OPEN ACCESS

International
Journal
Of Modern Engineering Research (IJMER)

Effect of Different Boundary Layer Control Techniques on the Flow of Compressor Rotor Blade

Navneet Kumar[1],Rohit Yadav[2]Andkhushal Raj[3]
[1,2,3]*P.G. Student, Department Of Aerospace Engineering, Indian Institute Of Technology, Kanpur, Uttarpradesh, India*[1]

***ABSTRACT**: The performance characteristics of transonic aircraft majorly depend on the compressor rotor blade efficiency and engine weight. To increase rotor blade efficiency flow separation over rotor blade must be prevented and controlled. The aim was to modify and control the flow behaviour over compressor rotor blade by using three different boundary layer control techniques i.e suction, blowing and combination of suction and blowing and compare the effect of these three methods on flow around rotor blade.*

Rotor blade geometry has been modelled in CATIA V5 and then one suction surface and one blowing surface has been created on the blade surface. These geometries have been imported to ANSYS CFX 14.5 for computational simulation and analysis of rotor blade flow with these three techniques and without boundary layer control technique. Suction slot has been applied at the trailing edge of the suction surface and blowing slot has been applied at 55% of the blowing surface and shear stress transport model has been used for computational analysis.

Two mass flow rates of 1kg/s and 1.5kg/s have been used here for boundary layer control method and boundary layer separation effects have been observed and this could be readily seen as the reattachment of velocity vectors which prevented the flow separation and increment in pressure ratio has been found to be 0.052, 0.09 and 0.12 for suction, blowing and combination of these two methods respectively.

***Keywords**: Boundary Layer, Blowing, compressor, isentropic efficiency, Rotor blade, Pressure ratio, Suction*

I. INTRODUCTION

Efficiency of aircraft gas turbine engines in transonic flow can be increased with the use of compressor having multiple stages and high pressure ratio but this kind of configuration leads to increment in weight and cost also. A lot of researches were done in this field and our aim is achieving high pressure ratio so that efficiency of compressor can be increased. Performance of compressor blade in transonic flow which is partly high subsonic and low supersonic is very important due to occurrence of shock wave which leads to flow separation and this causes different type of losses and this will affect the performance of compressor and therefore performance of engine also reduces. To avoid these losses and to prevent or delay separation different methods like actuating devices, modification in geometry of axial fans etc. are used today and in this paper BL control methods are used to delay or modify flow separation of blade.

To reduce the losses and delay or prevent separation caused by shock different kind of modifications has been applied to the rotor blade model some of them are introducing plasma actuating devices on the trailing edge of the blade, Boundary layer control methods such as suction, blowing etc. In the present study the flow field of transonic axial compressor rotor blade has been numerically simulated to clarify the mechanism of boundary layer control by suction, blowing and combination of these two methods simultaneously.

Axial compressors are designed and developed in such a manner that these could be used to operate under steady axisymmetric flow, where pressure rise across compressor depends upon compressor mass flow rate. In axial compressor flow enters the first blade and leaves axially which is parallel to the axis of shaft. Higher mass flow rates give higher thrust. Axial compressor has lower pressure ratio per stage then the centrifugal one. In transonic aircraft axial flow compressors have been used therefore to increase performance of transonic aircraft pressure ratio across blades of the compressor should be high and to achieve this flow over the blade needs to be modified.To prevent compressor flow instabilities different methods and devices are available today. Methods like tiny grooves or slot on endwalls, casing treatments etc. have been employed to compressor rotor blade in the late 1980s'. In Transonic flow performance is very important because this type of flow is partly high subsonic and low supersonic which leads to the formation of shock and due to this losses increases and overall engine performance reduces. Compressor rotor blade geometry is very important because its geometry decides where shock appears on the blade and therefore separation of flow depends on the blade geometry.

1.1 Rotor Blade Specifications
Table 1 Properties of Rotor Blade

Design Pressure Ratio - 1.61
Mass flow rate - 32 kg/s
Rotational speed - 16000
Tip speed - 426 m/s
Solidity at the hub - 2.95
Solidity at the tip - 1.33

1.2 Boundary layer and losses

Boundary layer is the layer where flow is retarded due to significant viscosity effect and this layer play key role in deciding performance parameters because flow separation depends on this layer and parameters like pressure, entropy, temperature, viscous layer depends on flow separation. Early flow separation reduces the performance of engines therefore boundary layer needs to be controlled to delay or prevent separation.
In this type of turbo machinery loss mechanism is divided into two groups i.e. Internal and Parasitic loss mechanism. Internal is associated with the main flow through the compressor while parasitic is associated with minor flow loss leaking away from major flow of the compressor. Major losses in transonic flow compressor are as follows:

- Shock
- Blade loading and leakage
- Clearance and profile
- Annulus and secondary

