

# **CFD ANALYSIS OF NASA ROTOR 67 WITH AND WITHOUT BOUNDARY LAYER CONTROL**

**A MAJOR PROJECT REPORT**

SUBMITTED IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE AWARD OF  
THE DEGREE OF

**BACHELOR OF TECHNOLOGY**  
(Aerospace Engineering)

**SUBMITTED TO**  
**UNIVERSITY OF PETROLEUM AND ENERGY STUDIES, DEHRADUN**

**BY**

Name of Students  
Ajeet Sarkar  
Bhavesh Singla  
Navneet Kumar

University Roll No.  
R290211003  
R290211005  
R290211013

**SUPERVISED BY**  
**MR. KARTHIK SUNDARRAJ**  
Assistant Professor

April 2015



**COLLEGE OF ENGINEERING STUDIES**  
**UNIVERSITY OF PETROLEUM AND ENERGY STUDIES, DEHRADUN**

## **CERTIFICATE**

I hereby certify that the work which is being presented in the B.Tech. Major Project Report entitled “**CFD Analysis of NASA Rotor 67 with and without Boundary layer control**”, in partial fulfillment of the requirements for the award of the **Bachelor of Technology in Aerospace Engineering** and submitted to the Department of Aerospace Engineering, COES, University of Petroleum and Energy Studies, Dehradun is an authentic record of my own work carried out during a period from **July 2014 to December 2015** under the supervision of **Karthik Sundarraaj & Assistant Professor, Aerospace Department.**

The matter presented in this Project Report has not been submitted by me for the award of any other degree elsewhere.

### **Submitted By:**

Ajeet Sarkar [R290211003]

Bhavesh Singla [R290211005]

Navneet Kumar [R290211013]



This is to certify that the above statement made by the student(s) is correct to the best of my knowledge.

**Date: 13 April 2015**

**Karthik Sudarraaj  
Assistant Professor**

**Aerospace Department**

**Om Prakash**

**Head**

**Aerospace Department**

## **ACKNOWLEDGEMENT**

I would like to place on record my deep sense of gratitude to Prof. Om Prakash, HOD-Dept. of Aerospace Engineering, UPES, Dehradun, India for his generous guidance, help and useful suggestions.

I express my sincere gratitude to Prof. Karthik Sundarraj, Dept. of Aerospace Engineering, for his stimulating guidance, continuous encouragement and supervision throughout the course of present work.

I also wish to extend my thanks to Prof. Rajesh Yadav and other colleagues for attending my seminars and for their insightful comments and constructive suggestions to improve the quality of this project work.



Ajeet Sarkar [R290211003]

Bhavesh Singla [R290211005]

Navneet Kumar [R290211013]

## ABSTRACT

The aim of this project is to carry out an optimization of NASA Rotor 67 blades to maximize pressure ratio and isentropic efficiency. This is achieved by modifying or controlling boundary layer to prevent losses and this is done by introducing suction and blowing layer on rotor blades. NASA rotor 67 is among the most popular rotors that have been manufactured to date. It was designed and tested in early 1980's by Reid and Moore at NASA Glenn Research Center . It consists of 22 blades with multiple arc profiles and having isentropic efficiency of 0.8916 and pressure ratio of 1.5558. The well-known test case NASA Rotor 67 is numerically tested for the application of aspiration. Aspiration has been applied to both the blade surface as well as the endwall boundary layers.

Detailed flow field analysis of rotor blade is done with the help of Computational fluid dynamics (CFD) predictions obtained by ANSYS-CFX 14.5 or FLUENT with k- $\epsilon$  turbulence model, and compared with experimental data of Rotor 67 blades which has been taken by NASA.

In order to understand the possible benefits of boundary layer control, a meanline parametric study over both conventional and aspirated fan has been done. Viscous, secondary and shock losses has also be estimated for conventional and aspirated fan stage. Previous researches suggest that aspiration can have largest impact on a fan performance.

## LIST OF FIGURES

<b>Figure</b>	<b>Title</b>	<b>Page</b>
1.1	NASA Rotor 67	2
1.2	Estimation of losses	3
1.3	Shock Wave Configuration Inside Rotor	4
2.1	NASA Rotor 67 Blades	6
3.1	Model of NASA Rotor 67	9
3.2	Model of Single Rotor Blade	10
3.3	Import Model in ICEM	11
3.4	Creating Parts in ICEM	11
3.5	Steps in Creating Parts of the Geometry	12
3.6	Model of Single Blade with Hub	12
3.7	Steps to Define Meshing Parameters	13
3.8	Defining Prism Parameters	14
3.9	Defining of Part Mesh Setup	14
3.10	Meshing of Model	15
3.11	Meshing of Single Plate and Hub	15
3.12	Opening Method for Turbo Mode in CFX-Pri	17
3.13	Classification of Bibliographic Literature	21
3.14	Meshing of Suction Method	22
3.15	Meshing of Blowing Method	24
3.16	Meshing of Suction and Blowing Method	27
4.1	Velocity Vector at 20% Span	30
4.2	Velocity Vector at 50% Span	31
4.3	Velocity Vector at 80% Span	31
4.4	Suction Velocity Vector at 20% Span	32
4.5	Suction Velocity Vector at 50% Span	33
4.6	Suction Velocity Vector at 80% Span	33
4.7	Blowing Velocity Vector at 20% Span	34
4.8	Blowing Velocity Vector at 50% Span	35

4.9	Blowing Velocity Vector at 80% Span	35
4.10	Suction and Blowing Velocity Vector at 20% Span	36
4.11	Suction and Blowing Velocity Vector at 50% Span	37
4.12	Suction and Blowing Velocity Vector at 80% Span	37



## LIST OF TABLES

<b>Table</b>	<b>Title</b>	<b>Page</b>
3.1	Important Parameters of Model	10
3.2	Mesh Parameters of Single Blade Model	16
3.3	Mesh Parameters of Suction control Model	22
3.4	Mesh Parameters of Blowing control Model	25
3.5	Mesh Parameters of suction and Blowing control Model	27



# TABLE OF CONTENTS

	<b>Page No.</b>
<i>Acknowledgement</i>	<i>i</i>
<i>Abstract</i>	<i>ii</i>
<i>List of Tables</i>	<i>iv</i>
<i>List of Figures</i>	<i>v</i>

## **Chapter 1: INTRODUCTION**

1.1	NASA Rotor 67	1
1.2	Boundary Layer	2
1.3	Losses	3
1.3.1	Shock Losses	4
1.3.2	Blade Loading Loss and Leakage Loss	4
1.3.3	Clearance Loss	4
1.3.4	Secondary Losses	5
1.3.5	Annulus Losses	5
1.3.6	Profile Losses	5

## **Chapter 2: LITERATURE REVIEW**

2.1	Boundary Layer Control Methods	7
2.1.1	Boundary Layer Control by Suction Method	8
2.1.2	Boundary Layer Control by Blowing Method	8

**Chapter 3: METHODOLOGY**

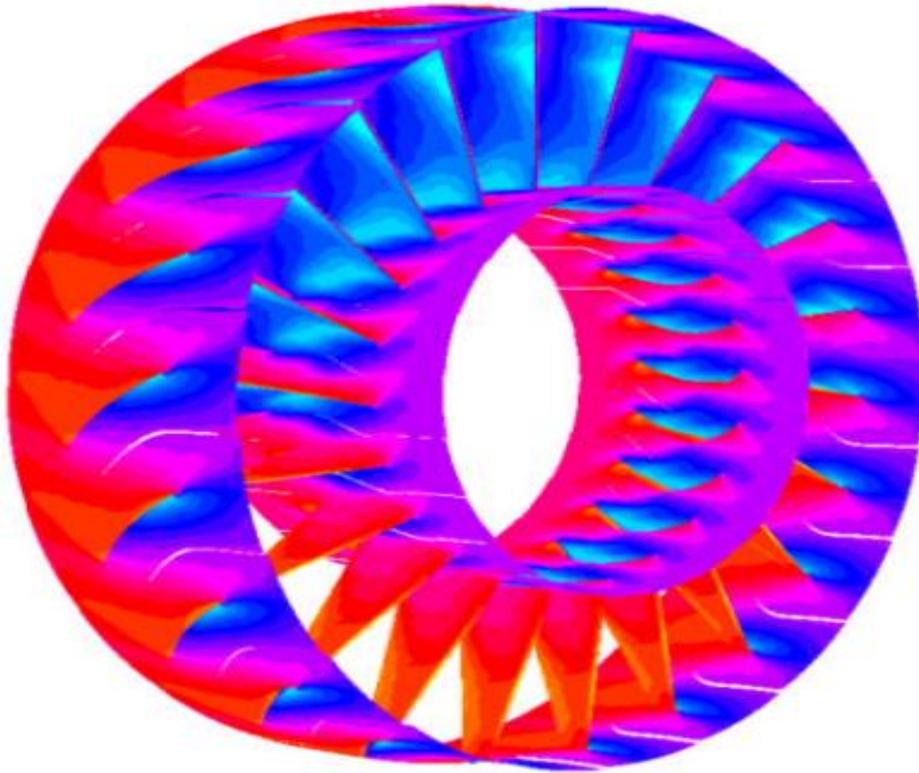
3.1	Modeling of NASA Rotor 67	9
3.2	Meshing of Model	10
3.3	Introduction to CFD	16
3.4	Computational Fluid Dynamics Setup	17
3.4.1	CFX-Pre	17
3.4.2	CFX Solver Manager	20
3.4.3	CFX Post	20
3.5	Boundary Layer Control	21
3.5.1	Suction Boundary Layer Control	22
3.5.1.1	Meshing	22
3.5.1.2	CFX Pre-Setting	23
3.5.2	Blowing Boundary Layer Control	24
3.5.2.1	Meshing	24
3.5.2.2	CFX Pre-Setting	25
3.5.3	Suction and Blowing Boundary Layer Control	26
3.5.3.1	Meshing	26
3.5.3.2	CFX Pre-Setting	27
	<b>Chapter 4: RESULTS AND DISCUSSIONS</b>	30
	<b>Chapter 5: CONCLUSIONS</b>	39
	<b>Chapter 6: REFERENCES</b>	40

## **CHAPTER 1 INTRODUCTION**

The performance of aircraft gas turbine engines in transonic flow depends majorly on improved efficiency and reduced engine weight. Efficiency can be increased with high pressure ratio compressor having multiple stages. When these type of compressor is used then this leads to increment in weight and is highly expensive also. A lot of research were done in this field and results of all these research led to the possibility of achieving high pressure ratio with least number of stages. If any of the quantity among pressure ratio and efficiency is increased then figure of merit of compressor blade increases which leads to increment of overall engine performance. In case of transonic flow which is partly high subsonic and supersonic performance of compressor blade is very important as in this type of flow there is occurrence of shock wave which leads to flow separation due to which losses like shock loss, tip loss etc. occur which affect the performance of compressor and therefore overall engine performance also reduces. To avoid these kind of losses or to prevent separation different methods are used today and in our project we are going to take NASA rotor 67 as a reference rotor blade and first going to analyze the flow types on this rotor and then do some modification in the design of this reference rotor and then analyze the result and after that performance of rotor under both condition is going to be compared. The method used for flow modification or avoid flow separation is called Boundary Layer control methods. A lot of different methods like actuating device, modification in geometry of axial fans etc. are also used to increase efficiency or performance of engines.

### ***1.1 NASA Rotor 67***

The NASA rotor 67 is an isolated axial transonic flow compressor which is generally used in transonic flow. Various experiments are done on this rotor in Glenn Research centre of NASA. This rotor has twenty two blades and the overall performance of this rotor was determined at the equivalent mass flow rates from the chock mass flow rates. Some of the parameters of NASA rotor 67 are listed in table given on the next page and all this parameters are taken as reference parameters for the design of rotor.



**Figure 1.1 NASA Rotor 67**

THE NATION BUILDERS UNIVERSITY

### ***1.2 Boundary Layer***

Boundary layer is the layer where viscosity effect is significant and flow is retarded due to this effect. This play very important role in deciding performance parameters because flow separation depends on this and entropy layer, viscous layer, pressure and temperature depend on this separation. Early separation leads to reduction of performance of engines therefore to prevent separation boundary layer needs to be controlled and this give rise to development of boundary layer control methods which became very important for transonic flow.

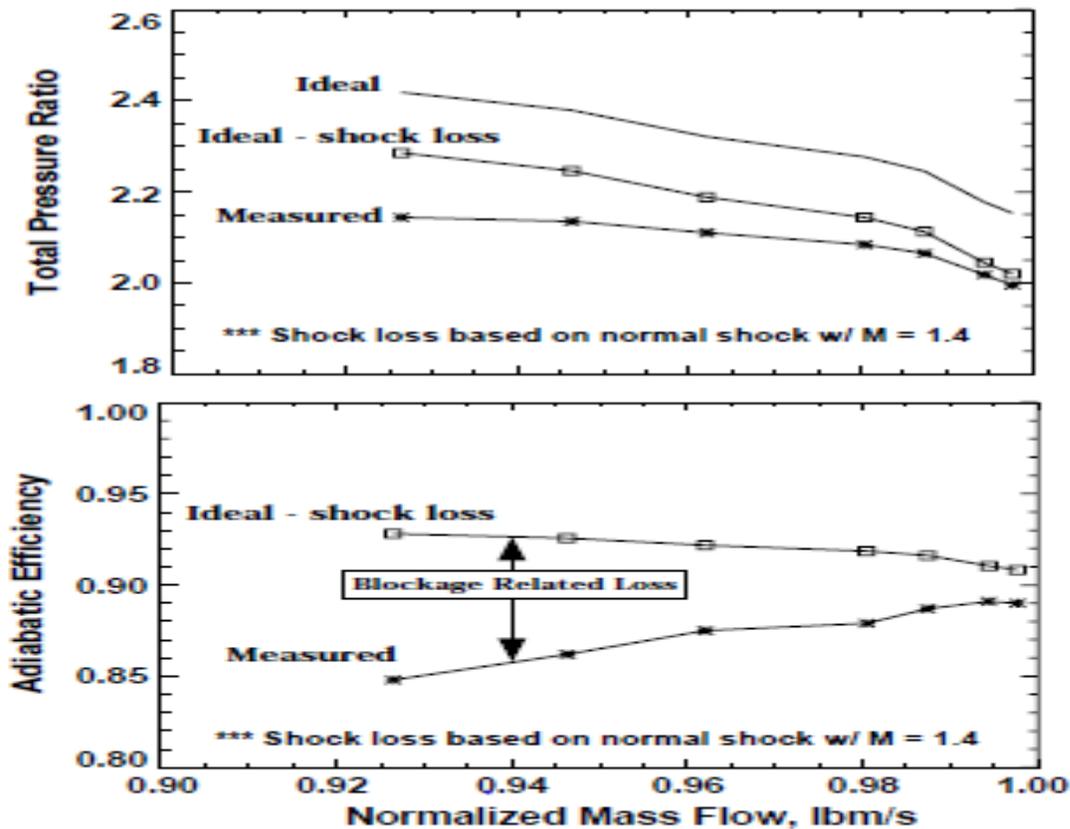


Figure 1.2 Estimation of losses

### 1.3 Losses

Loss mechanism in this type of turbo machinery is divided into two groups one is internal loss mechanism and other one is parasitic loss mechanism.

