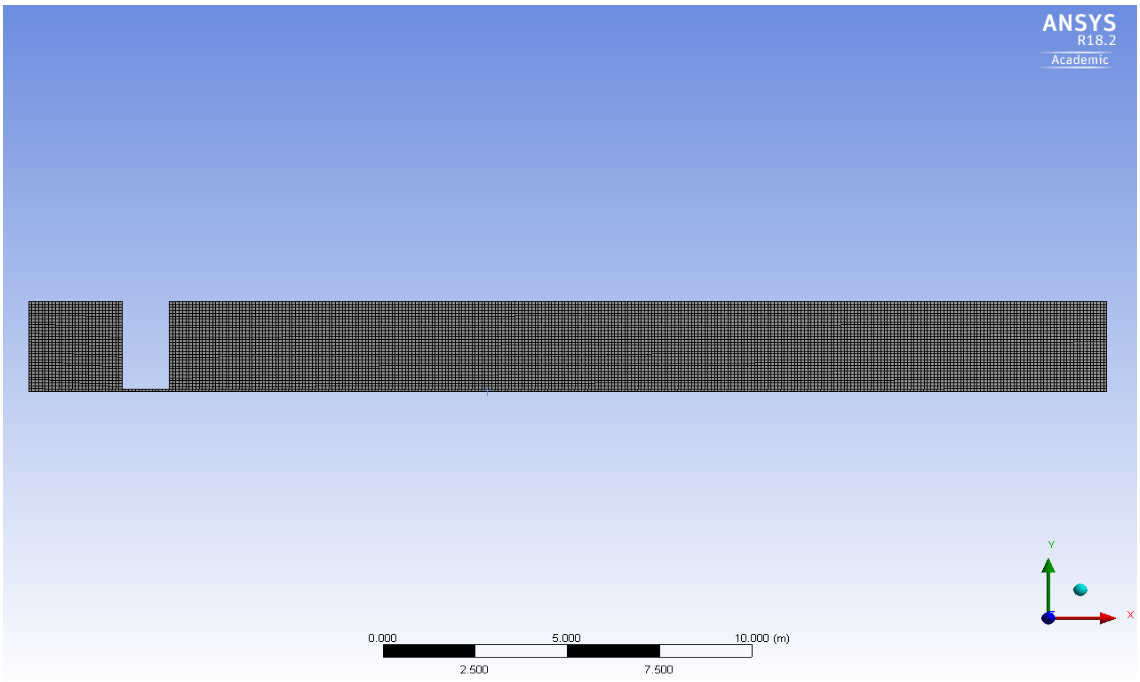

Figure 4.1: Computational grid of square channel.

The author of that study analysed the problem in actual case and design the geometry corresponding to that only. The Experiment had done to test the leakages as a function of the pressure difference and gap size. A plate with 108 mm in width and 50 mm in thickness would put inside of a square pipe to form a clearance gap. The height, width and lengths are 108 mm, 108 mm and 1150 mm respectively.

Figure shows the Ansys grid of the square pipe with clearance gap. A structure grid should be generated for the whole computational domain.

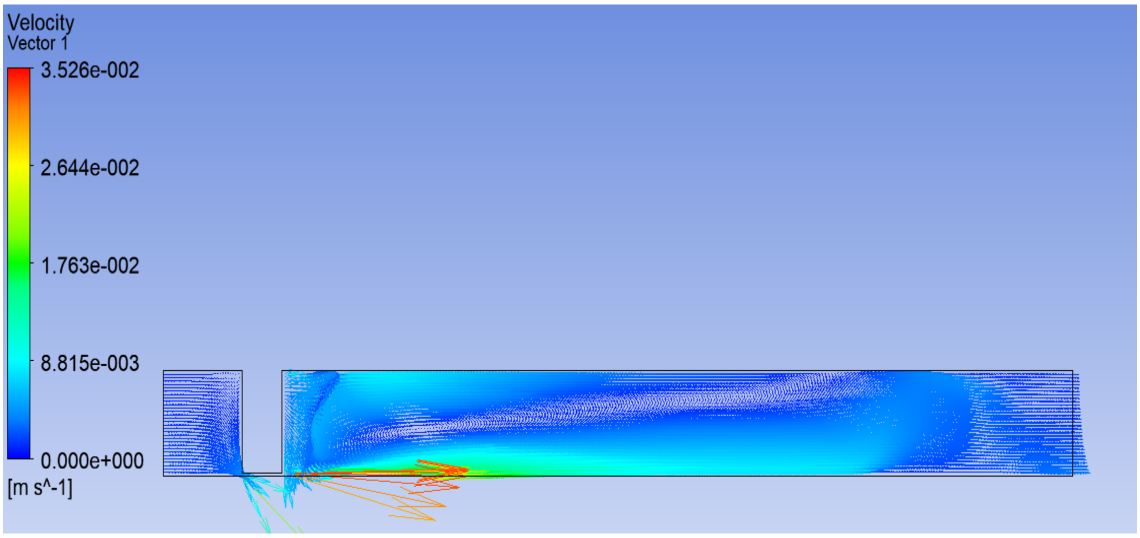

Figure 4.2: Velocity vector in ansys fluent

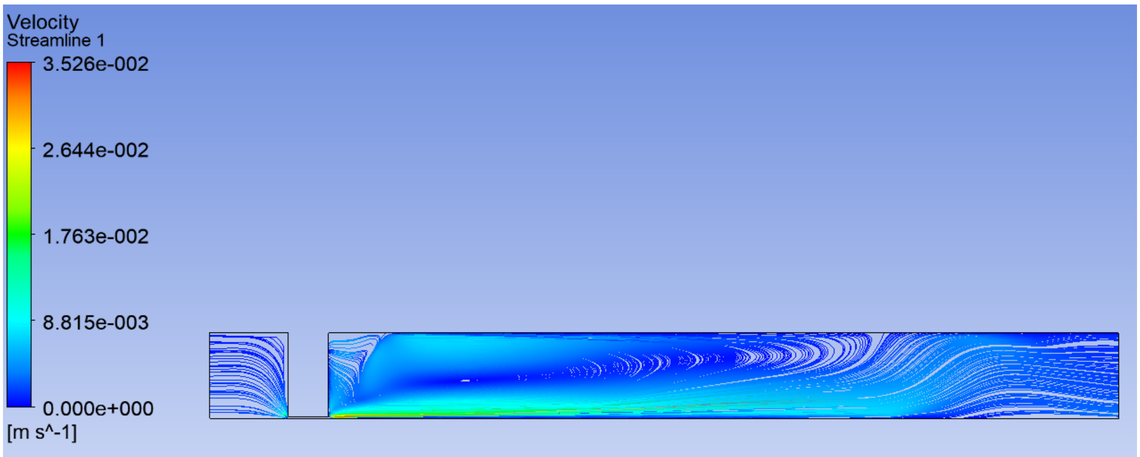

Figure 4.3: Stream line analysis in interior of geometry

Same experiment were completed with different gaps. Figure shows the simulation based on the commercial finite volume code, ANSYS FLUANT is employed. This method solves the Raynolds-average Navier-stokes on a grid. The total pressure is varied between 1 bar to 5 bar and on out let it would be 0.05 bar static pressure. Non slip boundary conditions are applied to up and down plates and other plates have the free slip wall. The maximum-cell based residual converge limitation would be 10^{-4}.

