RESEARCH ARTICLE   OPEN ACCESS

Flow Modification over Rotor Blade with Suction Boundary Layer Control Technique

Navneet Kumar
(Department of Aerospace Engineering, Indian Institute of Technology, Kanpur-16)

ABSTRACT
The efficiency of transonic aircraft engines depend upon the performance of compressor rotor. To increase compressor rotors performance flow separation around rotor blades must be delayed and controlled. The aim was to control the flow separation of blades using suction boundary layer control method.
Rotor blade has been modelled in designing software CATIA and then a suction surface has been created on blade and then import these geometries to ANSYS-CFX 14.5 for computational analysis of flow around blades. Suction slot has been applied at the trailing edge of suction surface and Shear stress transport model has been used for computational analysis.
Two different suction mass flow rates 1 kg/s and 1.5 kg/s have been used here and boundary layer separation effects have been changed and this could be readily seen that the velocity vectors have reattached, preventing the boundary layer separation at the suction surface of the blade.
Keywords – Boundary Layer, Compressor, Pressure ratio, Rotor blade, Suction

I. INTRODUCTION

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 onendwall etc. and these methods have been employed to compressor rotor blade in the late 1970s'.

In Transonic flow which is partly high subsonic and low supersonic geometry of compressor rotor blade is very important due to occurrence of shock performance of compressor depends upon rotor blade. Shock causes separation which increases losses and as a result efficiency of engine decreases therefore 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.

In the present study the flow field of transonic compressor rotor has been numerically simulated to clarify the mechanism of Boundary layer control by suction.

1.1 Compressor Rotor specifications
Rotor which has been used here was a low aspect ratio transonic axial flow fan having more than twenty blades. The design pressure ratio was nearly about 1.68, mass flow rate was 33 kg/s, having rotational speed of 16000 RPM and tip speed was 425 m/s. The solidity of rotor fan varies from 3.1 at the hub and 1.29 at the tip.

1.2 Boundary layer and Boundary layer control method

Boundary layer is the region where mass, momentum, heat transfers are felt, viscosity effect is significant here and due to these effects flow is retarded and cause separation. Flow separation depends on this layer and parameters like entropy layer, viscous layer, pressure and temperature depends on separation therefore this layer play key role in deciding performance. To increase performance separation must be prevented or delayed to get desired flow parameters.

To overcome the losses caused by separation and to achieve desire flow parameters boundary layer control methods are used. In this paper suction hole has been created on the rotor blade surface in such a manner that this will be used to reenergize the low energy fluid so that the separation point shifts and reduces the no slip condition.

1.3 Equation
Isentropic Efficiency:-

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

In the "equation 1" PR is the pressure ratio and γ 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.

II. METHODOLOGY

2.1 Modeling

Designing software CATIA has been used for the modelling of blades of compressor. Four points were joined together to form one surface and these points were taken on the basis of literature survey of previous technical paper. Due to complexity of curve of blade (it is twisted from hub to tip) modelling of compressor blade was not an easy job to do. If we do not have coordinates of point then modelling of rotor blade was not possible in this software and once coordinates of points were decided then by connecting all the points blade surfaces has been formed and geometric model of blade was ready and after this outer walls and casing hubs has been designed in this designing software.
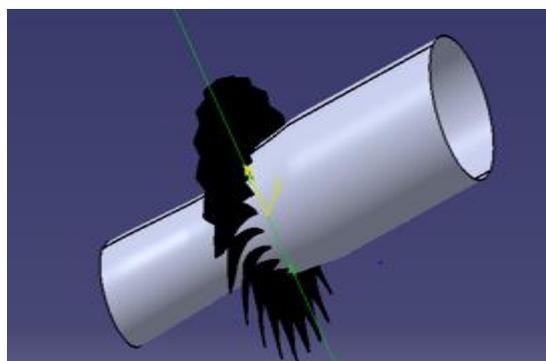
Figure1 modelling of Rotor blades

Table 1 Important Parameters of Model in mm

Maximum Chord length – 143
Maximum Span-- 250
Inlet Diameter-- 202
Outlet Diameter-- 147
Tip Clearance-- 4.25

There were more than twenty blades in rotor and no of elements for each blade was very large which would take too much computational time and cost. To overcome this our model has been restricted to single blade and computational analysis has been done on single blade model.
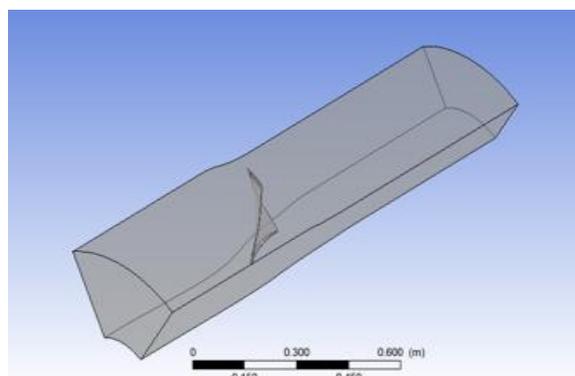