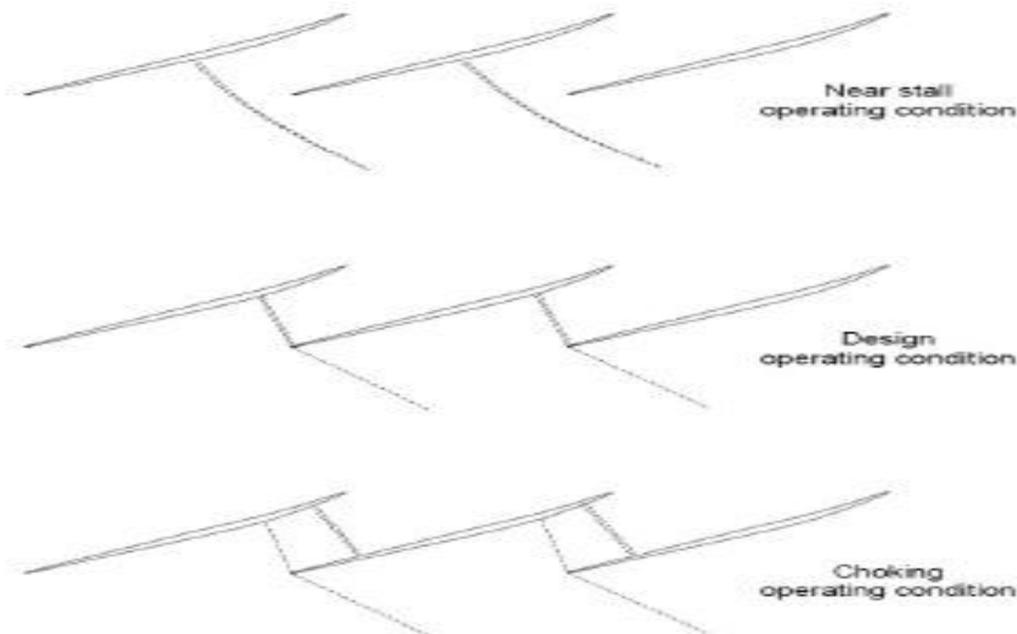
Internal loss mechanism is associated with the main flow through the compressor therefore this kind of losses affect pressure ratio and efficiency of the compressor. This includes losses at the entry of the impeller which can be grouped into three types i.e. blade loading loss, skin friction loss and clearance loss. While parasitic loss mechanism are associated with the minor flow loss leaking away from the major flow of the compressor. By the parasitic loss efficiency is reduced without any change in pressure ratio as these kind of loss mechanism will affect impeller discharge.

### 1.3.1 Shock losses

Transonic flow is combination of high subsonic and low supersonic flow and as the flow approaches toward the supersonic speeds a layer of shock has formed at the trailing edge of the blade which leads to the early separation of flow and due to this early separation efficiency as well as pressure ratio decreased by some factor and overall performance of engine reduces.

### 1.3.2 Blade loading loss and leakage loss

This type of loss is described as momentum loss due to build up of boundary layer on blade surface. This type of loss is function of diffusion factor therefore by controlling this factor this loss can be controlled. A gap is maintained in between casing and tip and fluid having less energy those fluids which undergoes through sudden contraction followed by sudden expansion there has chances of leakage of some fluid which cause leakage losses.



**Figure 1.3 Shock wave configuration inside transonic compressor rotor**

### 1.3.3 Clearance loss

For allowing free rotation a finite clearance gap is provided between the casing and the impeller and fluid will leak through this gap i.e. between the blade tip and casing from the higher pressure

region of the blade to the lower pressure region of the blade which is on the suction side of the blade. The mixing of the two flow results in clearance loss in the form of small vortex in the suction side of the blade. This kind of flow undergoes sudden contraction followed by sudden expansion across the clearance gap. This kind of loss occurs due to clearance between moving blade and casing and due to pressure difference which involves static pressure difference between suction side and pressure side air leakage takes place in compressor.

#### ***1.3.4 Secondary losses***

In the region of flow near end walls of blades having presence of different kind of unwanted circularly flows or gross flows secondary losses take place and cause of this kind of unwanted flows is pressure differentials and these losses depends upon hub to tip ratio.

#### ***1.3.5 Annulus losses***

These losses consists of loss caused by boundary layer separation on walls and boundary layer is subjected to centrifugal forces due to floor (hub) of stationary blade and this results in scrapping of outer casing due to rotation of blades.

#### ***1.3.6 Profile losses***

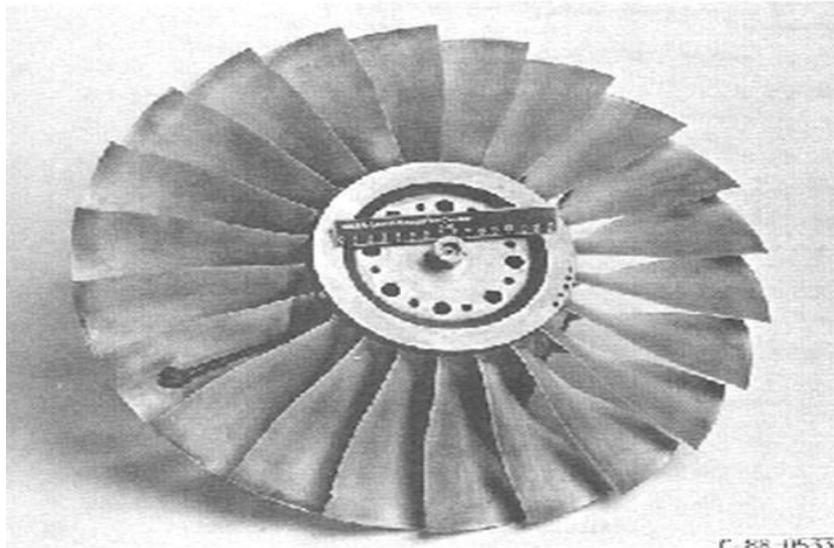
In these losses we are concerned about the growth of boundary layer on the profile of blade. When adverse pressure gradient becomes too steep then separation of boundary layer takes place and in general manner surface of blade which is more prone to this kind of separation is suction surface of blade. This separation depends on turbulence of flow , Reynolds number of flow and the incidence and we are not concerned about more supersonic flows and if flow is supersonic then extra losses will take place dueto shock formation.

Some other types of losses like end wall loss, tip vortices loss etc. also play major role in the separation of flow which reduces the pressure ratio and due to which overall performance of engine also reduces therefore all the losses should be as minimum as possible and therefore many modifications are done in the rotors to reduce losses and increase performance.

## CHAPTER 2 LITERATURE REVIEW

In aircraft engines transonic axial flow compressors are used to achieve high pressure ratio per stage and due to this high efficiency is also obtained. To reduce or decrease the size of stage of compressor stage with high pressure ratios are needed and by reducing size weight of compressor is also decreases.

NASA rotor 67 is a low aspect ratio transonic axial flow fan having twenty two blades. The Rotor design pressure ratio is nearly about 1.683 and having mass flow rate of 33.25 kg/sec. Rotor 67 has rotational speed of 16043 RPM and having tip speed of 429 m/sec. The solidity of this rotor fan varies from 3.11 at the hub and 1.29 at the tip.



**Figure 2.1 NASA Rotor 67 blades**

Boundary layer control is the controlling of behavior of fluid flow and in short it can be referred as BLC. When fluid flow can be controlled by applying different kind of new modification techniques which include boundary layer suction and blowing etc. losses can be minimized and losses on blades are low then their efficiency and performance which we are going to measure in pressure ratio will be maximum.

Performance of aircraft engines depends upon pressure ratio and efficiency. any increment in either of the quantity will leads to the reduction of engine weight and reduction of number of

stages. For improving efficiency and pressure ratios a lot of methods are introduced from the previous or ongoing research. Boundary layer control is one of the method to improve performance of engines. By the application of this method boundary layer of the rotor is controlled which prevents early separation and reduces losses like shock loss, leakage loss, tip vortices loss etc.

Boundary layer control methods or techniques are used to modify the flow so that separation can be delayed or ignored and as a result of this losses are reduced and when these losses are reduced pressure ratio of compressor stage increases and this will affect the efficiency of compressor. When pressure ratio increases then efficiency of blade or compressor also increases.

### ***2.1 Boundary layer control method***

Other methods include attachment of plasma actuating device on the trailing edge of the blade and modification in fans etc. Plasma actuating device is electrical device which is fixed on the separation point ( the point where flow separates) and by electrical means some amount of energy is provided so that the flow remains attached and less losses occur.

There are two methods used to prevent separation one is by creating suction holes on the rotor blade surface so that low energy fluid can be reenergized and mixed with the high energy fluid so that separation point shifts and it reduces the no slip condition which means velocity is zero by using this flow is smooth which reduces the chances of flow separation.

The boundary layer separation should be prevented or delayed to get better pressure recovery takes place and to reduce form drag and this can be done by various methods. One method is by giving profile of the body streamline shape which includes elongate the rear part of blade to reduce the pressure gradient and inject fluids through porous holes i.e. blowing high energy fluid particles through holes tangentially from the location where separation occurs to prevent separation this may leads to increment of skin friction due to turbulence caused and form drag is reduced. Boundary layer is the region where mass transfer , heat transfer and momentum transfer are felt and due to these effect flow is retarded and undergoes various types of losses which reduces performance therefore boundary layer control is very important to achieve high

performance of the engines. Boundary layer control refers to the controlling of behavior of fluid flow.

### ***2.1.1 Boundary layer control by Suction Method***

In this type of boundary layer control method a suction layer or surface is created on blade surface which helps in reenergizing the flow by sucking low energy fluid over the blade surface. In this low energy fluid are eliminated by this suction surface and this reenergize the boundary layer which prevent separation and as a result by using this we get better flow and pressure ratio is higher than blades without this type of suction surfaces due to which efficiency is improved.

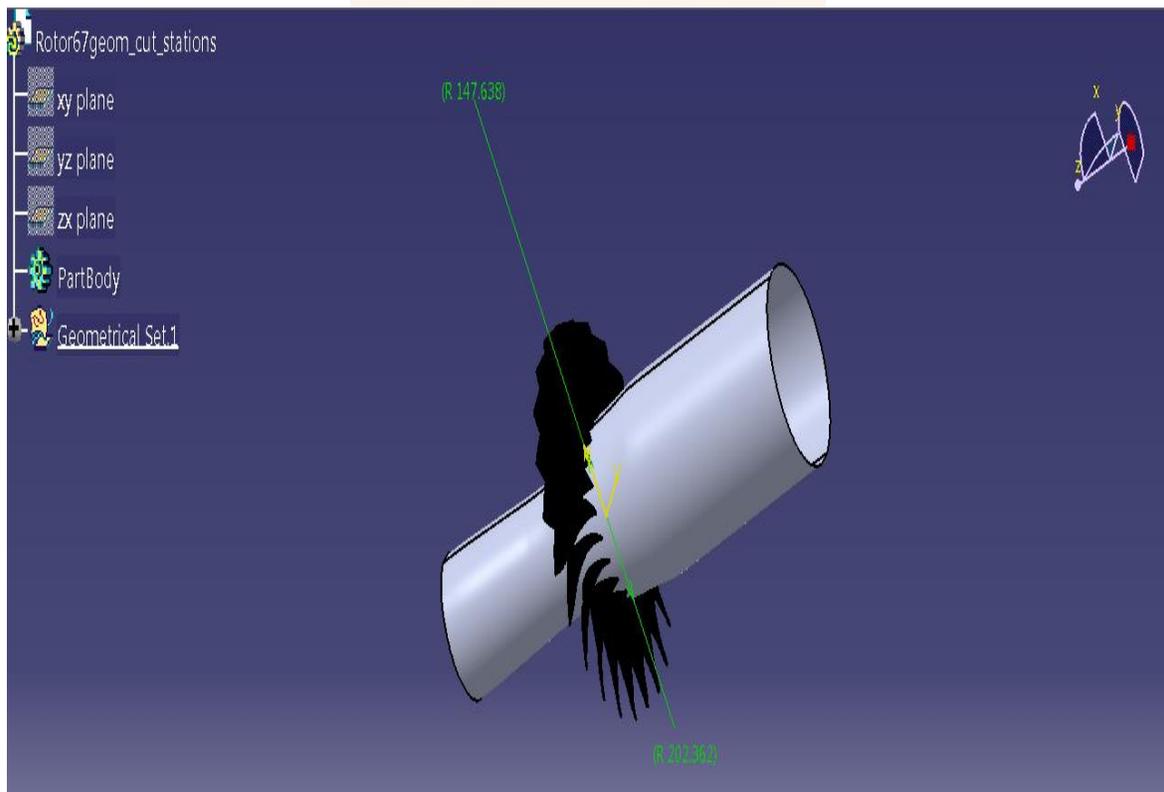
### ***2.1.2 Boundary layer control by Blowing Method***

As separation of flow occurs due to low energy of fluid over blades surface therefore by increasing the energy of this low energy fluid separation can be ignored or delayed and this is done by method called blowing. In this method high energy fluid is blowing over the low energy fluid flow surfaces and this high energy fluid get mixed with low energy fluid and as a result energy of previously fluid is increased and flow over blade surfaces is more smooth and efficiency as well as pressure ratio is improved by modifying flow over blades of surface using this method.