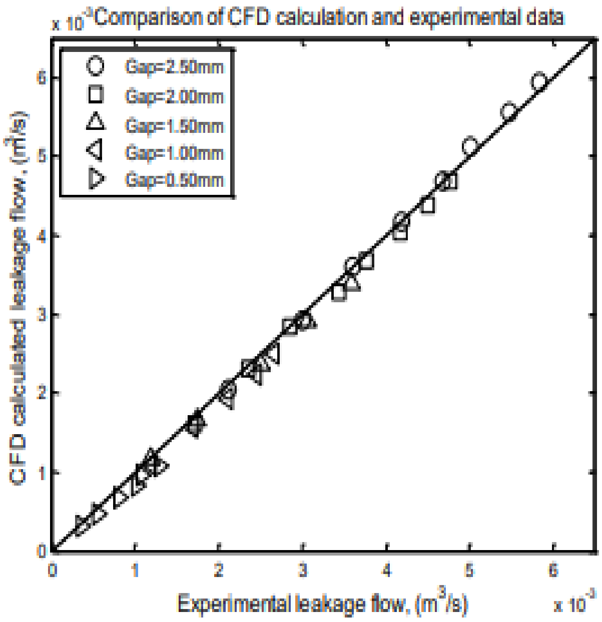

Figure 4.4: The parity plot of comparison of experimental leakage flow and predicted leakage flow by CFD

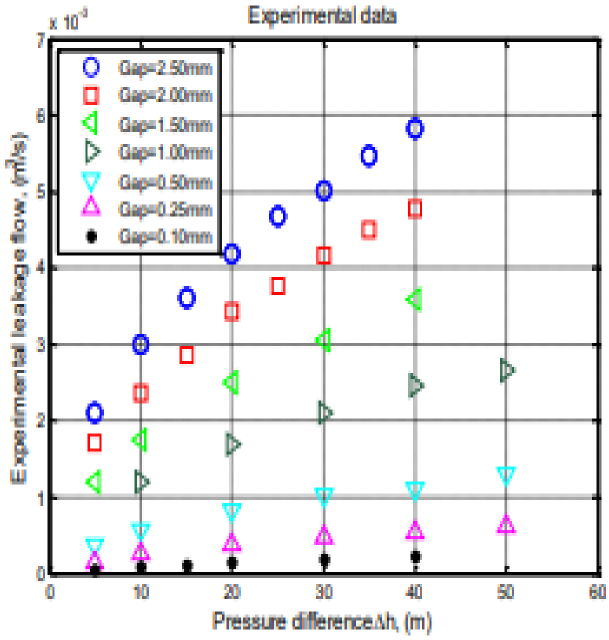

Figure 4.5: Experimental results for the seven different cases.

By this computational model some leakage data would be generated and that was directly compared to the data of the experimental data of the author and it was quite similar, and there was the difference of the readings between the experimental data and CFD data would be maximum difference of 11%.

Chapter 5

Computational investigation for nozzle massflow rate

On validate model of the simulation data should be analysed for the orifice gives the result and those data gave the similar result from the experimental data we have got in the experiments. Orifice, nozzle and nano channel gives the appropriate data but on the besis of leakages. Manufacturing of the orifice is comparatively less than the manufacturing of the nozzle. On the basis of the manufacturing cost first orifice would be analysed with the appropriate conditions as the actual case.

5.1 Computational model of 3d Nozzle

Simplification of the system would be give us a proper, simple, detailed understanding of the problem. For simulation we made a design of one simple channel. That give a proper idea about the problem. We did experiments and computer analysis on the same. That gave us the data from experiment and from the CFD. Then we compared all the data of the different diameter channels. By that we got the exact logic about the CFD. And so on we can get a validated tool for the nano channels.

After doing this we can do for smaller dimension geometry because we have to experiments too. Micro manufacturing is some what costlier. There for on first we can do for nano channels.

5.1.1 Geometry for simulation.

As discussed, a geometry would designed for actual case and here one nano channel would be designed for further simulation. Design would manufactured and experiments had performed on same design.

Design consideration would be changed as we consider different diameters and the different aspect of actual case. Here we have clearance gap between the rotor to rotor and rotor to case. So we take the geometry of slit, orifice and nozzle.

5.1.2 Meshing of geometry.

In this study, aim of doing CFD is getting the same result near to the experimental data. To get that reasonable result we do meshing of above geometry in the ansys meshing tool. Meshing is done in the physical preference of CFD and solver problem is fluent. This meshing is based on the curvature and proximity size function.

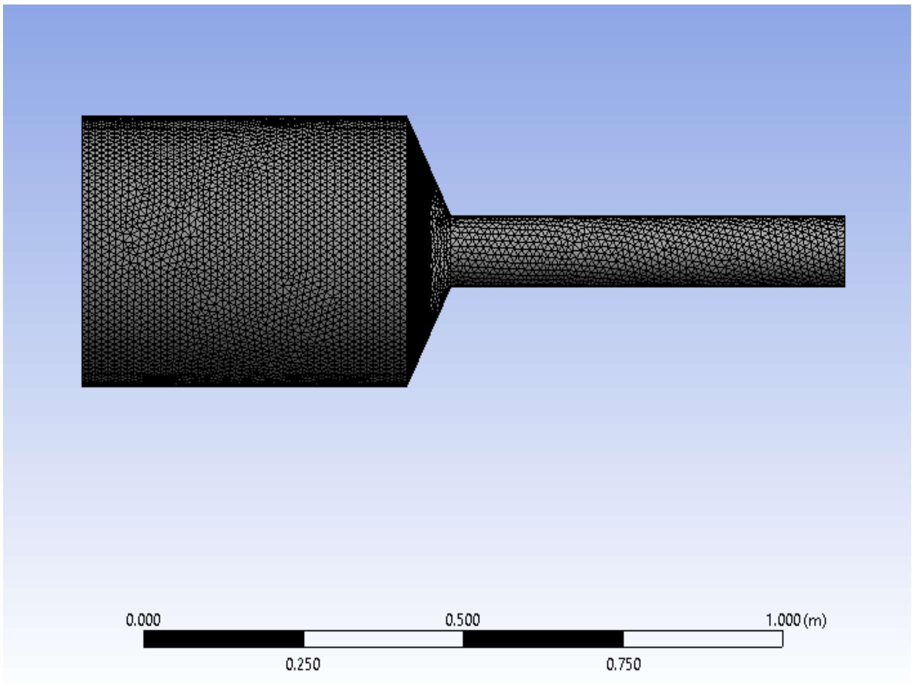

Figure 5.1: meshing that would be used in fluent

This meshing is done with the help of the sizing and inflation. 4,29,000 elements are placed in the mesh, from these all half of the element is used for the wall inflation because we need to observe the movements and heat transfer on the wall and in the turbulence.