Figure2 single blade model with hub and casing

Modelling of blade geometry with suction could be done by creating one suction surface on the blade and creating suction hole at the Trailing edge of the blade which has beens shown in the next figure.
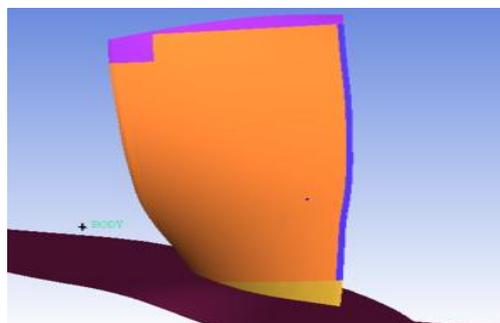

Figure3 rotor blade with suction surface and hole

2.2Mesh Creation and grid parameters
2.2.1Grid generation without Suction

Creation of mesh is very important in computational simulation of fluid flow. If mesh is not of good quality then this may affect our results. Two types of mesh has been used for our geometry one was Quad dominant and other one was volume tetra or mixed type mesh and these were obtained by shell meshing and patch dependent methods respectively.

When geometry has been imported from CATIA to ICEM CFD 14.5 then parts have been and each part was assigned a name. After creation of parts then geometry should be repaired by providing tolerance to 0.1 (the tolerance of 0.1 was given according to length of the smallest component in the model) and then apply and then delete surfaces have been selected and delete overlapping surface and any hole in geometry and once geometry has been repaired then this should be moved for meshing. Repairing of geometry was required beacuse in importing geometry from CATIA to ICEM some surfaces were overlapped or lost or some holes have been created in geometry and this would affect the meshing and if our meshing was not proper then we were not able to get result. After this body should be created by selecting two points diagonally.By doing this fine mesh could be created on the blade surface than the casing and outer walls.

For considering boundary layer effects on the flow height was kept fixed and number of prism layers were calculated using Y+ calculators.

Figure4parameters of part mesh setup

Once the global mesh parameters has been defined then part mesh setup needs to be defined which has been shown in the above fig. In the part mesh setup different element sizes has been provided through which coarsen mesh on the edge of blade should be possible. After defining all the parameters then create prism layer and then apply compute for generating meshing grids.

Figure5 meshing grids of blade and outer walls and casing

2.2.2 Grid generation with suction
In this geometry one new part named as suction layer was created and now follow all the steps which has been done previously to form mesh .
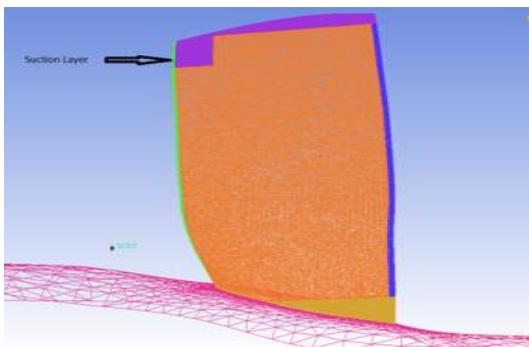

Figure6 meshing grid of blade with suction

2.3 Boundary Conditions for CFD analyis
Computational Fluid dynamics could be used to analyze and solvethe model flow assosciated with the fluid by usning numerical methods.

Different softwares are available today for CFD analysis in this study ANSYS-CFX is used for this.
2.3.1 CFX-pre setup for Model without suction surface
The settings for CFX- Pre used were based on the units of the mesh imported.
The parameters for the mesh units in mm were:-
➤ 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

2.3.2 CFX-pre setup for Model with suction surface

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 parameters for the problem applied were:-
➢ 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

2.3.3 CFX solver manager

The output file generated in CFX-pre was the input file for this part and this uses graphical interface method for CFD calculations. Double precision was selected because it gives more accurate numerical solution but when this method was used computer memory and grid size was also doubled when this solver execute the file.

2.3.4 CFX-post Processor

The output of the simulation or the result file was then loaded in this where the flow can be analyzed. 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.