## CHAPTER 3 METHODOLOGY

### *3.1 Modeling of NASA Rotor 67*

Design of blades has been done in CATIA which is software used for designing any model or object. In this software first we take four points for design one surface and then we join them together so that one surface of model is complete and we are taking these points on the basis of different technical paper that we have studied before starting our modeling of blade. Design of compressor blade is not an easy job due to complexity of curve i.e. it is twisted from hub to tip, unless we donot have points we are not able to model this blade. When connecting all points i.e.is formation of surfaces done then our geometric model of blade is ready then we are going to model the casing hub and outer walls of blades using designing tools of CATIA. The Casing of the model was designed on the basis of the tip clearance as taken from the literature review.

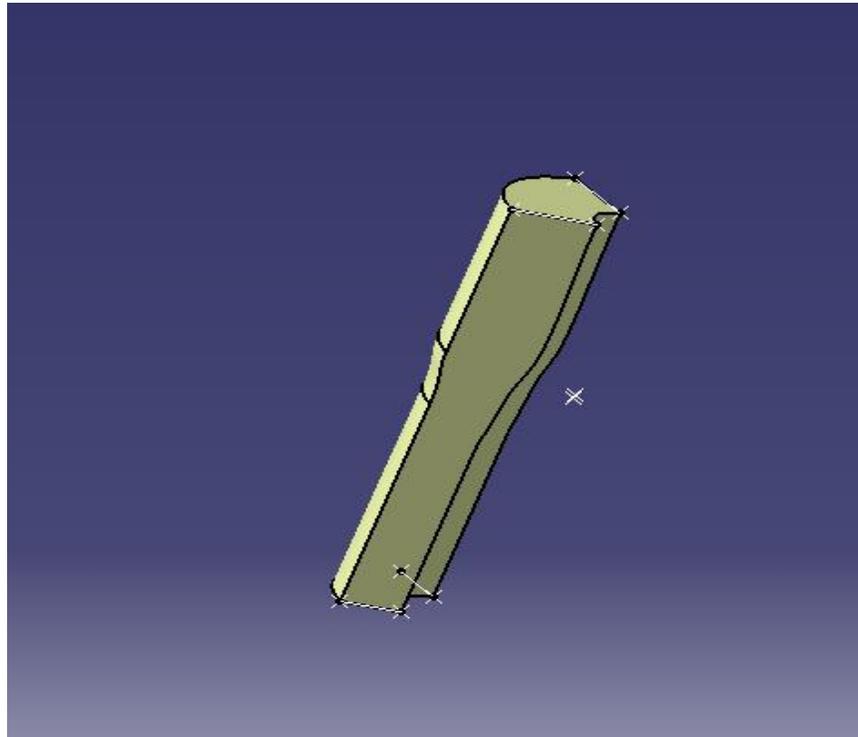


**Figure 3.1 Model of NASA Rotor 67**

Maximum Chord Length	143.68
Maximum Span	250.94
Inlet Diameter	202.362
Outlet Diameter	147.638
Tip Clearance	4.26

**Table 3.1 Important Parameters of Model**

Now the model was found out to be too large to work on the available resources and takes too much computational time. So, the model was changed so as to decrease the no. of blades and so a single blade design was constructed to work upon.

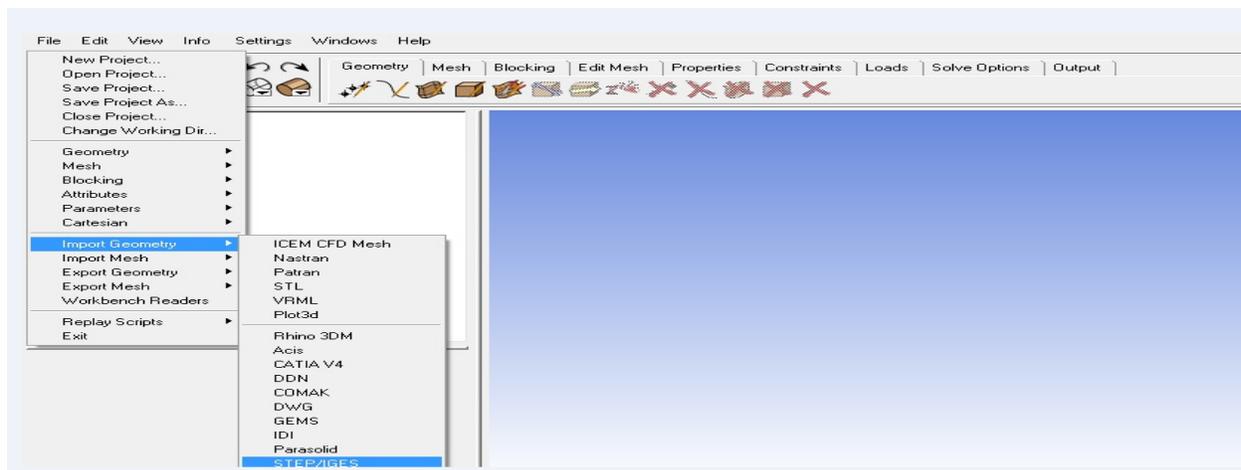


**Figure 3.2 Model of single rotor blade**

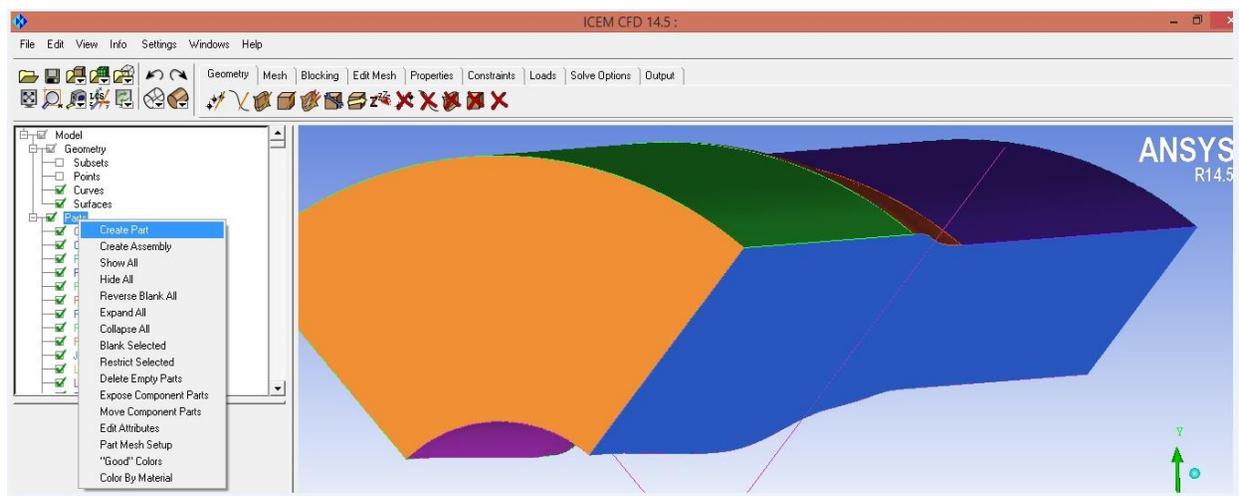
### ***3.2 Meshing of Model***

Mesh creation is very important phase in CFD simulations. If the mesh is bad then this will affect the result. In this thesis for shell meshing we have used Quad dominant mesh type and patch

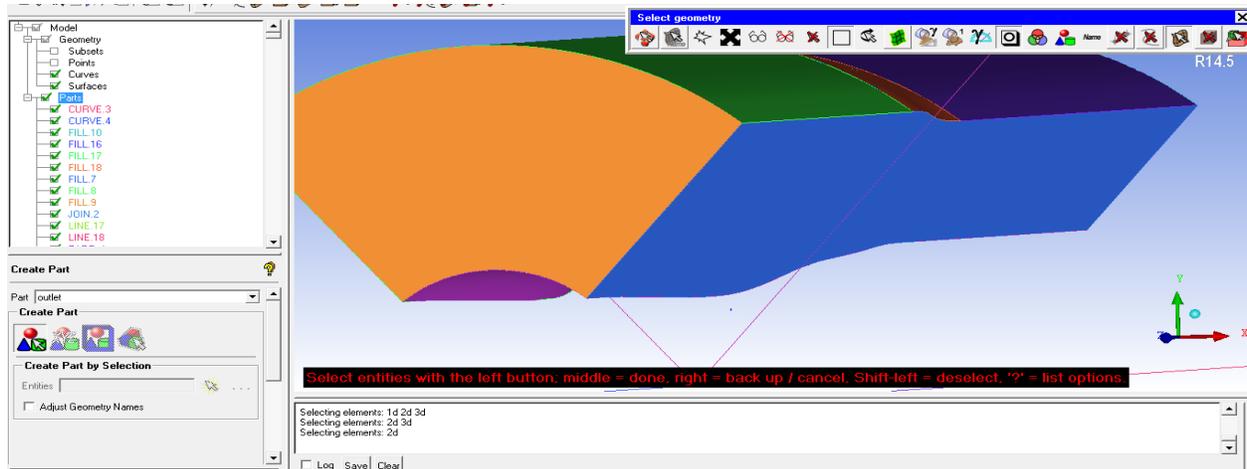
dependent method is used for obtaining this mesh and for volume mesh tetra or mixed type mesh is used and robust method helps in achieving this type of mesh. For calculating the boundary layer effects on the flow  $Y^+$  calculators are used to coarsen the mesh generation near the blade and so the height ratio was kept as 1.2 and then number of the prism layers required was calculated. After completing model design we import this model geometry in ICEM CFD 14.5. When our geometry is imported in CFD software select solid surface from the extension of surfaces then create parts by clicking right button on parts select create part and name the part and block toggle selection of curves, points and create inlet, outlet, casing, two side walls, blade and hub of geometry using following procedure



**Figure 3.3 Importing Geometry in ICEM**

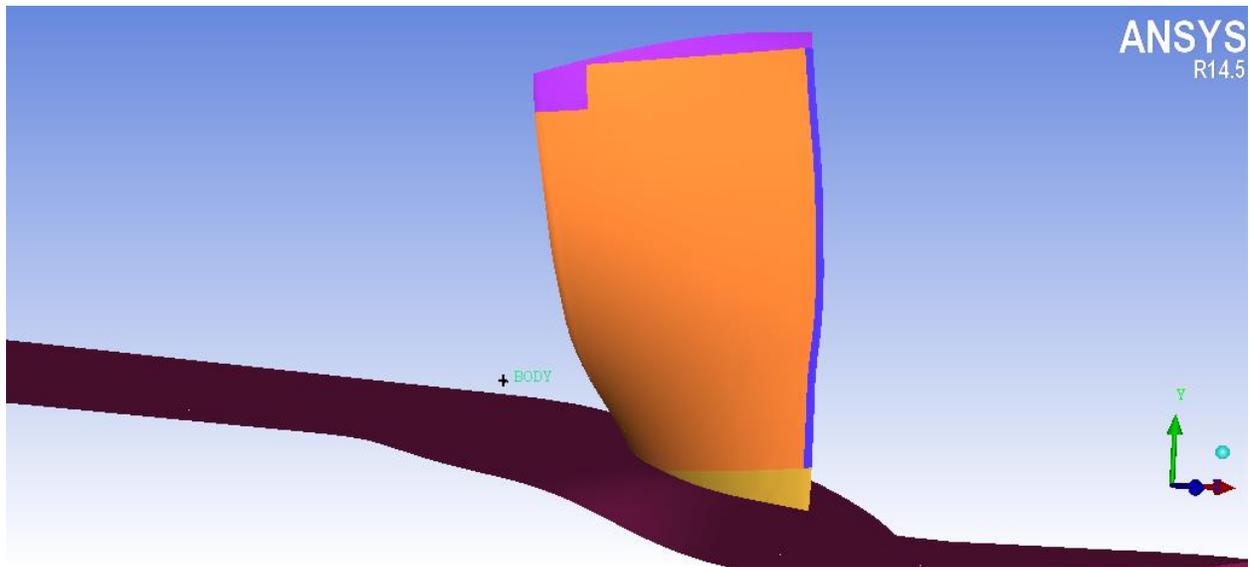


**Figure 3.4 Creating parts in ICEM**



**Figure 3.5 Steps following in creating parts of our geometry**

In our case, the finer meshes were required on the edges of the blade, so the blade was further subdivided into various parts like leading edge, lower surface, upper surface and blade.



**Figure 3.6 Model of single blade with hub**

After creating parts of the imported model or geometry then select repair geometry and in this give tolerance to 0.1 ( the tolerance of 0.1 is given according to length of the smallest component in the model) and then apply then select delete surfaces and delete overlapping surface and any hole in geometry and after repairing geometry of our model go to meshing section by selecting

mesh option from above options. Repairing of geometry is required because in importing geometry from CATIA to ICEM some surfaces are overlapped or lost or some holes are created in geometry and this will affect the meshing and if our meshing is not proper then we are not able to get result. After repair geometry create body by selecting two points diagonally.