1.3 Boundary Layer control methods

BL is the region where mass, momentum and heat transfer are felt and due to these flow is retarded and undergoes various types of losses which reduces performance therefore BL control is necessary to achieve desire performance. BL control refers to the controlling behaviour of fluid flow over rotor blades.
Suction, blowing and combination of suction and blowing methods are used simultaneously here as BL control methods. In Suction method a suction surface and suction hole has been created on blade surface and which helps in reenergizing the flow by sucking low energy fluid on the blade surface whereas in Blowing type blowing surface and blowing hole has been created near half of the chord of blade surface through which high energy fluid has been blown over the low energy fluid flow surfaces and this high energy fluid got mixed with low energy fluid and as a result energy of previously fluid has been increased and flow over blade surfaces becomes more smooth and separation has been delayed or prevented. Features of suction and blowing methods are combined in one single method and then observing the effect of this combined method.

1.4 Equation

Isentropic efficiency

$$\eta = \frac{PR^{\frac{\gamma-1}{\gamma}} - 1}{\frac{T_2}{T_1 - 1}} \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \text{(1)}$$

In the “equation 1” PR is the pressure ratio and γ is the specific heat ratio for air which has the standard numerical value of 1.4. T_2 &T_1 are the total temperatures of outlet and inlet respectively.

II. RELATED WORK

To increase the performance of transonic aircraft many researches has been made in this field. There are various devices and methods available today to prevent compressor flow instabilities. Some of the methods are casing treatment, tiny grooves or slots made on endwall etc. and these methods have been employed to compressor rotor blade in the late 1970s’. Many methods like adding plasma actuating devices on the trailing edge of blade, Boundary layer control by suction etc. have been developed to delay or prevent separation.

Features of suction and blowing method has been used separately on the rotor blade separately but not at the same time. Before using these control surfaces in our geometry we have done a lot of literature survey to get appropriate coordinates of rotor blade surface and we got these points from the researches made in 90's. Suction and blowing has been used in many other researches to prevent flow separation and these methods has delayed flow separation and reduces the boundary layer effects especially on airfoils. Due to the occurrence of shocks in transonic flow, Performance of compressor rotor has been decreased therefore researches has been made in this field so that performance of the rotor could be increased by some means.

In the previous researches suction and blowing both methods has been used but in this research we want to use features of both the methods at the same time and by this mean we want to avoid or delay flow separation so that flow remain attached to the blade and velocity vectors should be smooth so that we got high pressure ratio per stage. When suction method has been used in the previous researches then increment in pressure ratio was not much effective and when blowing has been used then increment in pressure ratio was more than suction therefore we got the idea of using both the methods at the same time to reduce compressor instabilities and to increase performance of transonic axial compressor.

III. METHODOLOGY

3.1 Modelling

Modelling of rotor blades has been done in CATIA V5 (a software used for modelling and designing any object).Modelling of compressor rotor blade is not an easy job due to complexity of curve i.e. it is twisted from hub to tip, unless we do not have proper coordinates of points, blade modelling is not possible and these coordinates of point can be obtained from literature survey of researches made in this field. First modelling of single surface has been done by joining coordinates of four points and then formation of all surfaces of blade has been done by joining coordinates of all the points then geometric model of blade is ready and after this casing hubs and outer walls of blades has been modelled and this depends on the tip clearance chosen.

Table 2 Important parameters of Model

Maximum chord length	143.68
Maximum span	250.94
Inlet Diameter	202.362
Outlet Diameter	147.638
Tip Clearance	4.26

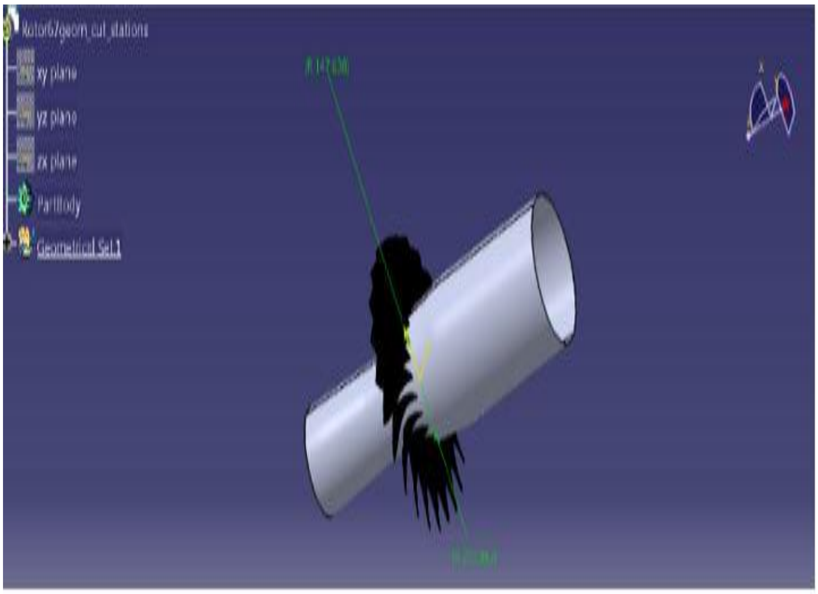


Figure1 Geometric model of rotor blades

There were thirty six blade in single rotor stage of compressor which has been shown in Figure1 and number of elements corresponding to each blade was very high due to which this would take too much of computational time and cost.