III. Results and Discussion

Above defined models were analyzed in CFX-post processor. First model with no boundary layer control method was analyzed and results were obtained as velocity vectors at various span locations for two models with changed rpm and length units.
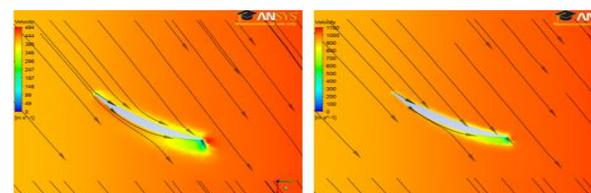

(Length units in mm)   (Length units in cm)

Figure7 velocity vectors at 25% span

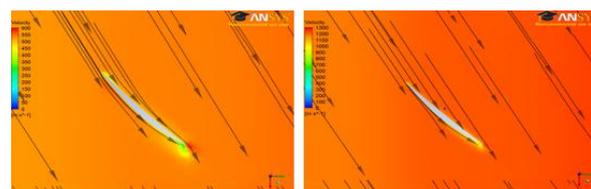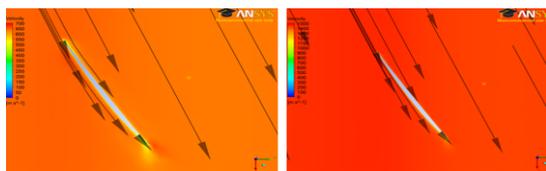(Length units in mm)   (Length units in cm)

Figure8 velocity vectors at 45% span

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

Figure9 velocity vectors at 85% span

The above three fig. shows that how the flow separation occurs when there was no suction surface present in the model and this was analyzed for model having two different length units but the results were similar. At the point where flow separation occurredcolor changed from orange to green and among three figures boundary layer effects could be seen as dominant in the fig. 7 near quarter of span where some strong vortex are also forming.

Due to some computational error and single blade model consideration and its range confined to 60 degrees pressure ratio comes nearly about 1.6.

According to the results obtained here we need to develop some steps to control boundary layer separation and to achieve this suction method was selected here.

In the suction method a suction hole was created at the trailing edge of the suction surface and to obtain results two mass flow rates (1 kg/s and 1.5 kg/s) has been used.
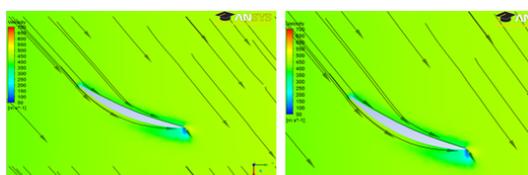
(Suction mass flow rate of 1.5 Kg/s)   (Suction mass flow rate of 1Kg/s)

Figure10 suction method velocity vectors at 25% span
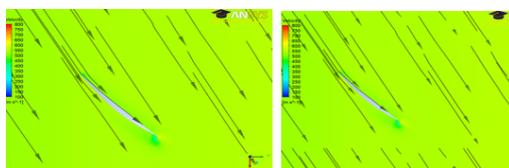
(Suction mass flow rate of 1.5 Kg/s)   (Suction mass flow rate of 1Kg/s)

Figure11 suction method velocity vectors at 45% span
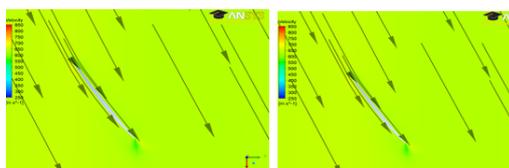
(Suction mass flow rate of 1.5 Kg/s)   (Suction mass flow rate of 1Kg/s)

Figure12 suction method velocity vectors at 85% span

Above three fig. were showing the results of flow over blade using suction surface for two different mass flow rates.

IV. CONCLUSION

From the results obtained for flow with and without suction method, it was clear that flow separation has been delayed by the use of suction surface. Boundary layer effects were dominant near quarter chord span of blade and some strong vortex were also seen when suction surface has not been used but when this control method has been used it would reenergize the low energy fluid to delay separation and also very weak vortex were observed and as a result pressure ratio was observed as 1.652 and increment in pressure ratio has been found to be as 0.052.

ACKNOWLEDGEMENTS

REFERENCES

Journal Papers:
[1]. 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, 2(1),* 2008.
[2]. Abate, Giada, Aerodynamic optimization of transonic axial compressor rotor, *Journal of Propulsion and Power,* 54(5), 2003, 41-55.
[3]. Benini and Ernesto, Three dimensional multi objective design optimization of a transonic Aerodynamic optimization of transonic compressor rotor, *International Journal of Gas Turbine, Propulsion and Power System,* 54(5), 2004.
Theses:
[4]. A. Sarkar, B. Singla and N. Kumar, *CFD analysis of Rotor 67 Blades,* University of Petroleum and Energy Studies, Dehradun, India, 2015.
Proceeding Papers:
[5]. Navneet Kumar, CFD analysis of rotor blade with and without boundary layer control technique, *proc. 6th International and 43rd National Conf. on Fluid Mechanics and Fluid Power,* Allahabad, India, 2016.