For meshing select mesh option from above option in ICEM and in meshing first select global mesh option and in global mesh parameter give maximum element size. The global element size defines the maximum size of the element formed globally and so a global element size of 150 was provided. When global mesh size parameters are defined then go to volume meshing parameters and apply this and then go to prism parameters and give the initial height and final height which is calculated by Y+ calculator and give height ratio 1.2 because maximum height ratio should be equal to 1.2 and then compute number of prism layers and then select ok which will apply the parameters that we have applied for our meshing.

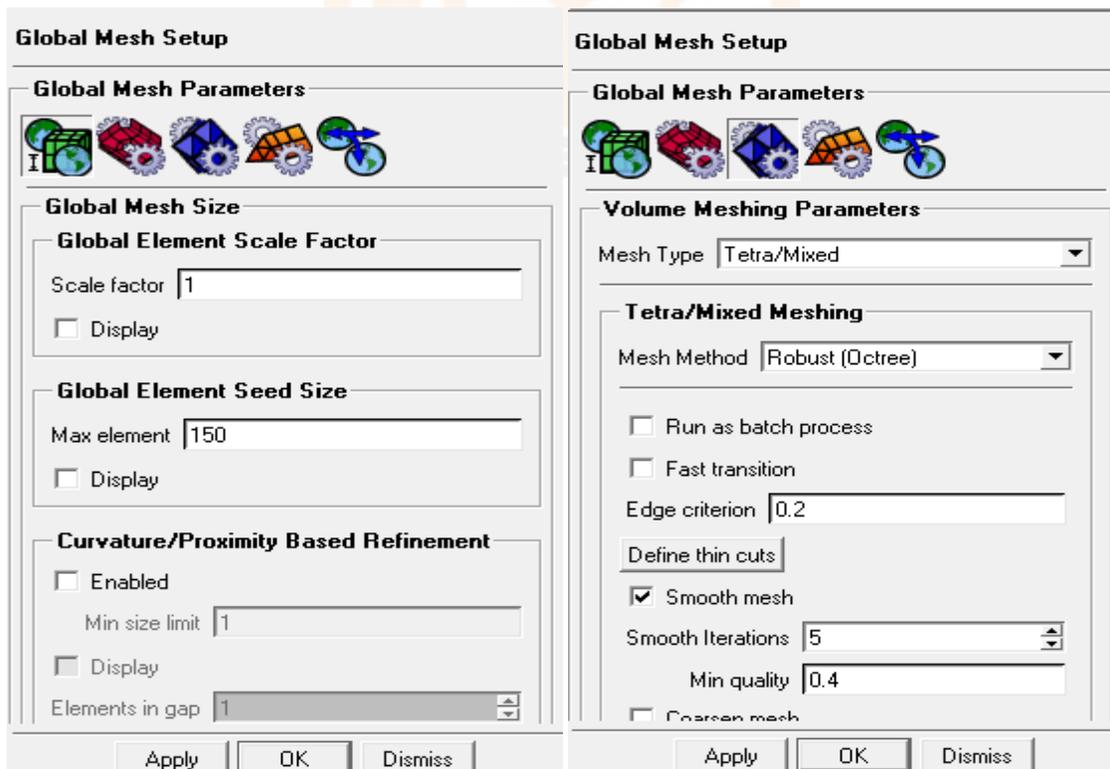


Figure 3.7 Steps to define meshing parameters



**Figure 3.8 Defining Prism Parameter**

After Completing global mesh parameters go to part mesh parameter and select and give mesh setup parameters for each part that we have created and also mention whether prism layer is available for that part or not and also mention the maximum element size for that part.

part	prism	hexa-core	max size	height	height ratio	num layers	tetra size ratio	tetra width	min size limit	max deviation	int wall	split wall
BLADE	<input checked="" type="checkbox"/>	<input type="checkbox"/>	3	0	0	0	0	0	0	0	<input type="checkbox"/>	<input type="checkbox"/>
BODY	<input type="checkbox"/>	<input type="checkbox"/>										
CASING	<input type="checkbox"/>	<input type="checkbox"/>	60	0	0	0	0	0	0	0	<input type="checkbox"/>	<input type="checkbox"/>
FILL.7	<input type="checkbox"/>	<input type="checkbox"/>	0						0	0		
FILL.8	<input type="checkbox"/>	<input type="checkbox"/>	0						0	0		
FILL.9	<input type="checkbox"/>	<input type="checkbox"/>	0						0	0		
FILL.10	<input type="checkbox"/>	<input type="checkbox"/>	0						0	0		
FILL.16	<input type="checkbox"/>	<input type="checkbox"/>	0						0	0		
FILL.17	<input type="checkbox"/>	<input type="checkbox"/>	0						0	0		
FILL.18	<input type="checkbox"/>	<input type="checkbox"/>	0						0	0		
GEOM	<input type="checkbox"/>	<input type="checkbox"/>							0	0		
HUB	<input type="checkbox"/>	<input type="checkbox"/>	60	0	0	0	0	0	0	0	<input type="checkbox"/>	<input type="checkbox"/>
INLET	<input type="checkbox"/>	<input type="checkbox"/>	60	0	0	0	0	0	0	0	<input type="checkbox"/>	<input type="checkbox"/>
JOIN.2	<input type="checkbox"/>	<input type="checkbox"/>										
LINE.17	<input type="checkbox"/>	<input type="checkbox"/>										
LINE.18	<input type="checkbox"/>	<input type="checkbox"/>										
LOWER_BLADE	<input checked="" type="checkbox"/>	<input type="checkbox"/>	1	0	0	0	0	0	0	0	<input type="checkbox"/>	<input type="checkbox"/>
OUTLET	<input type="checkbox"/>	<input type="checkbox"/>	60	0	0	0	0	0	0	0	<input type="checkbox"/>	<input type="checkbox"/>
PART_2	<input type="checkbox"/>	<input type="checkbox"/>	0						0	0		
PART_3	<input type="checkbox"/>	<input type="checkbox"/>	0						0	0		
PART_4	<input type="checkbox"/>	<input type="checkbox"/>	0						0	0		
PART_5	<input type="checkbox"/>	<input type="checkbox"/>	0						0	0		
PART_6	<input type="checkbox"/>	<input type="checkbox"/>	0						0	0		
PART_7	<input type="checkbox"/>	<input type="checkbox"/>	0						0	0		
PART_8	<input type="checkbox"/>	<input type="checkbox"/>	0						0	0		
PART_9	<input type="checkbox"/>	<input type="checkbox"/>	0						0	0		
PART_10	<input type="checkbox"/>	<input type="checkbox"/>	0						0	0		
PART_11	<input type="checkbox"/>	<input type="checkbox"/>	0						0	0		
PART_12	<input type="checkbox"/>	<input type="checkbox"/>	0						0	0		
PART_13	<input type="checkbox"/>	<input type="checkbox"/>	0						0	0		

**Figure 3.9 Defining of part mesh setup**

In our case in the part mesh setup, different element sizes were given on the various subsections of the blade so as to provide coarsen mesh on the edges of the blade.

After defining each mesh parameters then select compute from the given option of mesh and then select create prism layer and then apply compute from the given option and when we apply this compute option meshing will start.

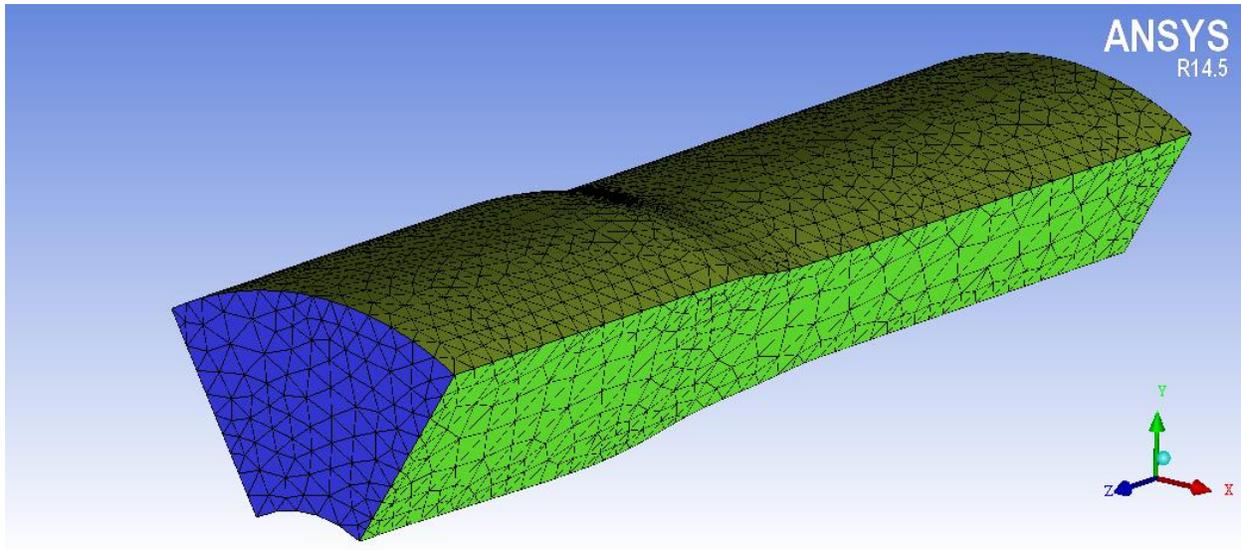


Figure 3.10 Meshing of the model

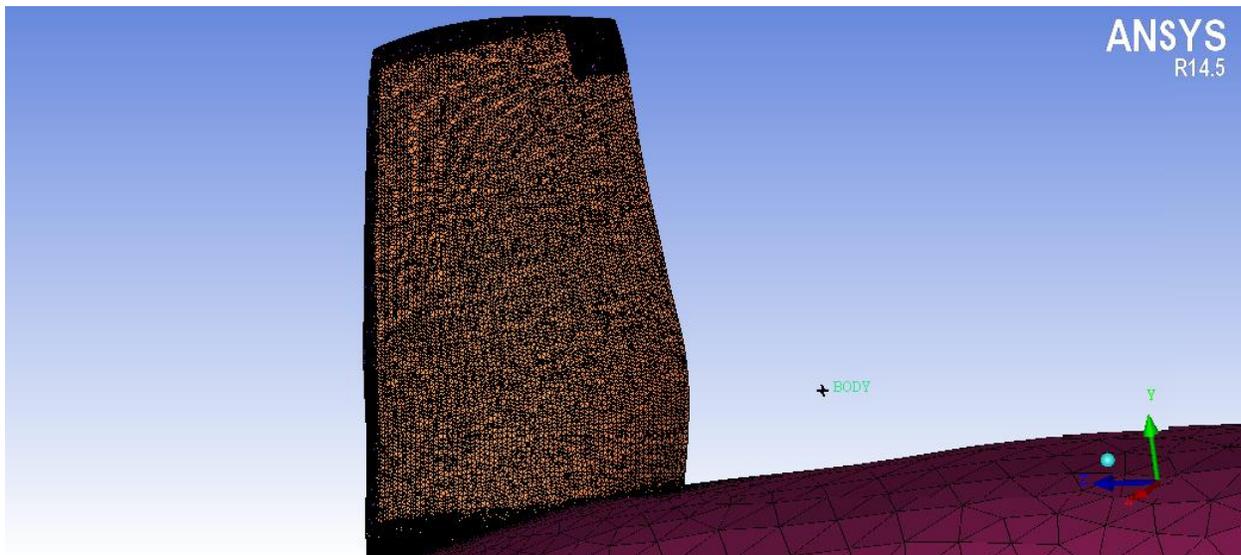


Figure 3.11 Meshing of the blade and hub

In the table below, the prism parameters as well as the final mesh elements and nodes are specified.

Initial Height	0.000169291
Total Height	0.0121949
Total Elements	1669986
Total nodes	566300

**Table 3.2 Mesh Parameters of Single Blade Model**

### ***3.3 Introduction to CFD***

CFD (Computational Fluid Dynamics) uses numerical methods and several algorithm to solve the model or problem and analyze problems associated with fluid flows. It uses finite element principle to solve problem. For doing CFD analysis different software's are used and in our thesis ANSYS 14.5 is used to do computational fluid dynamics analysis. On the basis of analysis we are deciding whether our models or designs are right or these models need some kind of modification or not and boundary condition that we applied to our models are right or not. Analysis is also done to check theories are correct or not and by providing characteristics at certain altitude it gives fluid flow behavior at that altitude around the body we design. In the modern world these software have large application. Structure analysis can also be done by this software. In our thesis we are doing unstructured analysis due to complexity of our model.

Software that we are going to use i.e. ANSYS will help us in predicting the impact of flow on our models. ANSYS CFD analysis tool include ANSYS Fluent and ANSYS CFX. Due to solver robustness and speed and advanced modeling capabilities and experience, solution provided by this software can be trusted. Fluent or CFX can be used separately or together to do the analysis of fluid flow around our models. This software also has the capabilities of unparalleled fluid flow which can be used to design, model and optimize new equipment.

### 3. 4 COMPUTATIONAL FLUID DYNAMICS (CFD) Set-up

Three dimensional simulations have been done by using simulation software ANSYS CFX and this can be divided into three parts.

1. CFX -Pre
2. CFX -Solver manager
3. CFX - Post

#### 3.4.1 CFX - Pre

For the simulation software ANSYS CFX this works as preprocessor. The mesh file or model which is produced in ICEM can be imported to CFX -Pre by first generating cfx input file of that model and then in CFX- Pre open this case file by going to open case and data in preprocessor software and then open it. After importing mesh model then create domain in this preprocessing software. The problem was defined in the turbo mode of the CFX software. Turbo mode is a special mode provided in CFX with predefined settings for the use in turbo machinery.