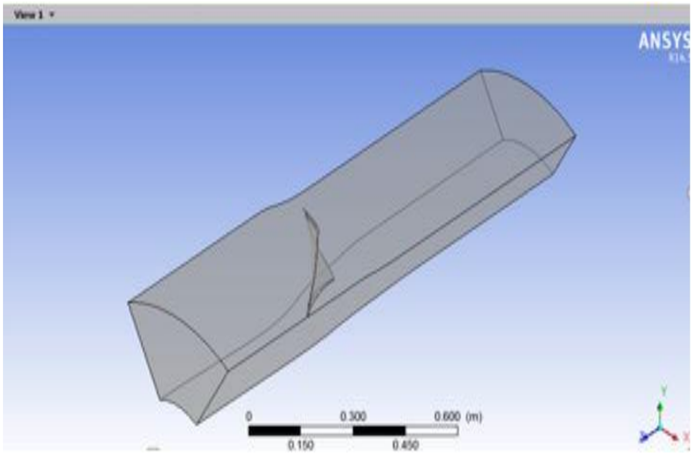


Figure2 Single blade model with outer walls and casing

To overcome this problem and to reduce our computational cost and time our model has been restricted to single blade geometric model which has been shown below in Figure2 and results and analysis has been done for this model only.

3.2 Grid and mesh parameters

3.2.1 Grid generation and mesh parameters without boundary layer control

Grid creation was very important phase in CFD simulations and if grid was of bad quality then our results were not good. In this paper for shell meshing Quad dominant mesh type has been used and patch dependent method has been used for obtaining this and for volume mesh tetra or mixed type mesh has been used and robust method helps in achieving this.

After completion of geometric modelling in designing software CATIA V5 then geometry has been imported to ICEM CFD 14.5 and parts has been created here and all parts were assigned a name. This parts creation has been done because by doing this it was possible to create finer grids in some parts where flow parameters were very important. In importing the model from CATIA to ICEM blade geometry has been distorted therefore when all the parts has been created geometry should be repaired by providing tolerance to 0.1 and this tolerance has been selected on the basis of smallest length associated with the model. Once repairing process has been done then delete the overlapped surfaces and any holes in the geometry and then model was ready for grid generation. In importing geometry some surfaces were lost, some holes were appearing and surfaces were overlapped also and to avoid this repairing process was required.After Repairing process body should be created by selecting two points diagonally, by doing so fine mesh has been created on blade then the outer wall and casing.

To consider boundary layer effects on the flow height was kept as constant and number of prism layers has been calculated using Y+ calculators.

Part Mesh Setup

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	☑		3	0	0	0	0	0	0	0	☐	☐
BODY	☐	☐										
CASING	☐		60	0	0	0	0	0	0	0	☐	☐
FILL.7	☐		0						0	0		
FILL.8	☐		0						0	0		
FILL.9	☐		0						0	0		
FILL.10	☐		0						0	0		
FILL.16	☐		0						0	0		
FILL.17	☐		0						0	0		
FILL.18	☐		0						0	0		
GEOM	☐		0						0	0		
HUB	☐		60	0	0	0	0	0	0	0	☐	☐
INLET	☐		60	0	0	0	0	0	0	0	☐	☐
JOIN.2												
LINE.17												
LINE.18												
LOWER_BLADE	☑		1	0	0	0	0	0	0	0	☐	☐
OUTLET	☐		60	0	0	0	0	0	0	0	☐	☐
PART_2	☐		0						0	0		
PART_3	☐		0						0	0		
PART_4	☐		0						0	0		
PART_5	☐		0						0	0		
PART_6	☐		0						0	0		
PART_7	☐		0						0	0		
PART_8	☐		0						0	0		
PART_9	☐		0						0	0		
PART_10	☐		0						0	0		
PART_11	☐		0						0	0		
PART_12	☐		0						0	0		
PART_13	☐		0						0	0		

Figure3 Parameters setup for part mesh

First global mesh parameters has been defined and after this part mesh parameters has been defined which has been shown in the above Figure3 In the part mesh setup different element sizes have been defined to create coarsen mesh on the edge of the blade. When all the parameters have been defined then create prism layer and then apply compute for generating mesh grids. Grid generation of single blade has been shown in Figure4 and Grid of outer walls and casing has been shown in Figure5.