5.1.3 Computational study and results

In this study we are doing some analysis of the element which has been simulated in the system. We are put some boundary conditions as actual case is having the working medium. Here we are giving the working fluid as air and the solid as the carbon steel. System had done with the boundary condition as the pressure inlet and pressure outlet are given pressure and 1bar pressure respectively. Working model is the k-epsilon for turbulence and wall boundary condition is no slip. And we do steady simulation on the above data.

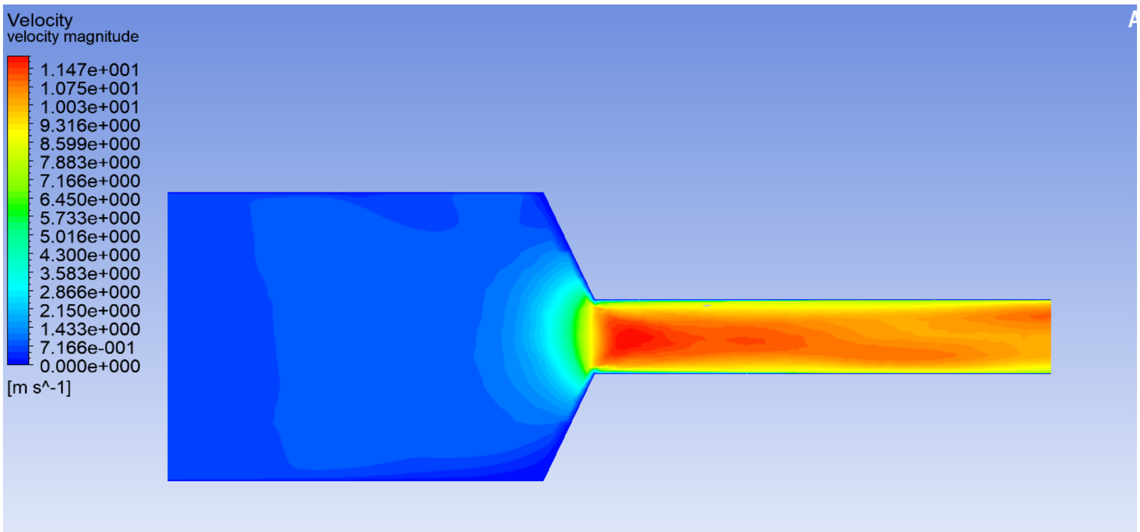

Figure 5.2: Velocity magnitude in the channel

As the result of the pressure difference there should be some pressure flow through the system. Main task would be finding out the mass flow rate from outlet. By this system we would found the data for different pressure and different geometry. That data will directly compared to the data of the experiments those data would tell us about the simulation is accurate or not.

As we have the data for leakages form outlet. We also find the data by experiments and analytically. Those data would be directly compared to the experiments.

5.2 Simulation of the 2D Slit flow

In the present study, we are doing work on the blowhole leakages and the interlobe leakages. When it is interlobe leakage, the area from where the leakage is taking place is throughout the sealing line length and in-between the clearance gap. If this area is simplified then cross section should be rectangular.

Now here all work is done for getting the relation between nozzle flow and rectangular channel flow. As we know that experimental set of the nozzle and circular channel if quite simple than the rectangular channel. An experimental set of a rectangular channel having a very thin width and very large length is more difficult and very expensive.

For Rectangular channel flow, there are some specific changes in the model of simulation. These changes are described here in the below explanations.

In the present work, the length and width of the rectangular channel are taken widely but the model of a computational study is keeping the same as described. As early mentioned that we are doing work for getting the equilibrium rectangular channel to the nozzle. The model of the computational study is described below in this the details of the meshing, post and pre-processing is also been there.

The mesh of the 2d model which was simulated have some specific preferences and these all are described below. Physical preference is CFD then solver preference is fluent. Size function should be the adaptive and relevant center is fixed to the coarse. There are edge sizing is done to all four sides by that meshing size would be controlled. In this model, fine meshing is must be there in the solver by that we can get more appropriate models. As having face mashing tool, here it should help to get more fin mesh. So face meshing is applied to the face of 2d geometry. As per the above characteristics, we got the mesh in the channel size 47*2.04mm is shown below.

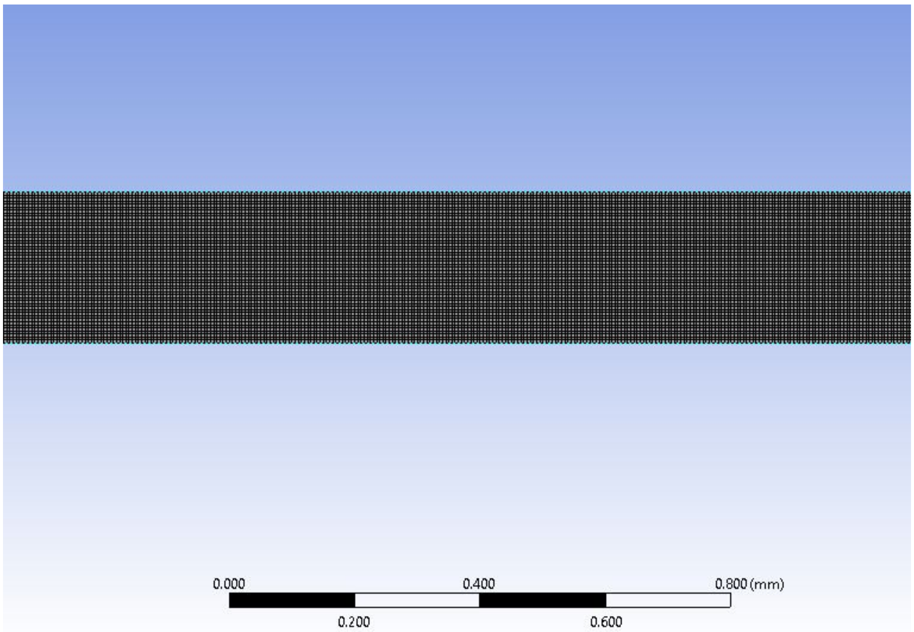

Figure 5.3: Meshimg of the rectangular channel in 2d simulation tool

In the pre-processing there are several assumptions and conditions are defined on that basis, a model is working. In set up general settings are solver type is set up to pressure based. Models are selected as energy equation is on the viscous model is laminar with low-pressure pressure slip condition. In the material selection, it should be air, and density model is set up to ideal gas. The most important factor of the editing of this computational modeling is Thermal Momentum Accommodation Coefficient and this would be set to 0.90 for the microchannel flow. This all data would be set up to the Actual case only.