It can be opened in CFX- Pre by using the following:-

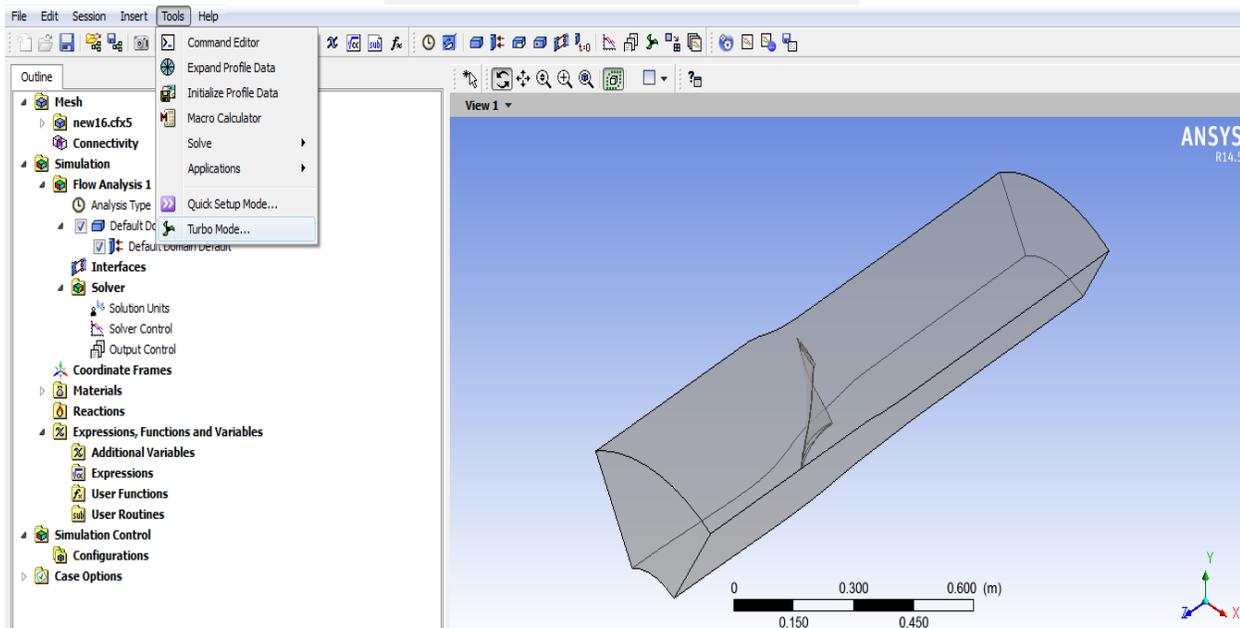


Figure 3.12 Opening method for turbo mode in CFX -Pre

The settings for CFX- Pre used are based on the units of the mesh imported.

The parameters for the mesh units in mm are:-

- Basic Settings
  - ❖ Machine Type : Axial Compressor
  - ❖ Rotation Axis : z
- Component type
  - ❖ Type : Rotating
  - ❖ Value -16043 [RPM]
  - ❖ Tip clearance at shroud : yes
- Fluid :-- Air Ideal Gas
- Analysis Type : -- Steady State
- Model data :
  - ❖ Reference Pressure :- 0 (Zero) Pa
  - ❖ Heat Transfer :- Total Energy
  - ❖ Turbulence :- Shear stress Transport
  - ❖ Wall Functions :- Automatic and compressible high speed heat transfer model
- Inflow/Outflow boundary templates: P-total inlet P-static outlet
- Inflow Boundary condition:
  - ❖ Mass and Momentum :- Static Frame Total Pressure
  - ❖ Relative Pressure ( P- Total) :- 101325 Pa
  - ❖ Flow direction :- Normal to the boundary
  - ❖ Static Frame Total temperature :- 288.2 K
- Outlet Boundary condition:
  - ❖ Mass and Momentum :- Static Pressure and mention pressure as 114500 Pa
- Solver Parameter :
  - ❖ Advection Scheme :- High Resolution
  - ❖ Time Scale Control :- Auto Timescale
  - ❖ Length Scale :- Conservative
  - ❖ Maximum Timescale :- 0.000001

- ❖ Convergence Residual Criteria:- MAX Type and Target as 0.00001

The parameters for the mesh units in cm are:-

- Basic Settings
  - ❖ Machine Type : Axial Compressor
  - ❖ Rotation Axis : z
- Component type
  - ❖ Type : Rotating
  - ❖ Value -3600 [RPM]
  - ❖ Tip clearance at shroud : yes
- Fluid :-- Air Ideal Gas
- Analysis Type : -- Steady State
- Model data :
  - ❖ Reference Pressure :- 0 (Zero) Pa
  - ❖ Heat Transfer :- Total Energy
  - ❖ Turbulence :- Shear stress Transport
  - ❖ Wall Functions :- Automatic and compressible high speed heat transfer model
- Inflow/Outflow boundary templates: P-total inlet P-static outlet
- Inflow Boundary condition:
  - ❖ Mass and Momentum :- Static Frame Total Pressure
  - ❖ Relative Pressure ( P- Total) :- 101325 Pa
  - ❖ Flow direction :- Normal to the boundary
  - ❖ Static Frame Total temperature :- 288.2 K
- Outlet Boundary condition:
  - ❖ Mass and Momentum :- Static Pressure and mention pressure as 114500 Pa
- Solver Parameter :
  - ❖ Advection Scheme :- High Resolution
  - ❖ Time Scale Control :- Auto Timescale
  - ❖ Length Scale :- Conservative
  - ❖ Maximum Timescale :- 0.000001

❖ Convergence Residual Criteria:- MAX Type and Target as 0.00001

In CFX -Pre physics of problems or models are defined in this fluid models type, inlet, outlet etc are defined. Whether velocity inlet or outlet boundary condition is taken or pressure boundary condition is given even in pressure boundary condition either pressure is static pressure or total pressure , any heat transfer takes place , fluid is ideal or not and model condition is steady or transient etc. all these parameters are defined here therefore this is very important phase and all the parameters must be defined correctly otherwise this may affect the result even result may not come due to error which is produced by wrong parameters.

### **3.4.2 CFX Solver Manager**

The input of CFX solver is the defined file generated in the CFX pre. This part of software uses graphical interface method for CFD calculations. CFX solver manager also allows doing numerical precision and in our calculation double precision method is chosen which gives more accurate numerical solution. By using this double precision method computer memory and grid size is also doubled when this is going to be executed. For running file input file selected from the solver input file option and then defined the place where output file has to be saved by selecting option working directory and then start run and after running this graphical interface is opened on left side and number of iteration and some flow parameters are shown on right side of this graphical interface page.

### **3.4.3 CFX -Post**

The output of the simulation or the result file is then loaded in this where the flow can be analyzed. when the results are initialized thermodynamic properties can also be extracted from optimization process therefore expression of isentropic efficiency and pressure ratio are also created in expression tab of CFD-Post.

Isentropic Efficiency :-

$$\eta = \frac{PR^{\frac{\gamma-1}{\gamma}} - 1}{T_2 - T_1}$$

In the expression of isentropic efficiency of rotor PR is the pressure ratio and  $\gamma$  is the specific heat ratio for air which has the numerical value of 1.4.  $T_2$  and  $T_1$  are the total temperatures of outlet and inlet respectively.

### 3.5 Boundary Layer Control

The Boundary Layer Control Methods used were based on the basis of the post processing of the model without boundary layer control methods. And so appropriate boundary layer controls to techniques were used to allow for decrement in the boundary layer separations. Now the usage and location of these boundary layer control techniques was based on the literature review i.e.

Authors	Type of flow control and devices	Type of study	Application	Flow control device position	M (%) of inlet flow)
Loughary et al. [1]	Aspiration and blowing by slots	Experiments	Stator cascade	25-60%c	0.5-1.5%
Kerrebrock et al. [2]	Aspiration by scoops	CFD and experiments	Rotor cascade and compressor stage	Rotor:40-55%c, Stator: 50-85%c	Rotor:3.4%, stator:7.2%
Kerrebrock et al. [3]	Aspiration by slots	CFD and experiments	Fan stage	40%c	Rotor:3%, stator:3%
Marchant[4]	Aspiration by slots	CFD	Compressor stage	Rotor:40%c, 40-100% span, Stator: 40%c, 0-100% H	Rotor:1.5%, stator: 2%
Marchant et al. [5]	Aspiration by slots	CFD	Compressor stage	Rotor: slot (40-100%cH), Casing: slot Stator: slots (0-100% H, 25-75%c), circumferential slot (hub)	Rotor: 3% Stator: 4%
Schuler et al. [6,7]	Aspiration by slots and holes	CFD and experiments	Fan stage	NA (Not available)	Rotor: 2.25% Stator: 2.45%
Dang et al. [8]	Aspiration by slot	CFD	Rotor cascade	NA (Not available)	Rotor:2% (baseline), 0.3% (optimised configuration) 2%
Hüblich et al. [9]	Aspiration by slot	CFD and experiments	Rotor cascade	44% , 53% c	
Chan et al. [13]	Aspiration by slot	CFD	Stator cascade	40-80%c (parametric studies), slot span: 0-100% H	0.5%-1.5%
Günzler et al. [12]	Aspiration by slot and holes	CFD	Compressor stage	Rotor: casing slot, stator: casing hole	Rotor:1%, stator:3%
Godard et al. [27]	Aspiration by slot	CFD	Diffuser	Diffuser slot at the separation (after selection)	3.2%
Chakraborty et al. [22]	Aspiration by holes	Experiments	Stator cascade	Holes at 23- 59%c	0.15%
Ghobedo et al. [25]	Aspiration by slots	CFD and experiments	Stator cascade	Slots on blade and endwall	0.7%
Hiergt et al. [23]	Passive control by vortex generators	CFD and Experiments	Stator cascade	NA	NA
Kirtley et al. [10]	Blowing by holes	CFD and experiments	Stator cascade	Holes on the blade suction side aft of the computed separation line	0% to 1.5%
Song et al. [30]	Blowing by holes (passive)	CFD	Stator cascade	Holes on blade suction side at 70-90%c and 10-15%span	1.35-1.49%

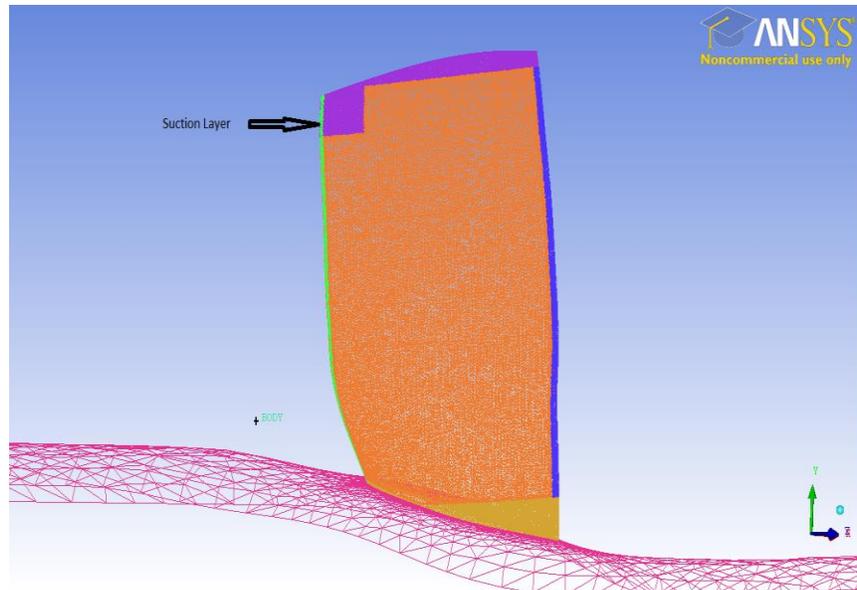
Figure 3.13 Classification of bibliographic literature

### 3.5 Suction Boundary Layer Control

The Suction Boundary Layer Control Method is used to remove the low energy fluid from the boundary layer so as to prevent boundary layer separation. So according to the post processing from the simpler model without the boundary layer control, a suction slot was applied on the trailing edge section of the suction surface.

#### 3.5.1 Meshing

For meshing, the suction layer was named differently in the part name as suction and the required meshing parameters were applied for doing meshing in ICEM on the suction model.



**Figure 3.14 Meshing of Suction model**

In the table below, the prism parameters as well as the final mesh elements and nodes are specified.

Initial Height	0.000169291
Total Height	0.0121949
Total Elements	1736711
Total nodes	591559

**Table 3.3 Mesh Parameters of Suction control Model**

### 3.5.1 CFX - Pre

In CFX – Pre the boundary conditions of the analysis are put up. As above, Turbo mode was used for providing some pre- specified conditions. The suction boundary layer was applied as outlet and the two conditions were put up on it according to the two mass flow rates selected. The mass flow rates for the suction boundary layer were put up according to the literature review which states that the aspiration of 2% mass flow rate is most effective in controlling the boundary layer separation. So mass flow rates of 1Kg/s and 1.5Kg/s were used.

The parameters for the problem applied are:-

- Basic Settings
  - ❖ Machine Type : Axial Compressor
  - ❖ Rotation Axis : z
- Component type
  - ❖ Type : Rotating
  - ❖ Value -16043 [RPM]
  - ❖ Tip clearance at shroud : yes
- Fluid :-- Air Ideal Gas
- Analysis Type : -- Steady State
- Model data :
  - ❖ Reference Pressure :- 0 (Zero) Pa
  - ❖ Heat Transfer :- Total Energy
  - ❖ Turbulence :- Shear stress Transport
  - ❖ Wall Functions :- Automatic and compressible high speed heat transfer model
- Inflow/Outflow boundary templates: P-total inlet P-static outlet
- Inflow Boundary condition:
  - ❖ Mass and Momentum :- Static Frame Total Pressure
  - ❖ Relative Pressure ( P- Total) :- 101325 Pa
  - ❖ Flow direction :- Normal to the boundary
  - ❖ Static Frame Total temperature :- 288.2 K
- Outlet Boundary condition:

- ❖ Mass and Momentum :- Static Pressure and mention pressure as 114500 Pa
- Suction:
  - ❖ Mass and Momentum :- Outlet and mass flow rate of 1Kg/s or 1.5 Kg/s
- Solver Parameter :
  - ❖ Advection Scheme :- High Resolution
  - ❖ Time Scale Control :- Auto Timescale
  - ❖ Length Scale :- Conservative
  - ❖ Maximum Timescale :- 0.000001
  - ❖ Convergence Residual Criteria:- MAX Type and Target as 0.00001

### 3.7 Blowing Boundary Layer Control

The Boundary Boundary Layer Control Method is used to reenergize the low energy fluid from the boundary layer so as to prevent boundary layer separation. So according to the post processing from the simpler model without the boundary layer control, a blowing slot was applied on the 55% chord of the suction surface.