Table 3 Mesh parameters of Grids of model with and without blowing boundary layer control surface

Properties	Without boundary layer control	With suction BL control	With blowing BL control	With Combination of suction and blowing simultaneously BL control
Initial Height	0.000169291	0.000169291	0.000169291	0.000169291
Total Height	0.0121949	0.0121949	0.0121949	0.0121949
Total Elements	1669986	1704628	1716482	1786968
Total Nodes	566300	579554	585573	610165

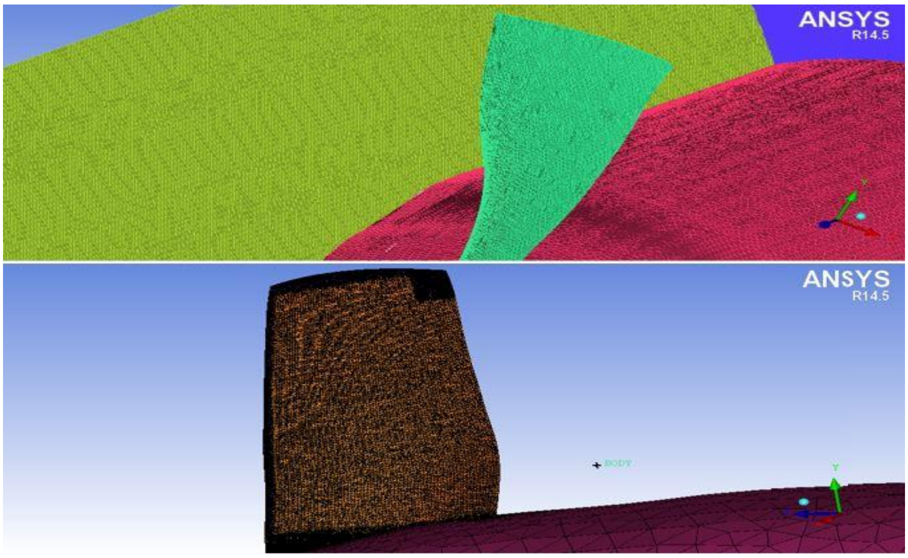


Figure4 Grid generation of blade on single blade model

Grid or mesh generation on the blade surfaces has been shown in Fig.4 and two different views has been shown in this Fig. at the surface of blade grids were very fine because this was the domain where flow behaviours were very important.

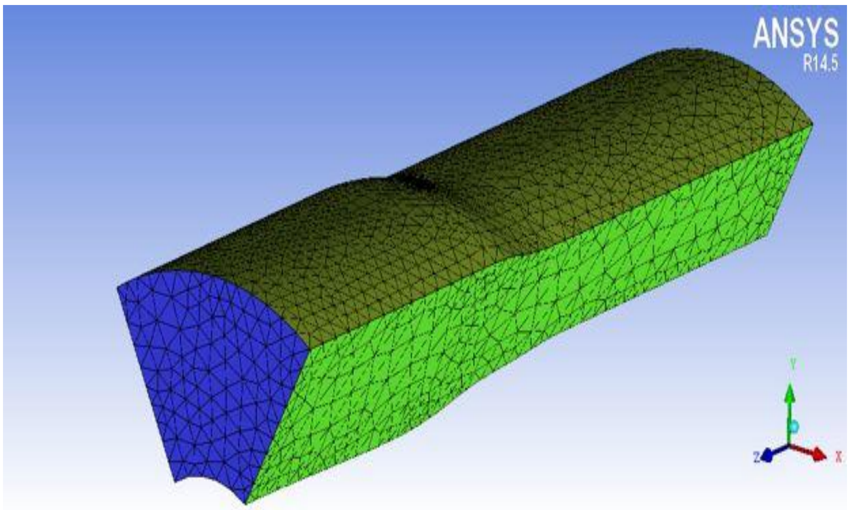


Figure5 Grid generation of blade on outer walls and casing

Grids formation on the geometry other than blade surface has been shown in the Figure5 and grids of these outer walls and casings were not very fine because flow behaviour around these surfaces were out of our interest.

3.2.2Grid generation with boundary layer control surface

In the first method one new suction surface with suction hole have been employed in the blade geometry and while creating parts one new part named as suction layer was created and in the blowing method blowing surface with hole have been created and prism layer has been employed for both cases. All other steps were similar to the previous section. In last method this geometry has two control surfaces and two holes. One was suction surface and other was blowing surface and one was suction hole and other was blowing hole. In the part creation method these surfaces were also assigned a name and prism layer were calculated for both suction and blowing surface. Prism layers were applied to these surfaces because we want finer mesh on these surfaces also and remaining steps to develop grids were similar to previous case.