Now next step is setting up the boundary conditions. There are an inlet, outlet and the wall conditions are there in the model. Inlet boundary conditions are fixed as the static gauge pressure is 6bar and outlet pressure is 1 bar. As different takes are done on the same nozzle, where the inlet pressure is changed. The inlet pressure is set up to different range like 7 bar to 1 bar then convergence criteria is set up to the 10^{-4} in the monitors and in monitors it would be residuals. Initialization would be done with the reference of the inlet conditions.

This pressure-based system is working under the steady solution method and then in the run calculation tab number of iterations would give it to the system. Results are shown in the post-processing tab after completing this calculation.

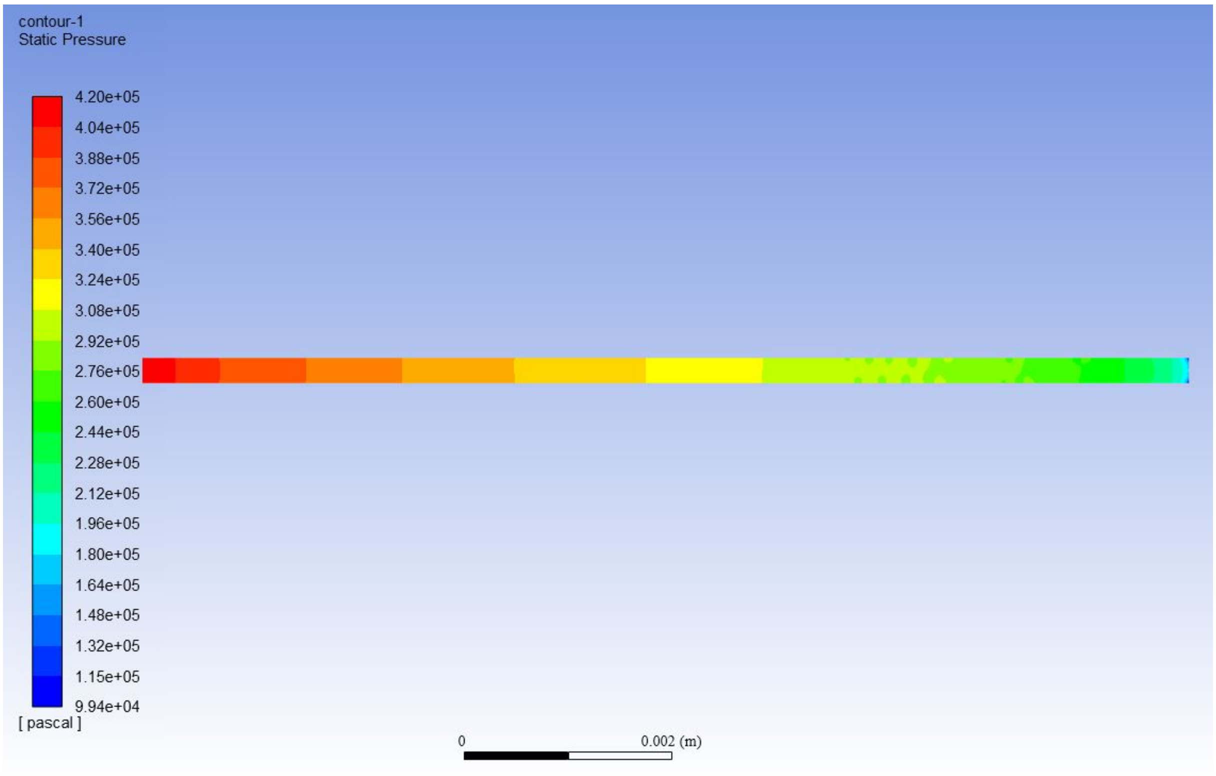

Figure 5.4: Pressure contour of the results 2d rectangular channel

In the above figure, the pressure contour is shown that this is the laminar with transient flow occurs in the microchannels. Even pressuere contour is showing the flow characteristics of the fluid. Here Area-weighted average velocity also be plotted in the post-processing and by that, we can cross check the flow rate by the equation $m = \rho A u$.

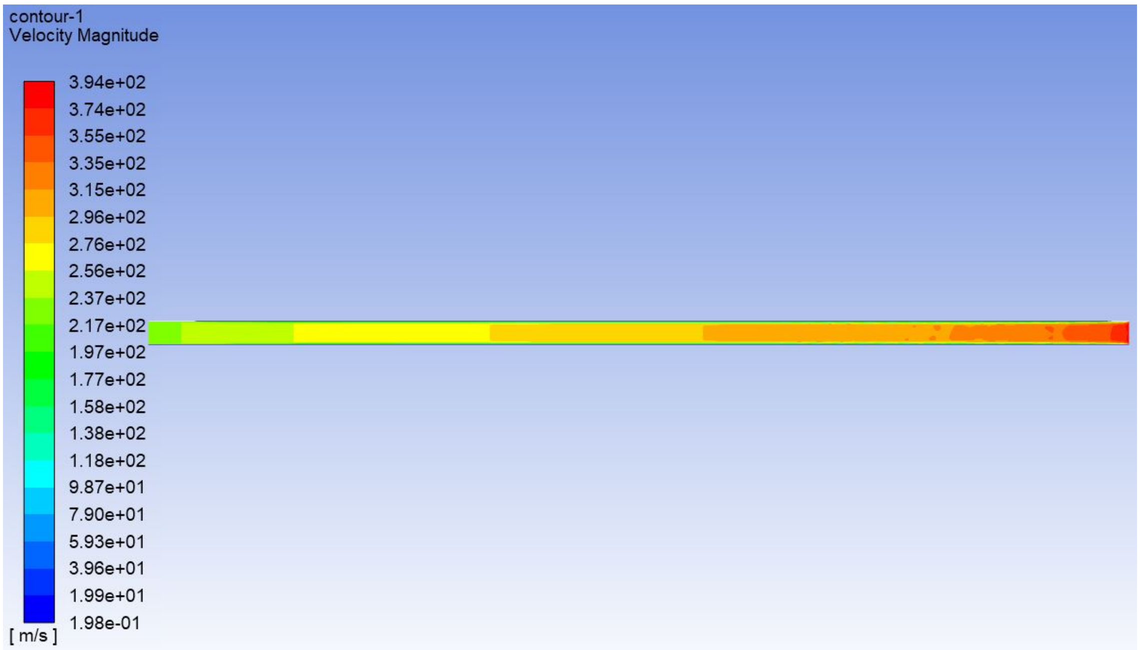

Figure 5.5: Velocity magnitude contour of the results

In the post-processing, we can find the velocity diagram and many more things which would be helpful to find out the system is working well or not? Next figure shows the velocity magnitudes by that we can assume the exact flow characteristics of the flow.

Chapter 6

Experimental studies

In this study as the assumption for the actual case and as CFD would be done for the case where we put a channel in between the pressure ratio. In that experiment channel had situated as the element between two reservoirs. One of this reservoir was sated on 7 to 3 bar as we change the pressure to 3 to 7 bar. Second one was sated on the atmospheric pressure.