#### 3.7.1 Meshing

For meshing, the blowing slot was named differently in the part name as blowing and the required meshing parameters were applied for doing meshing in ICEM on the suction model.

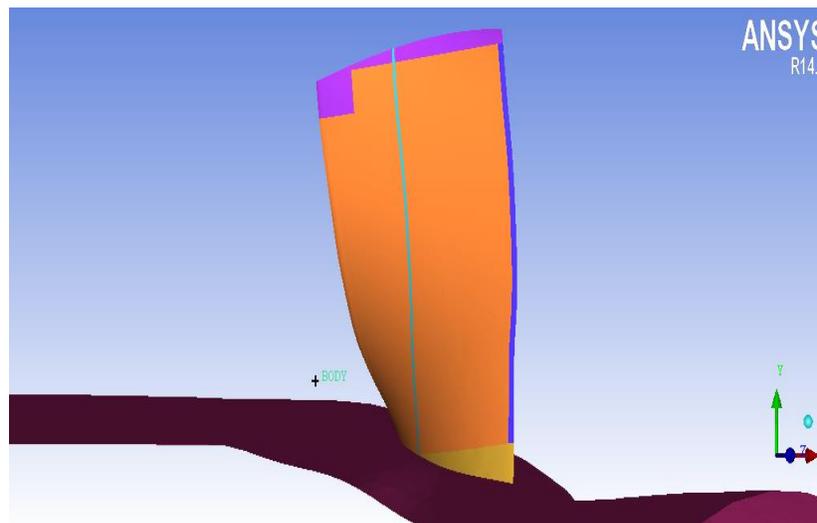


Figure 3.15 Meshing of Suction model

In the table below, the prism parameters as well as the final mesh elements and nodes are specified.

Initial Height	0.000169291
Total Height	0.0121949
Total Elements	1716462
Total nodes	585573

**Table 3.4 Mesh Parameters of Blowing control Model**

### 3.7.2 CFX - Pre

In CFX – Pre the boundary conditions of the analysis are put up. As above, Turbo mode was used for providing some pre- specified conditions. The boundary layer was applied as outlet and the two conditions were put up on it according to the two mass flow rates selected. The mass flow rates for the suction boundary layer were put up according to the literature review which states that the aspiration of 2% mass flow rate is most effective in controlling the boundary layer separation. So mass flow rates of 1Kg/s and 1.5Kg/s were used.

The parameters for the problem applied are:-

- Basic Settings
  - ❖ Machine Type : Axial Compressor
  - ❖ Rotation Axis : z
- Component type
  - ❖ Type : Rotating
  - ❖ Value -16043 [RPM]
  - ❖ Tip clearance at shroud : yes
- Fluid :-- Air Ideal Gas
- Analysis Type : -- Steady State
- Model data :
  - ❖ Reference Pressure :- 0 (Zero) Pa
  - ❖ Heat Transfer :- Total Energy
  - ❖ Turbulence :- Shear stress Transport
  - ❖ Wall Functions :- Automatic and compressible high speed heat transfer model

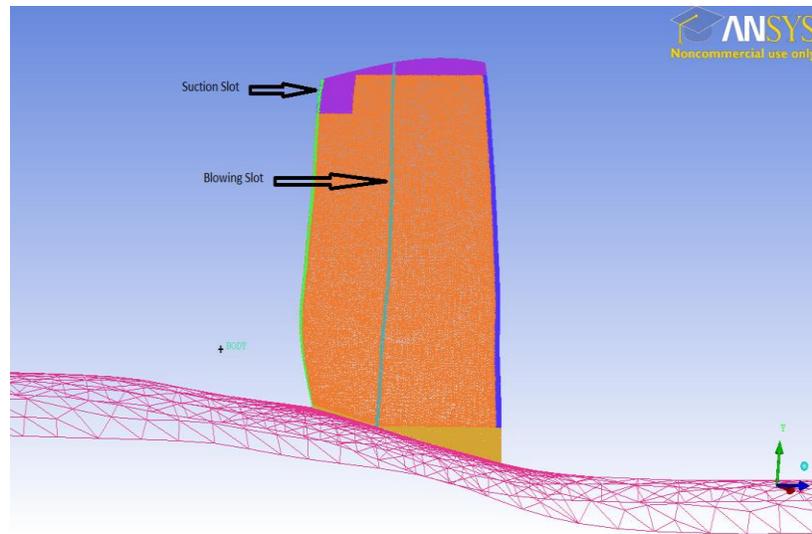
- Inflow/Outflow boundary templates: P-total inlet P-static outlet
- Inflow Boundary condition:
  - ❖ Mass and Momentum :- Static Frame Total Pressure
  - ❖ Relative Pressure ( P- Total) :- 101325 Pa
  - ❖ Flow direction :- Normal to the boundary
  - ❖ Static Frame Total temperature :- 288.2 K
- Outlet Boundary condition:
  - ❖ Mass and Momentum :- Static Pressure and mention pressure as 114500 Pa
- Blowing:
  - ❖ Mass and Momentum :- Inlet and mass flow rate of 1Kg/s or 1.5 Kg/s
- Solver Parameter :
  - ❖ Advection Scheme :- High Resolution
  - ❖ Time Scale Control :- Auto Timescale
  - ❖ Length Scale :- Conservative
  - ❖ Maximum Timescale :- 0.000001
  - ❖ Convergence Residual Criteria:- MAX Type and Target as 0.00001

### ***3.7 Suction and Blowing Boundary Layer Control***

The Suction and Blowing Boundary Layer Control Methods are used to reenergize the low energy fluid from the boundary layer and to suck away low energy fluid simultaneously so as to prevent boundary layer separation. So according to the post processing from the simpler model without the boundary layer control, a blowing slot was applied on the 55% chord of the suction surface and a suction slot was applied at the trailing edge of the suction surface.

#### ***3.7.1 Meshing***

For meshing, the blowing slot was named differently in the part name as blowing and the required meshing parameters were applied for doing meshing in ICEM on the suction model.



**Figure 3.16 Meshing of Blowing and Suction model**

In the table below, the prism parameters as well as the final mesh elements and nodes are specified.

Initial Height	0.000169291
Total Height	0.0121949
Total Elements	1786968
Total nodes	610165

**Table 3.5 Mesh Parameters of suction and Blowing control Model**

### 3.7.2 CFX - Pre

In CFX – Pre the boundary conditions of the analysis are put up. As above, Turbo mode was used for providing some pre- specified conditions. The boundary condition of suction slot was applied as outlet and The boundary condition of blowing slot was applied as inlet. The mass flow rates for the suction boundary layer were put up according to the literature review which states that the aspiration of 2% mass flow rate is most effective in controlling the boundary layer separation.

The parameters for the problem applied are:-

- Basic Settings
  - ❖ Machine Type : Axial Compressor
  - ❖ Rotation Axis : z
  
- Component type
  - ❖ Type : Rotating
  - ❖ Value -16043 [RPM]
  - ❖ Tip clearance at shroud : yes
  
- Fluid :-- Air Ideal Gas
- Analysis Type : -- Steady State
- Model data :
  - ❖ Reference Pressure :- 0 (Zero) Pa
  - ❖ Heat Transfer :- Total Energy
  - ❖ Turbulence :- Shear stress Transport
  - ❖ Wall Functions :- Automatic and compressible high speed heat transfer model
  
- Inflow/Outflow boundary templates: P-total inlet P-static outlet
- Inflow Boundary condition:
  - ❖ Mass and Momentum :- Static Frame Total Pressure
  - ❖ Relative Pressure ( P- Total) :- 101325 Pa
  - ❖ Flow direction :- Normal to the boundary
  - ❖ Static Frame Total temperature :- 288.2 K
  
- Outlet Boundary condition:
  - ❖ Mass and Momentum :- Static Pressure and mention pressure as 114500 Pa

➤ Suction:

- ❖ Mass and Momentum :- Outlet and mass flow rate of 1 Kg/s

➤ Blowing:

- ❖ Mass and Momentum :- Inlet and mass flow rate of 1.5 Kg/s

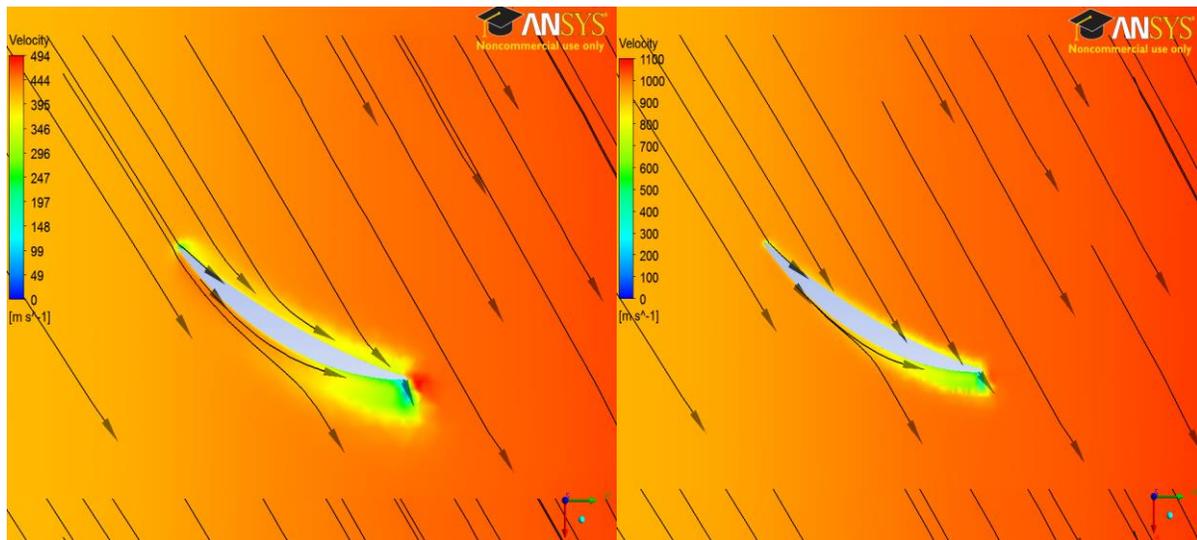
➤ Solver Parameter :

- ❖ Advection Scheme :- High Resolution
- ❖ Time Scale Control :- Auto Timescale
- ❖ Length Scale :- Conservative
- ❖ Maximum Timescale :- 0.000001
- ❖ Convergence Residual Criteria:- MAX Type and Target as 0.00001



## CHAPTER 4 RESULTS & DISCUSSIONS

The CFD analysis of the above defined models was post processed in CFX post processor. The post processing was also helped with the turbo report generated by CFX in post processing. Firstly for the results with no boundary layer control techniques was looked into so as to make way for the application, modeling & meshing of the models with boundary layer control techniques. So firstly in post processing the velocity vectors around the blade were looked into. This was checked not at various sections of span but also at two models with changed length units and rpm, as discussed earlier.

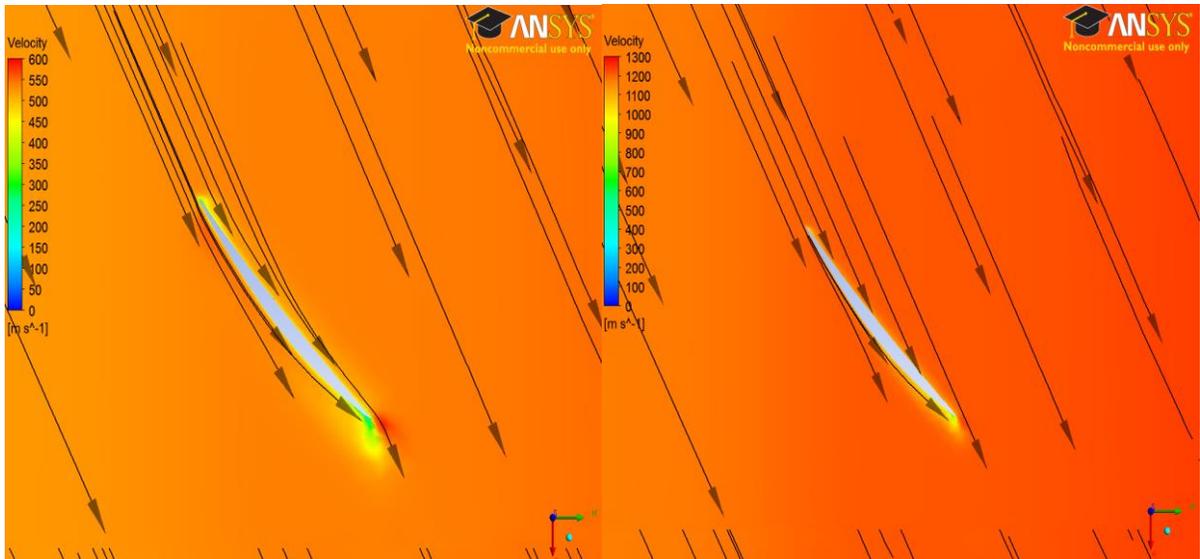


(Length units in mm)

(Length units in cm)