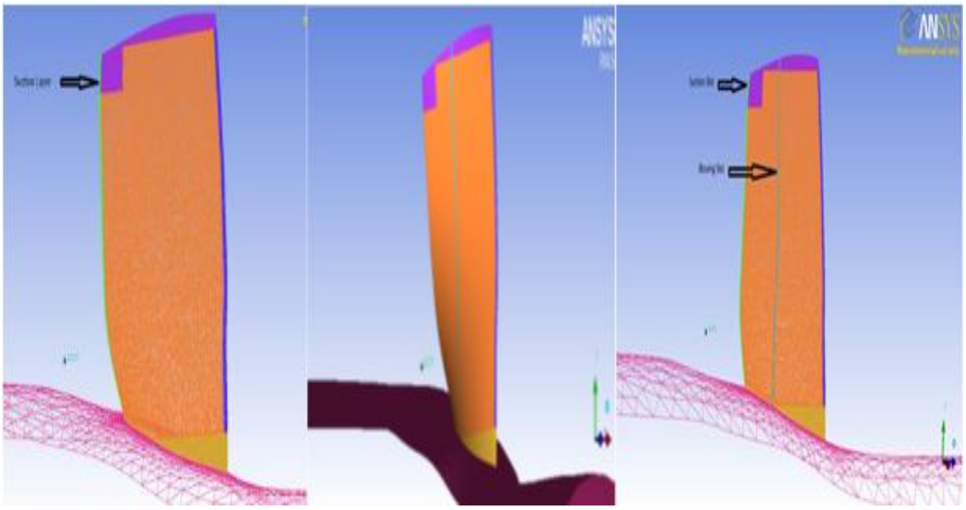


Figure6 Grid generation of blade model with control surfaces

Grid generation of control surfaces for all three methods has been shown in Fig.6 and on the suction and blowing surface grids were very fine because we have more interest on the flow behaviour on these surfaces.

3.3 Boundary conditions for computational simulation

Computational Fluid Dynamics has been used to solve the model associated with the fluid flow by applying numerical methods. There were many CFD software's like ANSYS- (GAMBIT, FLUENT and CFX etc.), OPEN FOAM, Gerrish Flow solver etc. available today for computational simulation and analysis. Among all these software's ANSYS CFX was high performance CFD tool which delivers accurate and reliable solutions therefore ANSYS CFX has been used in our study. This software has three tools to do computational simulation and analysis and they were CFX-pre-processor, CFX solver manager and CFX-post processor.

3.3.1CFX pre-processor

This part of CFD tool worked as pre-processor for the simulation software. Grid file produced in ICEM should be imported to the CFX-pre by generating CFX input file of that model. After this open this file from CFX-pre and then domain should be created in pre-processing software and this problem is defined in turbo mode. Turbo mode has been used to provide some pre-specified conditions.

3.3.1.1 CFX Pre **setup** for model without boundary layer control surfaces
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

3.3.1.2 CFX Pre setup for model with boundary layer control surfaces

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 was most effective in controlling the boundary layer separation. So mass flow rates of 1Kg/s and 1.5Kg/s were used. The blowing boundary layer was applied as outlet and the two conditions were put up on it according to the two mass flow rates selected. Mass flow rates of 1 kg/s and 1.5 kg/s has been used in this case.

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
- ➢ Blowing:
- ❖ Mass and Momentum :- Inlet and mass flow rate of 1 Kg/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

3.3.2 CFX solver manager

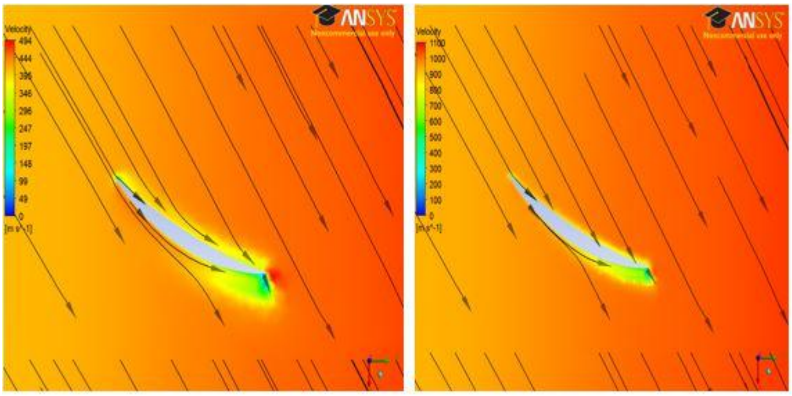
Output file generated in previous file was the input file for this part and for computational simulation graphical interface method has been used by this tool. In our study Double precision has been used because this would give results with more accuracy but when this solver has been applied to execute the file this would take double computational memory.

3.3.3 CFX post-processor

The result or output file of CFX solver manager worked as input file for this tool and first that output file has been loaded in to this tool. When the results were initialized thermodynamic properties can also be extracted from optimization process therefore expression of isentropic efficiency and pressure ratio has been also created in expression tab of CFD-Post.