Now, system was been on steady state condition then after we opening the valve for the system and by pressure difference the leakages should take place in system. In that there were pressure difference of different pressure gives different results and we simulate the same condition.

Data for the experiments is given below in the table. In next chapter we do direct comparison to CFD data and that would be as given in next chapter.

In experiment, there are 2 reservoirs and both are maintained at constant pressure by special arrangement of the instruments like compressors, valves, pressure gauges etc. both the reservoirs had connected by special leak proof arrangement. In between the system there is a part on which we have to do experiments. As a starting phase we are putting out the part by valve mechanism and that should be done to check that system is in steady state. After confirming that valve to the part which would be experimented is going to be open. That leakage number gives the amount of leakage from system.

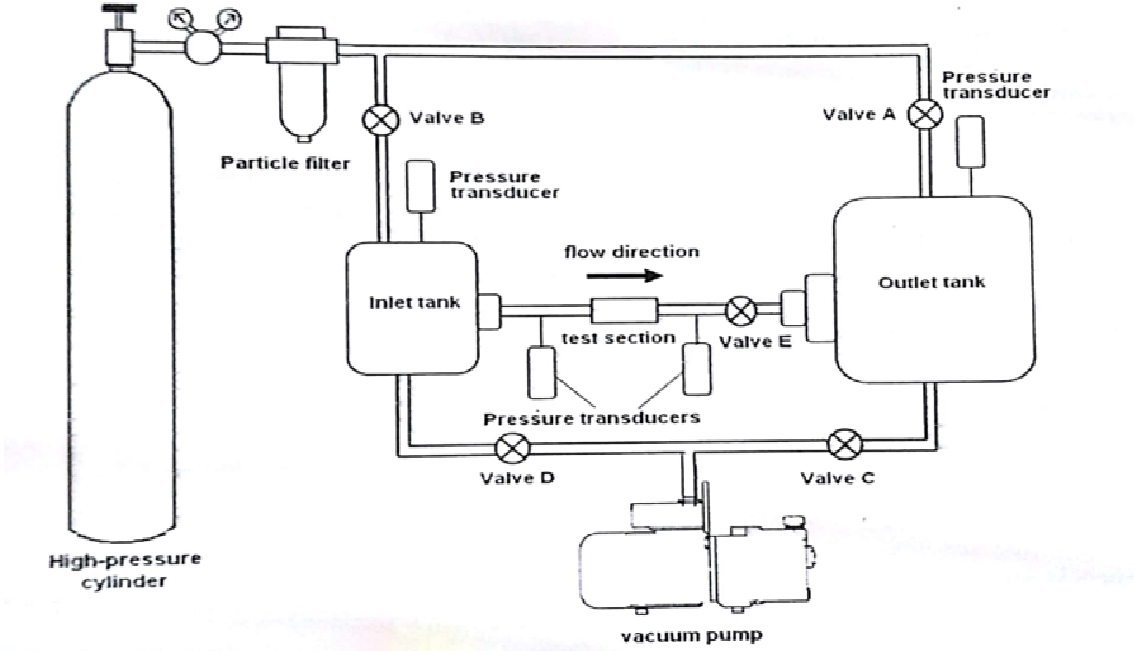

Figure 6.1: Details of experimental Setup

Chapter 7

Results and Discussion

In present study we had got data from both experiment and from CFD. Now as the comparison gives the complete perspective about the CFD tool. In experiment we are taking care about the other leakage and achieve the almost 97% accuracy.

Experimental, computational and analytical data has been generated in the present study. Micro-nozzle is subjected to the testing geometry. First, we do Analytical Modeling about to the Neno and Micro Channel. In the first chapter, some equations are mentioned for calculating the amount of leakages. As there are given data of the geometry and some properties are known. We can calculate the mass flow rate from the nozzle.

In the present modeling, pressure of the inlet and outlet would be known and here we take the different pressure on the inlet and keep outlet pressure constant. As we have some modified neno and micro nozzle equations, calculation of the mass flow rate is done.

Inlet Diameter(mm)	Outlet Diameter(mm)	Upstream pre.(Pa)	Coefficient of discharge	β (d/D)	Q_m(kg/s)
15.5	3	700000	0.60998	0.193	0.01024
15.5	3	650000	0.61005	0.193	0.009591
15.5	3	550000	0.61023	0.193	0.00828
15.5	4	700000	0.60936	0.258	0.01993
15.5	4	650000	0.60942	0.258	0.01866
15.5	4	550000	0.60956	0.258	0.01611
15.5	4.5	700000	0.60928	0.290	0.02522
15.5	4.5	650000	0.60933	0.290	0.02361
15.5	4.5	550000	0.60946	0.290	0.0203
15.5	5	700000	0.60932	0.322	0.03116
15.5	5	650000	0.60937	0.322	0.02917
15.5	5	550000	0.60949	0.322	0.02519
15.5	5.5	700000	0.60945	0.354	0.03774
15.5	5.5	650000	0.60950	0.354	0.03533
15.5	5.5	550000	0.60963	0.354	0.0305
15.5	6	700000	0.60968	0.387	0.04499
15.5	6	650000	0.60974	0.387	0.04258
15.5	6	550000	0.60986	0.387	0.03928

Table 7.1: Results of the analytical method

In the present study, the coefficient of discharge is the most important factor for calculating the exact flow. Hence some relations between the beta ratio and inlet diameter.

Here we have some equations of beta ratio and inlet diameter. These all are for nozzles and this is derived

$a = \sqrt{\frac{(1-\beta)^4}{Re}}$

$b = \beta^4 + 4\beta^{16}$

$c = (16.48 - (\frac{1.16}{D}))$

$d = \frac{0.52}{D}$

These are the correlations between the β and D. Now this all correlations are given one relation that would give a constant C_1.

$$C_1 = (d - 0.192 + (b * c)) * a \tag{7.1}$$

Here again, we have four sets of equations and this will give the equations for another constant C_2.

$A = \sqrt{1 - \beta^4}$

$b = \beta^4 + 2\beta^{16}$

$C = 0.3155 + \frac{0.0175}{D}$

$D = \frac{0.0044}{D}$

$C_2 = (0.5991 + D + C * B) * A$

Addition of this both constant is giving us the coefficient of the discharge. Coefficient of discharge is dependent on the beta ratio, diameter of inlet and the Reynolds number of the flow. This amount of coefficient of discharge would be given the mass flow rate and the results are very near about to experimental and computational results.

As per the above computational model, computational calculations should be performed on the different nozzles and the different input parameters. Also, these data would be very near about to experimental data and the analytical data.

In the computational work, plotting of the area-weighted average is used for velocity and we have the mass-weighted average density. By that, we are doing calculations of the mass flow rate from the outlet.