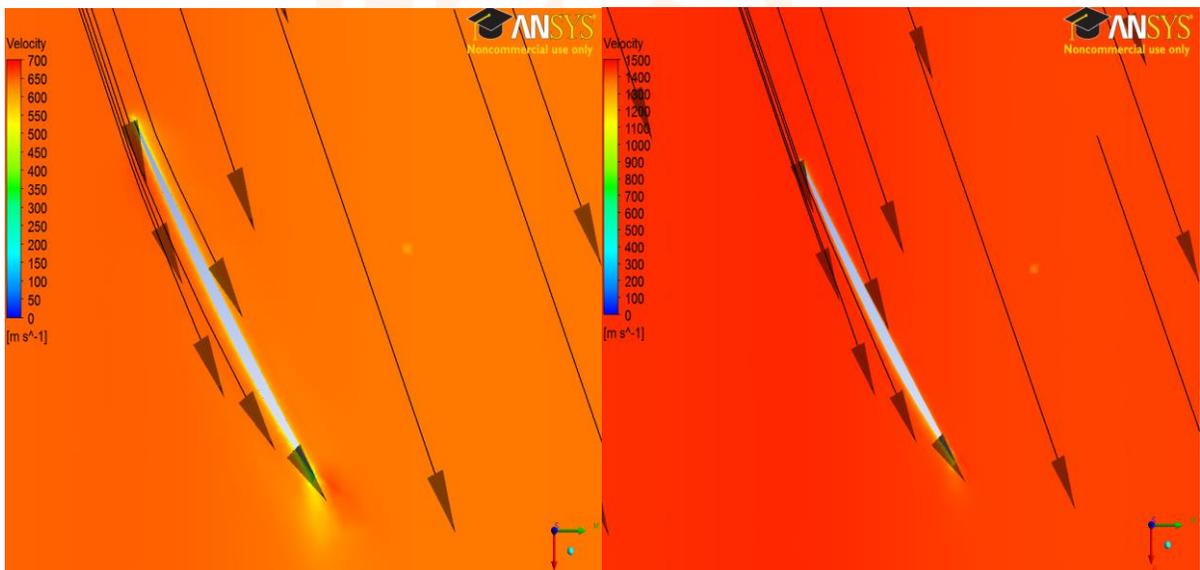
**Figure 4.1 Velocity vectors at 20% span**

The above figure shows how the boundary layer separation is occurring on the suction surface of the blade. Both the images show the similarity of having the boundary layer separation on the suction surface at around 60% chord. Close looking at the images show the boundary layer separation is occurring with velocity vectors showing the direction as well as the change in color from orange to green showing the vortex formation there.



(Length units in mm)      (Length units in cm)

**Figure 4.2 Velocity vectors at 50% span**



(Length units in mm)

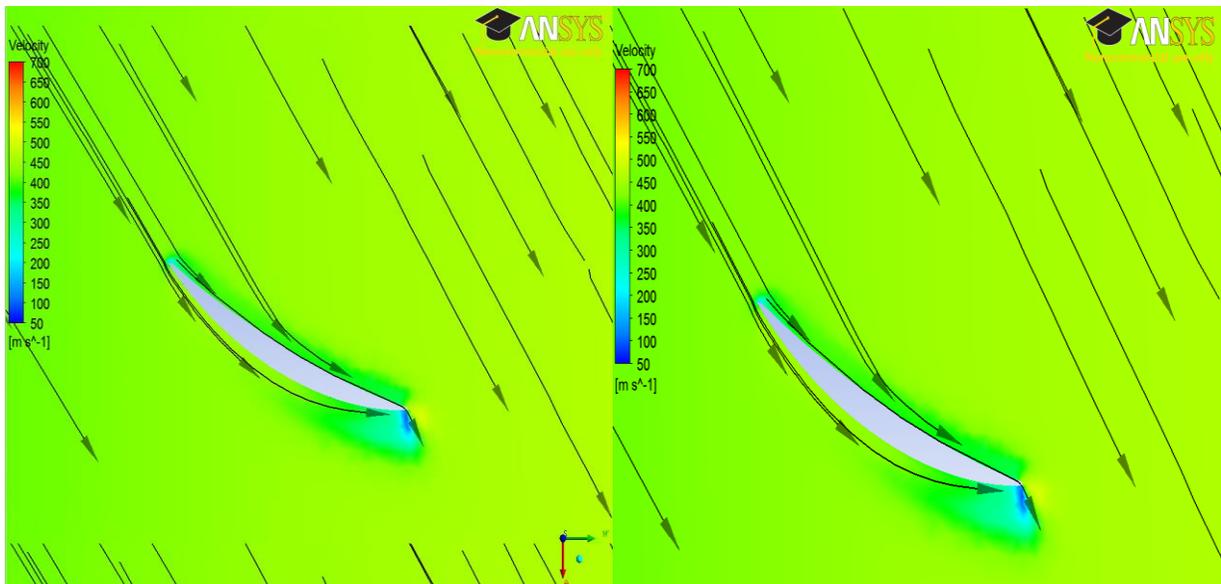
(Length units in cm)

**Figure 4.3 Velocity vectors at 80% span**

The above figures show the velocity vectors at 50% and 80% of span which show the effect of boundary layer separation or vortex formation on the blade. As compared with the results at 20% chord, it can be readily seen that the effects of boundary layer separations are more powerful at 20% chord.

For as such, the pressure ratio and efficiency for the models was calculated and found out to be around 1.52. This decrement in the pressure ratio as compared to the design pressure ratio is because of 2 things i.e. the computational error as well the design error, which is due to change in model to single blade and confining its range to 60 degrees.

So according to the results found here, appropriate steps were taken to control these boundary layer separations. And so the suction slot was applied at trailing edge of suction surface and two different suction mass flow rates were selected to verify the results.

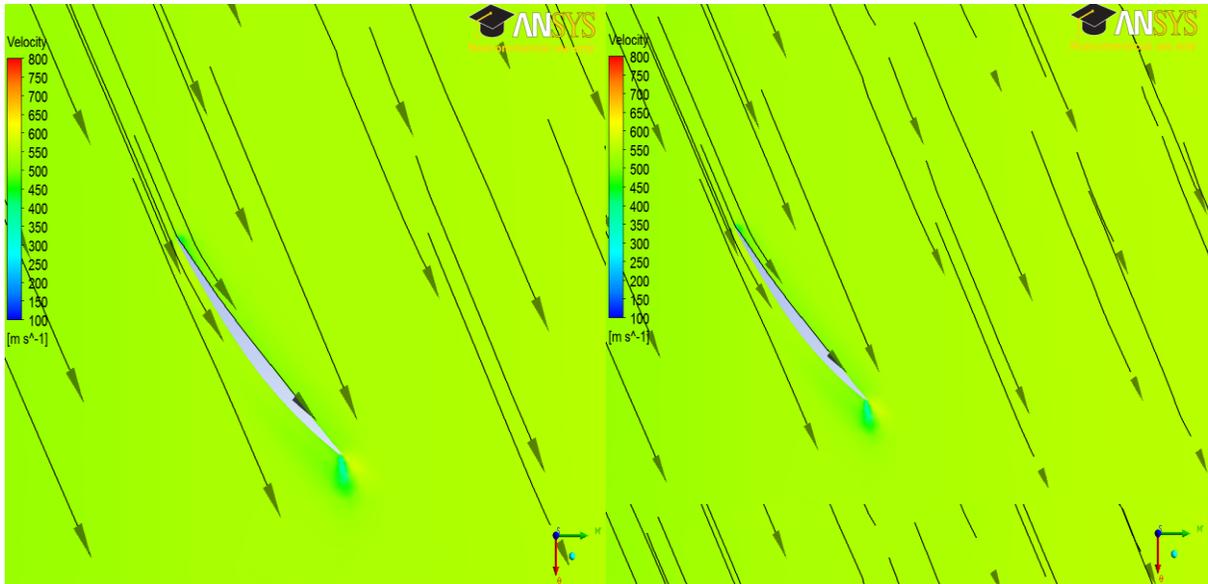


(Suction mass flow rate of 1.5 Kg/s)

(Suction mass flow rate of 1Kg/s)

**Figure 4.4 Velocity vectors at 20% span**

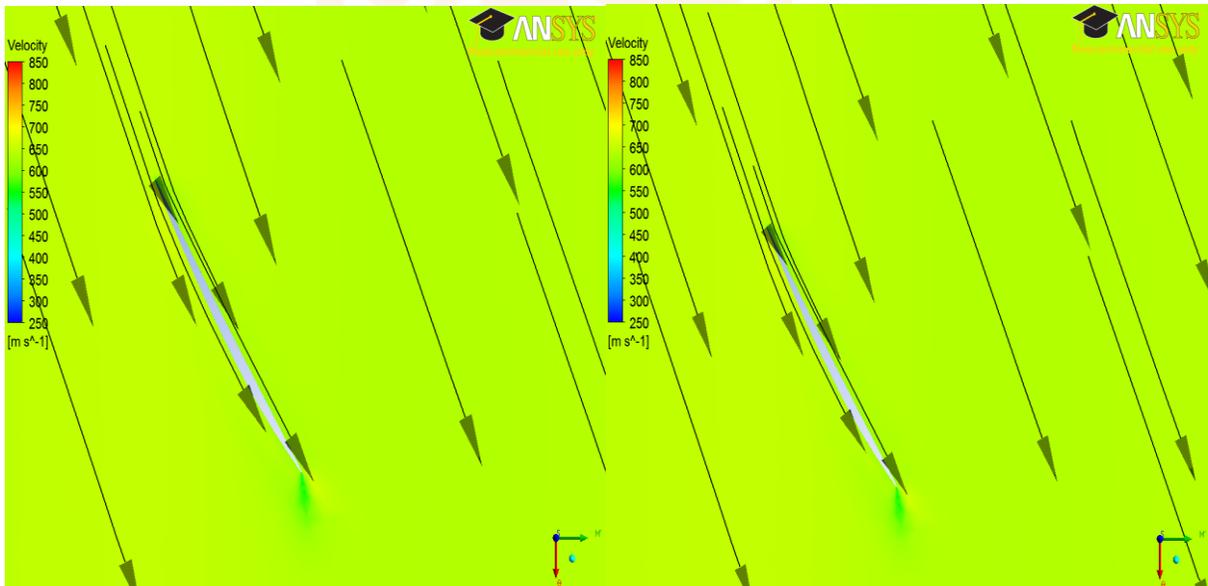
The figure shows how the boundary layer separation effect at 20% chord has changed with the application of suction boundary layer control technique. It can be readily seen that the velocity vectors have reattached, preventing the boundary layer separations at the suction surface of the blade.



(Suction mass flow rate of 1.5 Kg/s)

(Suction mass flow rate of 1Kg/s)

**Figure 4.5 Velocity vectors at 50% span**



(Suction mass flow rate of 1.5 Kg/s)

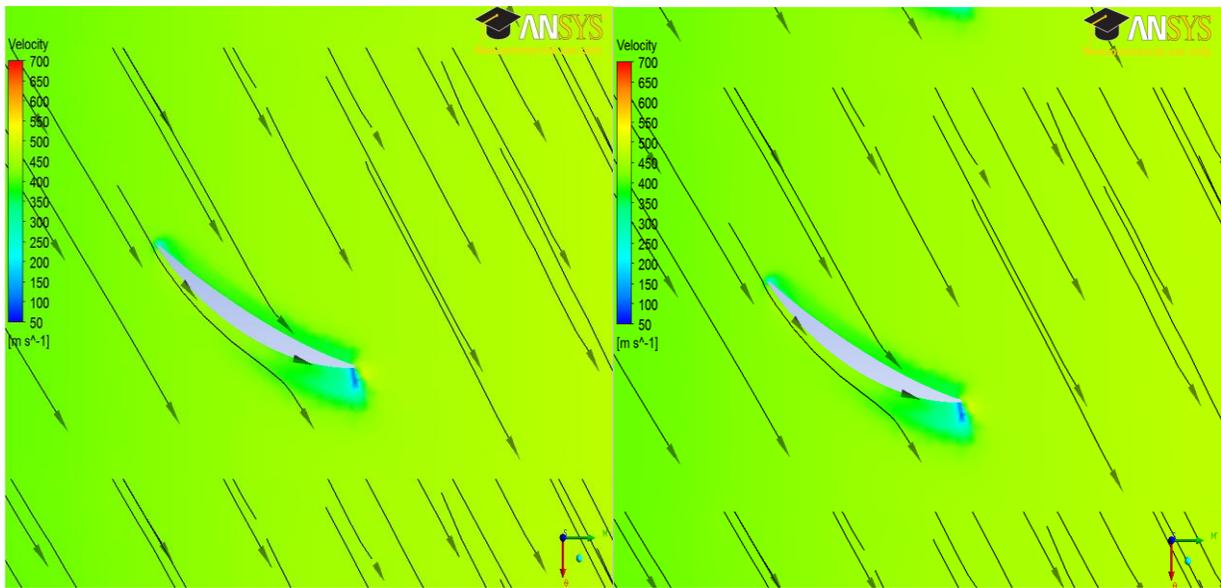
(Suction mass flow rate of 1Kg/s)

**Figure 4.6 Velocity vectors at 80% span**

From the above figures, it can be seen that the suction has not only prevented boundary layer separations at 20% chord but has also the decreased the effect of vortex formation at the trailing edge, as can be seen at 50% and 80 % span.

A disadvantage of suction as can be seen is increasing beta angle at trailing edge of the blade.

Now, blowing boundary layer control technique was checked with blowing slot at 55% chord. This technique was also checked with two different blowing mass flow rates of 1 Kg/s and 1.5 Kg/s.