IV. RESULTS AND DISCUSSION

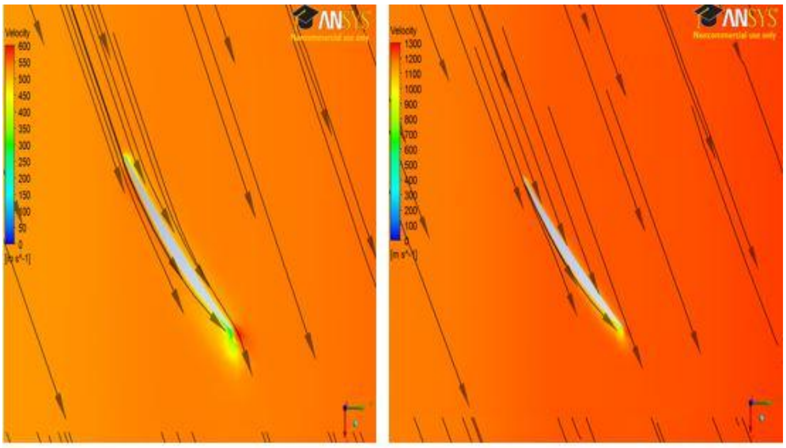
There were two defined models for analysis and this analysis has been done in CFX post-processor. First model without any boundary layer control surfaces has been analysed and then model with combination of suction and blowing boundary layer control surfaces. Three dimensional simulations have been done by using simulation software ANSYS CFX. In post processing results have been analysed in the form of velocity vectors. Velocity vectors have been checked not at various section of span but also at two models with changed length units and rpm.



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

Figure7 Velocity vectors at 25% span chord location

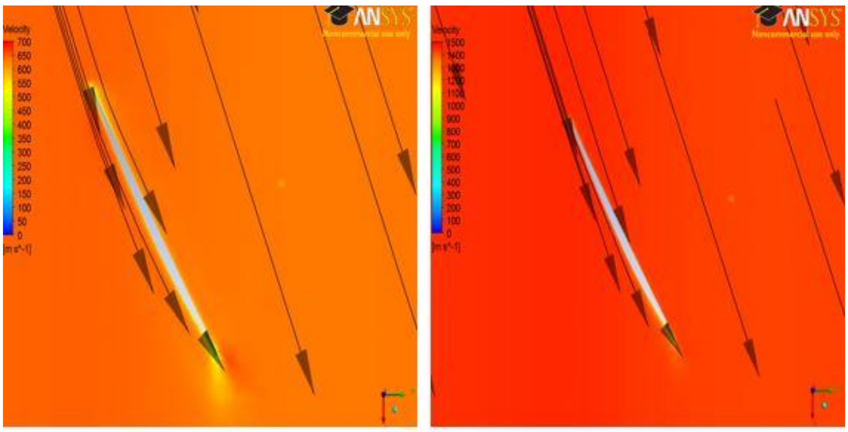
Flow behaviour in the form of velocity vectors at quarter chord location has been shown in the above fig. and this has been done for two different scale and vortex of strong strength could be seen here as the flow has been separated and boundary layer effects were dominant here.



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

Figure8 Velocity vectors at 50% span chord location

Flow behaviour at half of chord location has been shown in the above fig. and vortex of small strength has been observed and boundary layer effects were not dominant in this case.



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

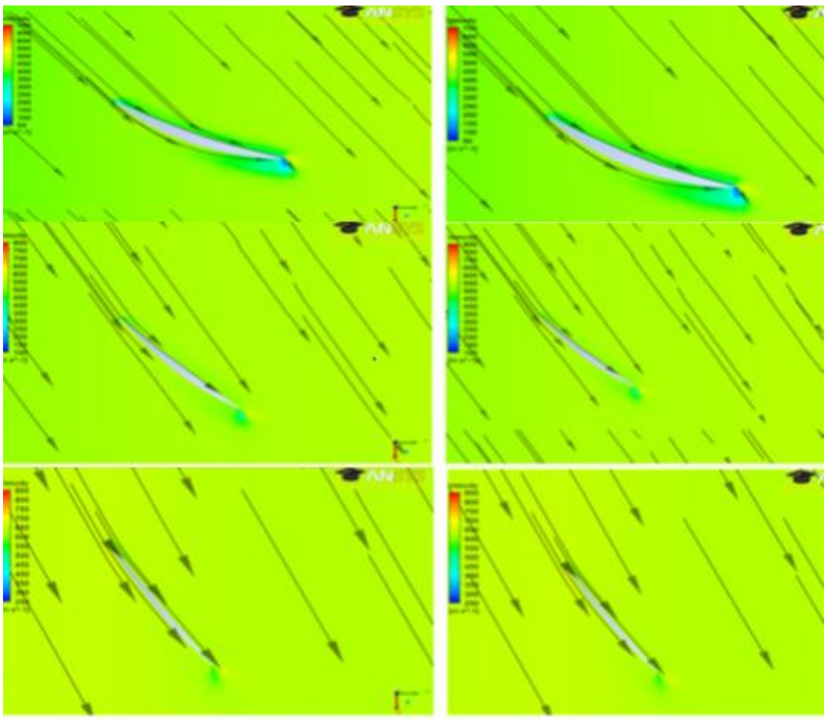
Figure9 Velocity vectors at 80% span chord location

Flow behaviour at 80% chord location has been shown in Fig.9 and two different length scale has been used and it has been found that results were similar for both length scale and in this case very tiny strength vortex has been observed and Boundary layer effects were negligible.