Inlet Diameter(mm)	Outlet Diameter(mm)	Upstream pre.(Pa)	β (d/D)	Q_m(kg/s)
15.5	3	700000	0.193	0.01079
15.5	3	650000	0.193	0.01097
15.5	3	550000	0.193	0.00879
15.5	4	700000	0.258	0.01901
15.5	4	650000	0.258	0.01812
15.5	4	550000	0.258	0.01586
15.5	4.5	700000	0.290	0.02456
15.5	4.5	650000	0.290	0.02248
15.5	4.5	550000	0.290	0.01947
15.5	5	700000	0.322	0.02875
15.5	5	650000	0.322	0.02704
15.5	5	550000	0.322	0.06087
15.5	5.5	700000	0.354	0.03689
15.5	5.5	650000	0.354	0.03356
15.5	5.5	550000	0.354	0.02965
15.5	6	700000	0.387	0.04158
15.5	6	650000	0.387	0.1028965
15.5	6	550000	0.387	0.0895265

Table 7.2: Results of the Computational model

By this, we are doing an exact comparison of the mass flow rate in the microchannel. This would be giving how computational model is accurate to the actual model.

In the present study, we have to conclude the exact computational modeling and the analytical modeling of the actual conditions. By having the exact flow rate in the mini-nozzle. We create the rectangular channel equilibrium to these nozzles. By doing that we are going to the rectangular channel of very small width and larger length.

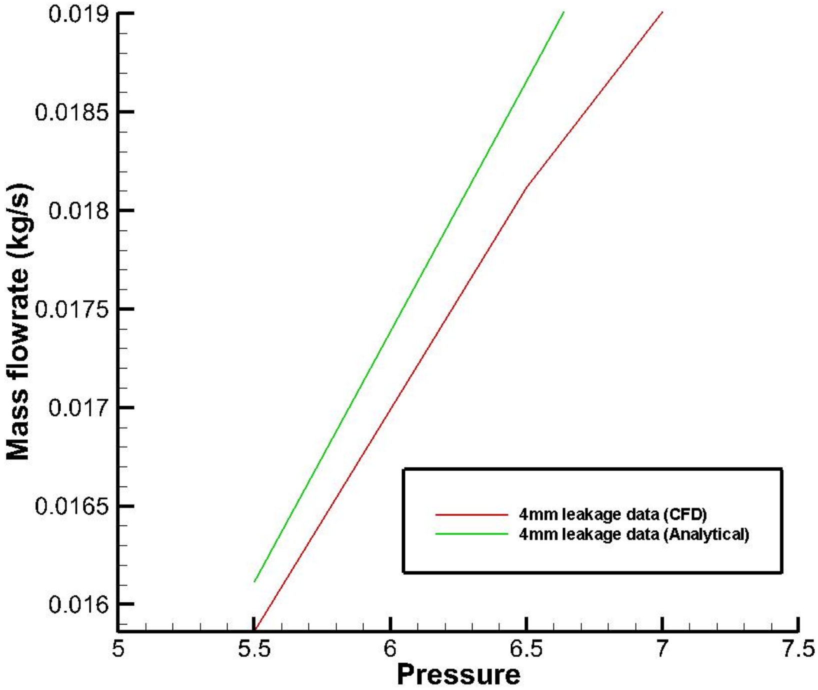

Figure 7.1: Comparison of Mass flowrate with different pressure in 4mm nozzle.

In the present work, we have computational data, experimental data and the analytical data. In the next paragraph, comparision of the data would be compared and those all are as below. All data would be compared and results would be in the range of 7%.

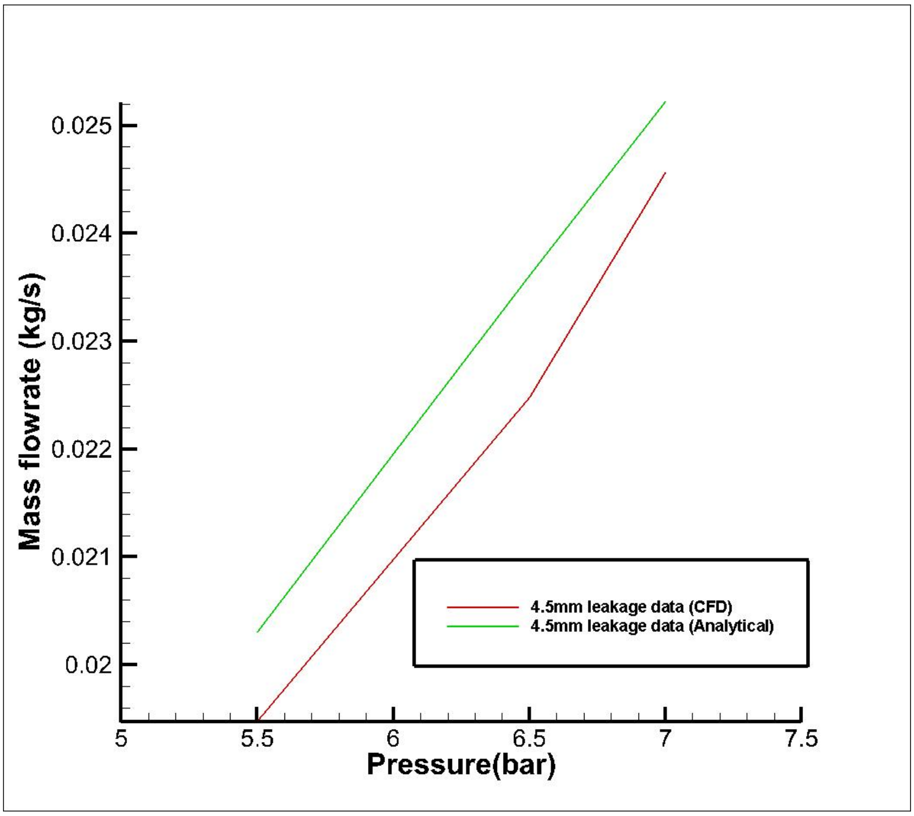

Figure 7.2: Comparison of Mass flowrate with different pressure in 4.5mm nozzle.

In next graph we compare data for the 5.5mm nozzle and this would also be in the same nature of the curv for the 4.5mm so it should show that some more accuracy about the model.