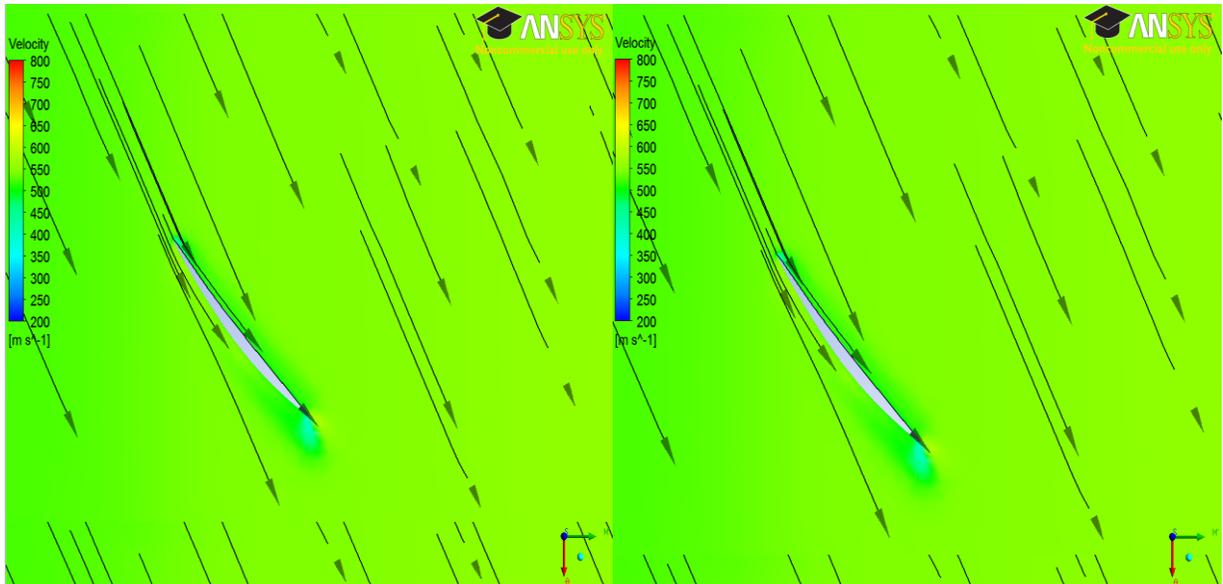


(Blowing mass flow rate of 1.5 Kg/s)

(Blowing mass flow rate of 1Kg/s)

**Figure 4.7 Velocity vectors at 20% span**

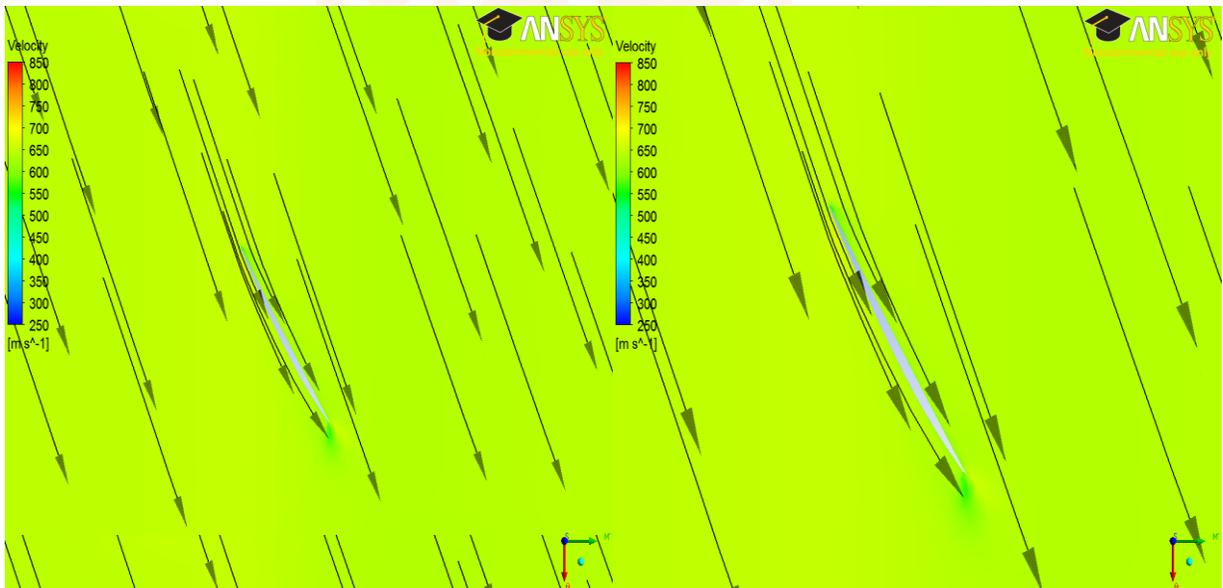
The figure above shows how the blowing boundary layer control technique has helped. It can be readily seen that the boundary layer separation has been pushed forward. So now the boundary layer separation occurs much ahead, which is a good thing seeing a only a little extra air is aspirated inside.



(Blowing mass flow rate of 1.5 Kg/s)

(Blowing mass flow rate of 1Kg/s)

**Figure 4.8 Velocity vectors at 50% span**



(Blowing mass flow rate of 1.5 Kg/s)

(Blowing mass flow rate of 1Kg/s)

**Figure 4.9 Velocity vectors at 80% span**

These figures show that the boundary layer control technique does not do much in the case of 50% and 80% span.

So to take the advantage of both suction and blowing, both the boundary layer techniques are used together. In this case the blowing mass flow rate of 1.5 kg/s was used, which was greater than the mass flow rate at suction which is 1 Kg/s. this was done according to literature review.



**Figure 4.10 Velocity vectors at 20% span**

In this figure it can be seen how the boundary layer separation have been moved to trailing edge. It also does not contain the disadvantage of the suction boundary layer control technique which causes the increment in beta angle. This shows that blowing and suction together increases the effect of blowing and suction boundary control.

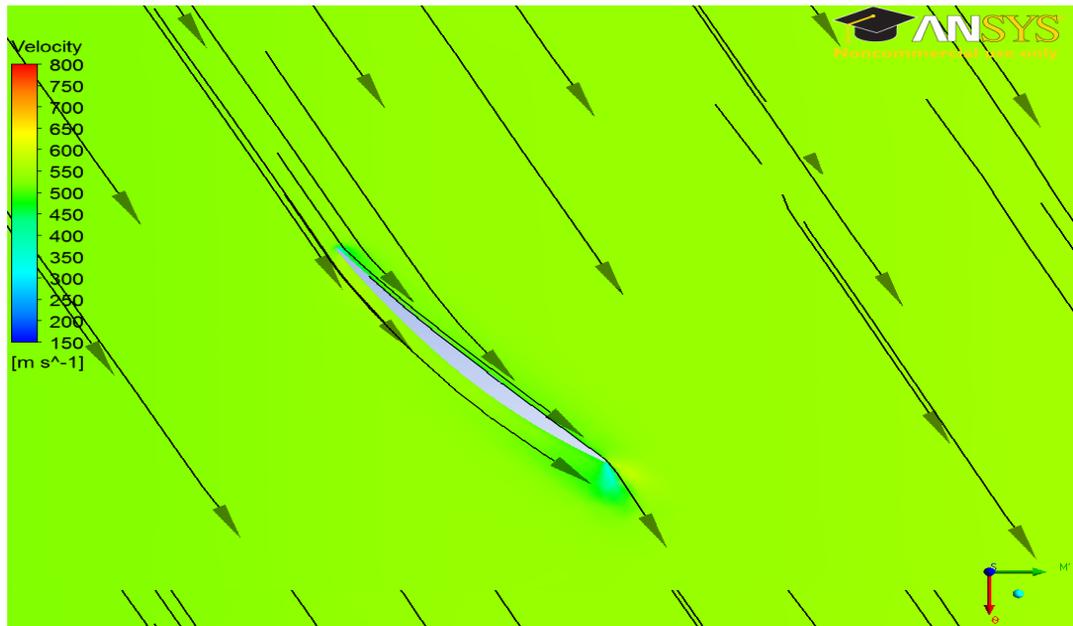


Figure 4.11 Velocity vectors at 50% span

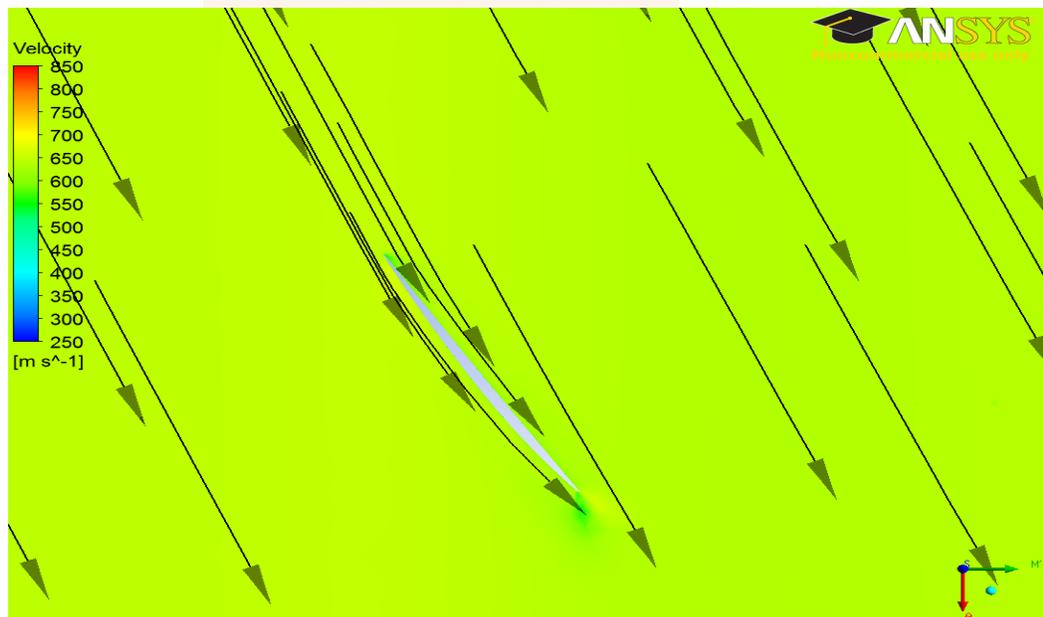


Figure 4.12 Velocity vectors at 80% span

It can also be seen that the effect of boundary layer separations have been decreased at 50% and 80% chord. So as compared with others the blowing and suction together have resulted in better results which can be easily seen. The boundary layer separations have decreased leading to a higher pressure ratio.

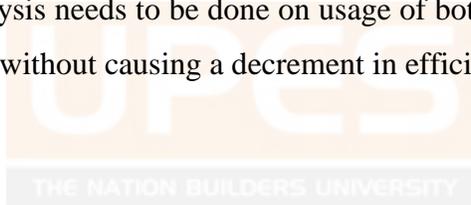


## CHAPTER 5 CONCLUSION

As is seen in the results most significant boundary layer separations occur at 20% chord with highest losses there. So to control those losses suction and blowing boundary layer techniques were used. The suction of 1 Kg/s and 1.5 Kg/s led to reattachment of boundary layer separation but also led to increment in beta angle. The Pressure ratio was seen to increase while using suction. The next boundary layer control technique used was blowing with mass flow rate of 1 and 1.5 Kg/s. This techniques does not fully prevented boundary layer separation but pushed the separation forward and also led to increment in pressure ratio, but not as much as suction.

So a combination of both blowing and suction was used which led to pushing of boundary layer separation on at trailing edge, without the changing the beta angle. Also it led to the much further increment in pressure ratio, but somehow led to decrement in efficiency. This decrement maybe caused due to the computational error.

So a more detailed analysis needs to be done on usage of both suction and blowing together on the blade so as can be used without causing a decrement in efficiency.



## **CHAPTER 6 REFERENCES**

1. Abate, Giada. (2003). Aerodynamic optimization of a transonic axial compressor rotor. *Journal of Propulsion and Power*. 54 (5), 41-55.
2. Anderson, John David (2012). *Computational Fluid Dynamics*. 4th ed. Delhi: Mc Graw Hill Education Pvt. Ltd.. 168-208.
3. Bazek, J (2001). *Computational Fluid Dynamics : Principles and Applications*. 8th ed. Switzerland: Elsevier Balcom PVT. Ltd.. 67-78.
4. Biollo, Roberto and Benini, Ernesto. (2006). State of art of Transonic Compressor . *International Journal of Gas Turbine, Propulsion and Power Systems*. 21 (8), 4-21.
5. Bruce E Poling , John M Prausnitz and John O conell (2004). *Properties of Gas and Liquids*. 5th ed. London: McGraw-Hill. 76-134.
6. Cumpsty, N (1989). *Compressor Aerodynamics*. London: Longman. 1-90.
7. Gander, L.B. and W.J Calvert. (1987). The Design of an advanced Fan Rotor. *ASME journal of Engineering for power* . 2 (1), 11-32.
8. Giarbo, A.. (2012). Performance assessment improvement of new Technology for VIGVs. Master Thesis, Tesi di laurea. 1 (1), 4-25.
9. G.R. Miller, G.W. Lewis, and M.J. Hartmann . (1961). Shock losses in Transonic rotor rows. *ASME journal of Engineering for power* . 8 (2), 54-68.
10. Himeno, Takehiro. (2008). Effect of Endwall Contouring on Flow Instability of Transonic Compressor. *International Journal of Gas Turbine, Propulsion and Power Systems*. 2 (1), 26-29.
11. H. Starcken and H.J. Lichfuss. (1970). Some experimental results on three dimensional compressor rotor. *ASME journal of Engineering for power* . 17 (5), 22-39.
12. Khalil, Mohammed K. (2009). High Speed Inlet Stage for an Advanced High-Pressure Ratio

Core Compressor: Numerical Flow Field Prediction and Verification. Aerospace Sciences and Aviation Technology, ASAT- 13. 13 (21), 1-15.

13. Laney, B. Culbert. (1998). Computational review. In: Taylor, C. Fayette *Computational Gas Dynamics*. Newyork: Cambridge University. 103-110.
14. Longhouse, R. (1977). Vortex Shedding of Low Tip Speed, Axial Flow Fans. Journal of sound and vibration. 53 (1), 25-46.
15. Mattingly, Jack D (2006). *Elements of Propulsion*. 3rd ed. Virginia: American Institute of aeronautics and astronautics. 705-767.
16. Sachdeva, Ankit. (2010). Study and control of Compressor Rotor Blade by Boundary layer Aspiration. International journal of Aerospace Engineering . 15 (4), 43-67.