Figure.7, 8 and 9 represented velocity vectors at 25%, 50% and 80% span location respectively. When results have been analysed then it has been observed that boundary layer effects were dominant near quarter chord of span which could be seen in the above Fig.7 At quarter chord location flow separation has been occurred i.ecolor changes from orange to green and some strong vortex has been also formed at this location. At 50% and 80% span location flow separation has been also occurred but boundary layer effects were not as dominant as in near quarter chord location and very small vortex of tiny strength has been formed also and this could be seen in Figure.8 and Figure.9.

Flow analysis has been done for models with two different length scale and it was found that results were similar for both models. Results have been shown in the form of velocity vectors, at quarter chord location velocity vectors were not smooth as the boundary layer effects were dominant here and due to this flow separation has been occurred and this could be seen in the above three Fig. and at other two span location velocity vectors has been found to be smoother than at quarter chord location. From the results obtained it was clear that we need to develop some steps through which flow separation could be delayed or prevented and for this three boundary layer control techniques have been employed. Suction, blowing and combination of suction and blowing methods have been used in this study.

Results for the model with suction surface, blowing surface and combination of suction surface and blowing surface have been analysed and velocity vectors for three span location of blade i.e. at 25%, 50% and 80% span location which have been shown in Figure.10,11 and 12 respectively. In the suction method (suck out low energy fluid which causes flow separation), suction surface and suction hole has been created on the blade surface and results has been obtained for mass flow rate 1kg/s and in blowing method (high energy fluid mixed with the low energy fluid and eliminates the flow separation), blowing surface and blowing hole has been created on the blade surface and results has been obtained for two mass flow rate 1.5kg/s. Suction surface has been used to suck the low energy fluid and blowing surface has been used to blow high energy fluid to the flow and here feature of both the methods have been used separately and features of these two methods were combined and used simultaneously as single BL control method.



(Suction mass flow rate of 1.5 Kg/s) (Suction mass flow rate of 1Kg/s)

Figure10 Suction BL control method velocity vectors at three different span locations

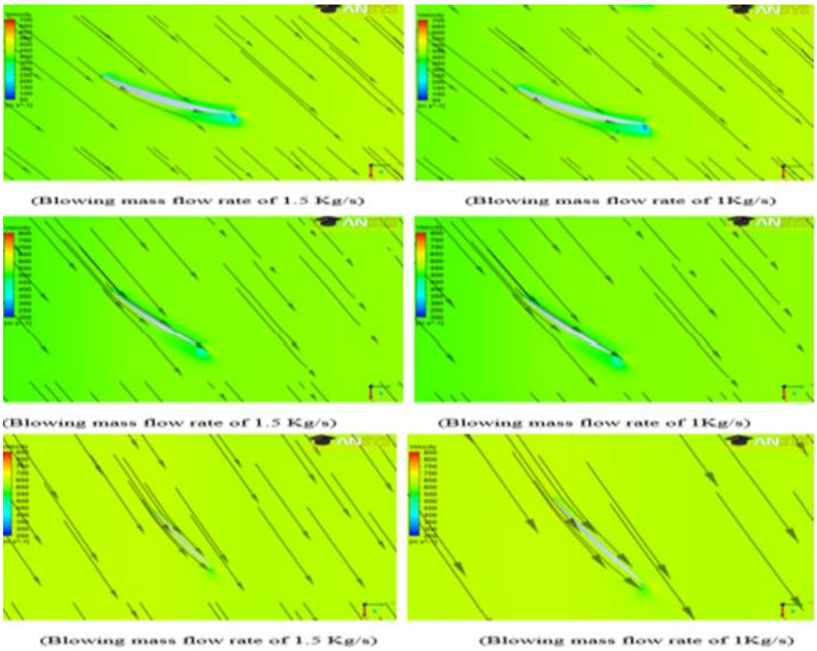


Figure11 Blowing BL control method velocity vectors at three different span locations

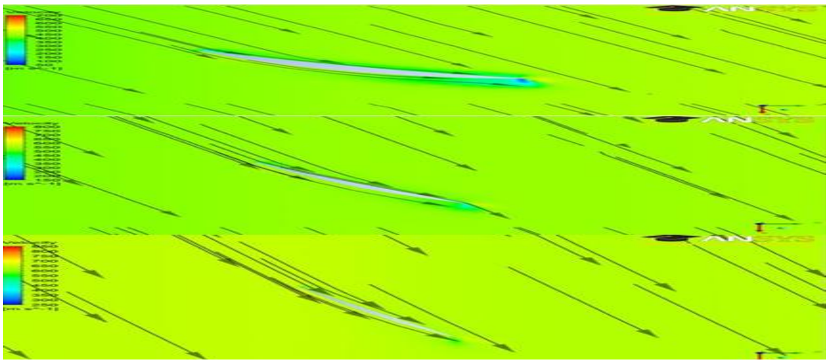


Figure12 velocity vectors for model with combined suction and blowing control surfaces