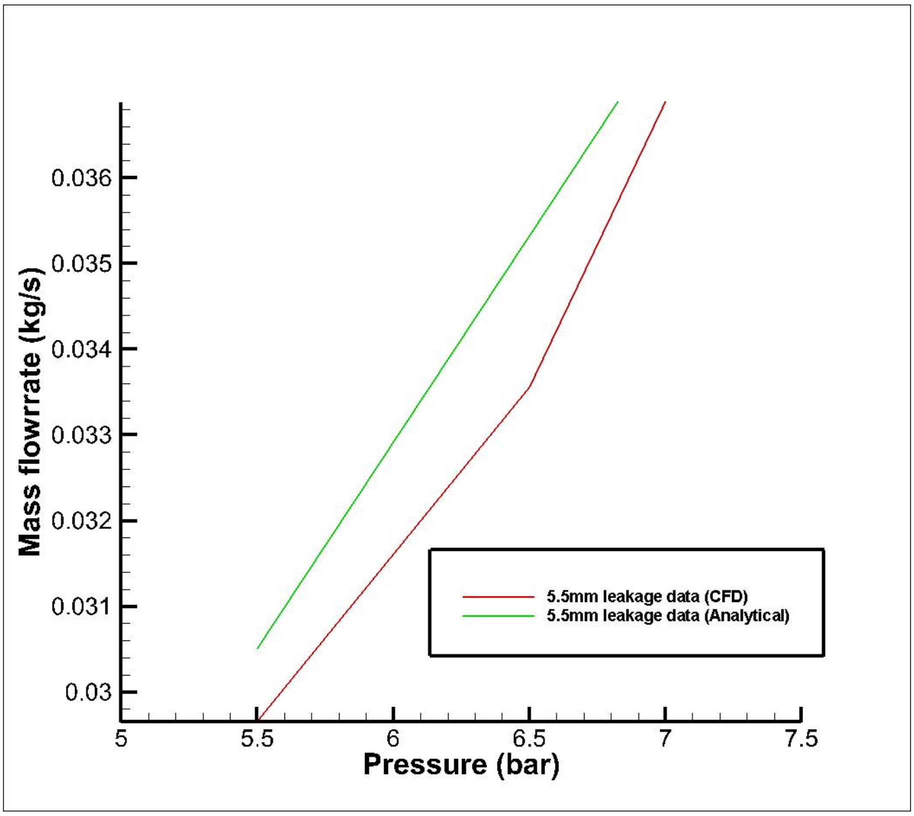

Figure 7.3: Comparison of Mass flowrate with different pressure in 5.5mm nozzle.

In these graphs, comparision is showing that the percentage change in the result of the computational data and analytical data is Around 2 to 6. This would suggest that computational work is done for further work with slits and microchannels.

In the present study, we have one computational validated tool which helps us to work further with microchannels and furthermore flow component.

Next step for the present work is, the results of the nozzle is directly compared to the results of the rectangular channel by hydraulic diameter equations and for equivalent diameter. Even in the modified equations of the equilibrium diameter.

By Darcy's equation, we have the hydraulic diameter equations and equivalent diameter equations. These equations are widely used in the refrigeration industries. We have done computational work on the particular 4mm with 7bar inlet pressure.

$$D_h = \frac{4ab}{2(a+b)} \tag{7.2}$$

$$D_e^* = 1.30(a*b)^{0.625}/(a+b)^{0.25} \tag{7.3}$$

$$D_e^{**} = 1.453(a * b)^{0.6}/2(a+b)^{0.2} \quad (7.4)$$

By this perticular equestion, computational data is working on the fixed 4mm nozzle and on the results are as below. We are doing CFD on different equations which are used in the different types of the research paper.

In the below table, we have equilibrium channel for the nozzle and its dimensions are given and the flow rate from that particular rectangular channel is compared with the experimental data.

D_e^*	a	b	mass flowrate
3.00	0.2	97	0.01921
4.00	0.2	199	0.03942
5.00	0.2	348	0.06894
3.00	0.18	120	0.01879
4.00	0.18	235	0.03858
5.00	0.18	420	0.06944
3.00	0.22	84	0.01758
4.00	0.22	165	0.03981
5.00	0.22	310	0.068007

Table 7.3: Calculated flow based on the equilibrium diameter $D_e$with different thickness.

Here we are showing some trend in the rectangular microchannel and the flow is similar in all equilibrium channel but in comparison with the experimental data it would be 30% more in the rectangular channel.

Now, we are doing the simulation on the basis of the other equation of the equilibrium diameter equation. The comparison of this would give us the exact perspective of flow rate in the rectangular channel and amount of mass flow rate.

D_e^*	a	b	mass flowrate
3.00	0.2	97	0.01171
4.00	0.2	199	0.02345
5.00	0.2	348	0.03014
3.00	0.18	120	0.01295
4.00	0.18	235	0.02045
5.00	0.18	420	0.03789
3.00	0.22	84	0.01558
4.00	0.22	165	0.02281
5.00	0.22	310	0.02689

Table 7.4: Calculated flow based on the equilibrium diameter $D_e$with different thickness.

Here we analyze the data and find some difference in the leakages in the mass flow rate. here right in this result, we got some less error in mass flow rate but it would be increased by the time of increasing the value of the equilibrium diameter. In future work, we have to generate one coefficient by the regression method.

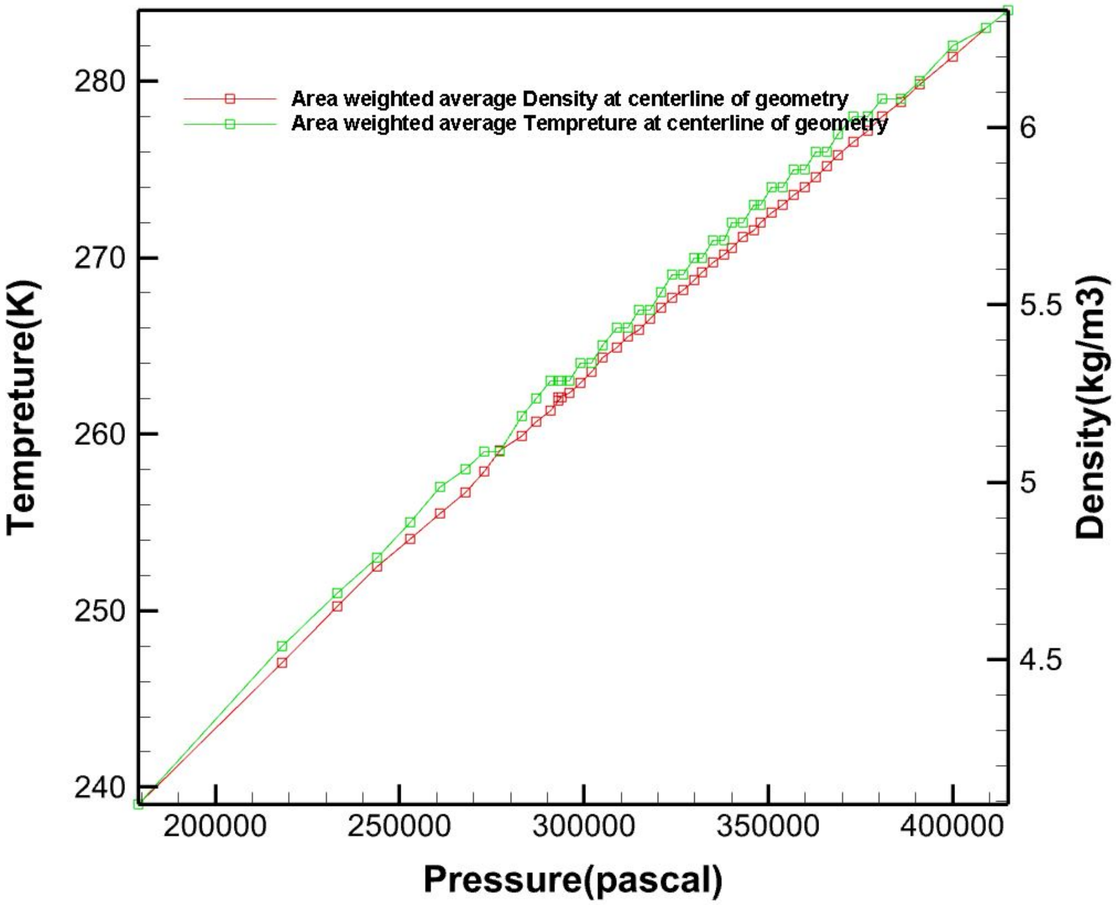

Figure 7.4: Graph of pressure Vs density and temprature.

In this graph, pressure vs temprature and density is shown and both the curv is almost the same nature. This would indicate that out computational model is following the true nature of the thermodynamics.

Chapter 8

Conclusions

In present study, how the leakages behave in actual case and in what amount is observed. Present study also gives the perticular amount of leakages by computational model. For this analysis, ansys and some theories utilized for the understanding the problem. The conclusion drawn from the present study include the following:

The conclusion get from present study are as following.

1. The performence of screw compressors are affected by leakages. The factors affecting leakages in screw compressors includes pressure ratio, temperature, diameter of nozzle, density of lubricating oil, geometry of compressors, speed of rotor and clearance provided to various location.
2. A CFD modelling of the gap inside the screw compressor is undertaken for simplifying and understanding the various leakages.
3. The variation between the cfd analysis of 2D rectangular micro-channel and 3D mini-channel for simulating the leakages with in the interlobe leakages and blowhole leakages respectively varies with the experimental results with in $\pm 12\%$. In the simulation of 2D rectangular micro-channel, considering TMAC (Thermal momentum accomodation coefficiant) as 0.9 provides good agreement with experimental results.
4. The experimental results related to leakages have verified with CFD results.
5. The Darcy equation for equivalent rectangular channel is improved from computational studies conducted.

Chapter 9

Future scope

1. CFD analysis using moving and dynamic mesh may be undertaken for further study of leakages.
2. The experimental results related to leakages may be varified with CFD results of equivelent channel.
3. The analytical and computational modelling may be used to develop tool to predict leakages with in the screw compressor.

Bibliography

[1] wikipedia.org. [Online]

[2] Numerical simulation of a twin screw expander for performance prediction by Iva Papes, Joris Degroote, Jan Vierendeels. Department of Flow, Heat and Combustion Mechanics, Ghent University, Belgium E-mail: iva.papes@ugent.be

[3] Numerical simulation and experimental study on the performance of screw expander by Wang, Zhigang Zhang, Yufeng Sun, Yuexia Wei, Lili (2010). ASHRAE Transactions.

[4] Rotor Geometrical Parameters in Refrigeration Helical Twin Screw Compressors(1996). International Compressor Engineering Conference. Paper 1074. http://docs.lib.purdue.edu/icec/1074

[5] Singh, P. J. and Bowman, J. L., Effect of Design Parameters on Oil-Flooded Screw Compressor Performance (1986). International Compressor Engineering Conference. Paper 517. http://docs.lib.purdue.edu/icec/517

[6] Boldvig, v. and Villadsen, V., A Balanced View of Reciprocating and Screw Compressor Efficiencies (1980). International Compressor Engineering Conference. Paper 350. http://docs.lib.purdue.edu/icec/350

[7] Koelet, P. Industrial Refrigeration, Principles, Design and Applications. s.l. : Mc Graw Hill, 1992.

[8] The analysis of leakage in a twin screw compressor and its application to performance improvement. Tang, J S Fleming and Y. 1995, Journal of Process Mechanical Engineering, pp. 125-136.

[9] Performance Analysis of Oil Injected Screw Compressor and its Application. Fujiwara, M., Osada, Y. 1995, International Journal of Refrigeration, pp. 220-227.

[10] Numerical simulation of a twin screw expander for performance prediction by Iva Papes, Joris Degroote, Jan Vierendeels. Department of Flow, Heat and Combustion Mechanics, Ghent University, Belgium E-mail: iva.papes@ugent.be

[13] Papes, Iva; Degroote, Joris; and Vierendeels, Jan, Analysis of a Twin Screw Expander for ORC Systems using Computational Fluid Dynamics with a Real Gas Model (2014). International Compressor Engineering Conference. Paper 2350. http://docs.lib.purdue.edu/icec/2350

[14] 3D CFD analysis of a twin screw expander. A Kovacevic, S Rane City University London, Centre for Positive Displacement Compressor Technology, UK ABSTRACT

[15] CFD simulation of a screw compressor including leakage flows and rotor heating August 2015IOP Conference Series Materials Science and Engineering 90(1):012009 DOI: 10.1088/1757-899X/90/1/012009

[17] Numerical simulation and experimental study on the performance of screw expander.(Report) ASHRAE Transactions July 1, 2010 | Wang, Zhigang; Zhang, Yufeng; Sun, Yuexia; Wei, Lili | Copyright

[18] Modelling of complex clearance flow in screw-type machines . Vimmr, Jan. 2007, Mathematics and Computers in Simulation, pp. 229–236.

[19] CFD studies on flow through screw compresor. Sola Avinash, Meesala Nagarjuna and P Ganna Teja, International Journal of Mechanical Engineering Research and Technology, Vol. 1, No. 2, November 2015.

[20] Optimization of Screw Expanders for Power Recovery From Low-Grade Heat Sources. MATTHEW READ, NIKOLA STOSIC and IAN K. SMITH. 2014, Energy Technology & Policy (2014).

[21] Numerical Simulation and Performance Analysis of Twin Screw Air Compressors W.S. LEEa'*, R.H. MAb, S.L. CHENb, W.F. WUb and H.W. HSIA, Department of Air Conditioning and Refrigeration, National Taipei University of Technology, Taipei, Taiwan 106, ROC,

[22] Rotor Profile Design for Twin Screw Compressor P.Jenno Xavier, K.Kanthavel, R.Uma Mythili. International Journal of Scientific & Engineering Research Volume 2, Issue 7, July-2011 1 ISSN 2229-5518

[23] GEOMETRY OF SCREW COMPRESSOR ROTORS AND THEIR TOOLS Nikola Stosic, Ian K Smith, Ahmed Kovacevic and Elvedin Mujic Centre for Positive Displacement Compressors City University London, Northampton Square, London EC1V 0HB, United Kingdom

[24] The analysis of leakage in a twin screw compressor and its application to performance improvement J S Fleming, BSc, PhD, CEng, FIMechE, MInstR and Y Tang, BS,

MSc Division of Dynamics and Control, Department of Mechanical Engineering, University of Strathclyde, Glasgow, Scotland