Flow behaviour at the quarter chord location has been shown in first part of Figure.10, 11 and 12 and in this case suction and blowing control surfaces has been introduced in the geometry and this could be observed that boundary layer effects were not dominant and some vortex of small strength has been observed also. Flow behaviour in the form of velocity vectors at half chord location with the introduction of suction and blowing control surfaces have been shown in second part of above three Figures and velocity vectors were smooth and very tiny vortex could be seen.Flow behaviour in the form of velocity vectors at 80% chord location with the introduction of blowing and suction control surfaces has been shown in the last part of above three figures and in this no vortex has been seen and flow separation has been prevented.

High energy fluid has been blown to the flow through blowing hole and suction surface used to suck the low energy fluid through suction holes and in this study both the processes has been worked simultaneously and fluid has been reenergized and there were no low energy fluid due to which flow separation occurred by this way flow separation has been prevented. From the above figure it could be shown that by the use of boundary layer control method velocity vectors found to be smoother and no strong vortex has been seen.

V. CONCLUSIONS

Results has been obtained for the both model i.e. model with and without boundary layer control surfaces and it has been obtained that flow separation has been delayed or prevented with the introduction of combination of suction and blowing surface in the blade geometry. When there was no boundary layer control surface then flow separation occurred and boundary layer effects were dominant near quarter chord location due to this some strong vortices has been formed but with the introduction of blowing and suction control surface in the blade geometry flow separation has been prevented and vortex of very tiny strength has been observed. As a result pressure ratio has been observed as 1.652, 1.690 and 1.724and increment in pressure ratio has been found to be 0.052, 0.09 and 0.11 for suction, blowing and combination of suction and blowing methods respectively.

ACKNOWLEDGEMENTS

Authors would like to thank university for providing computational facilities and colleagues for their continuous support. Special thanks to co-author Late Twisha Patel for her support. I heartily pay my gratitude to my colleagues Late Twisha Patel as she had been constantly helping me in it. Without their co-operation and support, working on this paper would have been very difficult.

REFERENCES

[1]. Kumar Navneet, "Flow modification over rotor blade with suction boundary layer control technique", International Journal of Engineering Research and Application,Vol.6, no.6,pp 1-5, 2016.
[2]. Kumar N., "Modification of rotor blade flow with blowing boundary layer control technique", International Journal of Multidisciplinary Educational Research, vol.5, no.7, pp209-228, 2016.
[3]. Kumar Navneet and Patel Twisha, "Flow Modification over Rotor Blade with Boundary Layer Control Technique", International Journal of Innovative Research in Science, Engineering and Technology, vol.5, Issue8, pp 15211-23, 2016.
[4]. Y. Ito, T. Watanabe and T. Himeno, "Effect of Endwall Countering on Flow Instability of Transonic Compressor", International Journal of Gas Turbine, Propulsion and Power system, Vol.2, no.1, 2008.
[5]. N. Ananthkrishnan, U. G. Vaidya and V. W. Walimbe, "Global stability and control analysis of axial compressor stall and surge phenomena using bifurcation method", Institution of mechanical Engineers J Power and Energy, vol.217, no.1, 2001.
[6]. Abate, Giada,"Aerodynamic optimization of transonic axial compressor rotor", Journal of Propulsion and Power, vol.54, no.4, pp.41-55, 2003.
[7]. Benini and Ernesto,"Three dimensional multi objective design aerodynamic optimization of transonic compressor rotor",International Journal of Gas Turbine, Propulsion and Power System, vol.54, no.5, 2004.
[8]. Brian Joseph Schuler, "Experimental Investigation of an aspirated fan stage, doctoral diss.", Massachusetts Institute of Technology, Cambridge, MA, 2001.
[9]. A. Sarkar, B. Singla and N. Kumar, "CFD analysis of Rotor 67 Blades, University of Petroleum and Energy Studies", Dehradun, India pp.23-52, 2015.
[10]. Tangler, James L. The evolution of rotor and blade design.National Renewable Energy Laboratory, 2000.
[11]. Lachmann, Gustav Victor, ed. Boundary layer and flow control: its principles and application. Elsevier, 2014.
[12]. Reijnen, Duncan Peter. Experimental study of boundary layer suction in a transonic compressor.Diss. Massachusetts Institute of Technology, 1997.
[13]. Tindell, Runyon H. "Blown boundary layer control system for a jet aircraft." U.S. Patent No. 5,447,283. 5 Sep. 1995.
[14]. Macaulay, Jack R., et al. "Aircraft boundary-layer control system." U.S. Patent No. 2,920,844. 12 Jan. 1960.