

Comparison of the Pressure among the Rotor Blade of the Compressor with and without Circumferential Case Grooving

Abinaya R

Assit Proff. , Department of Aeronautical Engineering
Bannari Amman Institute of Technology
(Affiliated to Anna University)
Erode, India

Gowshik S

Department of Aeronautical Engineering
Bannari Amman Institute of Technology
(Affiliated to Anna University)
Erode, India

Deepak D

Department of Aeronautical Engineering
Bannari Amman Institute of Technology
(Affiliated to Anna University)
Erode, India

Mr. Jawahar M

Department of Aeronautical Engineering
Bannari Amman Institute of Technology
(Affiliated to Anna University)
Erode, India

Abstract:- The article describes the analysis of the circumferential groove casing treatment in an axial compressor. To analyze the Pressure across the rotor blade of the axial compressor with and without circumferential case grooving, the control volume near the casing is divided into two parts: the control volumes inside and outside the tip gap. The flow mechanisms of stall margin improvement due to casing grooves in NASA Rotor 67 are quantitatively analyzed. Based on the axial distribution of the axial forces, the contribution of groove to the stall margin improvement is assessed. And the grooves that play a major role in stall margin improvement are ascertained. The pressure across the blade with case grooving is little bit higher than the pressure across the blade without case grooving.

Keywords:- NASA Rotor 67, Circumferential Groove Casing, Pressure Analysis, Axial Compressor.

I. INTRODUCTION

The aim and objective of the project is to determine the Pressure ratio across the blade where the shroud surface has been modified with circumferential groove casing. We all knew that present day researchers are trying to increase the efficiency of the compressor. In this project initially we are going to do analysis of the axial compressor having NASA67 rotor. Then using Unigraphics the required groove casing has been meshed using ICEM. As a result, we are going to compare the pressure ratio across the rotor blade without groove casing and pressure ratio across the blade with groove casing.

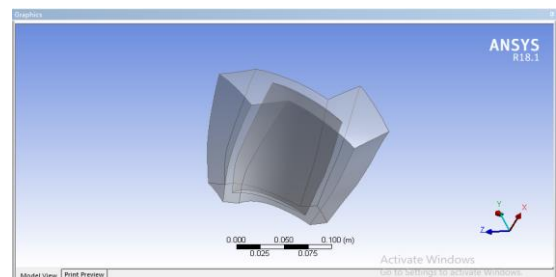


Fig.1. Blade domain

II. COMPRESSOR DESIGN AND ITS SPECIFICATIONS

- A. **CATIA V5:-** For our project we have considered the NASA Rotor 67 for the analysis purpose. We have imported the blade coordinates in the CATIA V5. Totally we have fifteen section of coordinates like 1b, 2b, 3b 15b. after importing every sections we have extruded the blade design which is the required design for the analysis.

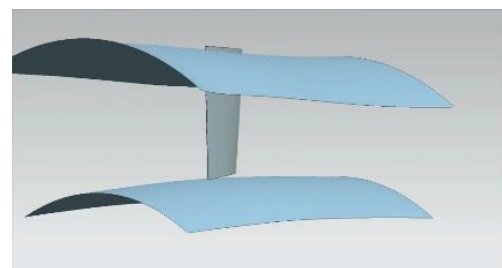


Fig.2. Blade design in CATIA V5

B. UNIGRAPHICS NX:-

After the completion of the design we have imported the blade to the Unigraphics NX to create the periodic surfaces across the blade. Here there is a small clearance between the blade and the hub, the clearance has been rectified by opting the tip clearance option. An offset has to be done along the blade tip touching shroud region. After making tip clearance and offset alternations we have created the periodic surfaces around the blade, the periodic surface includes the curve generation between the shroud regions preferably at the middle then the periodic surface is rotated in a circumferential direction according to the blade pitch. After creating the revolved body we have trimmed the periodic region according to the blade.

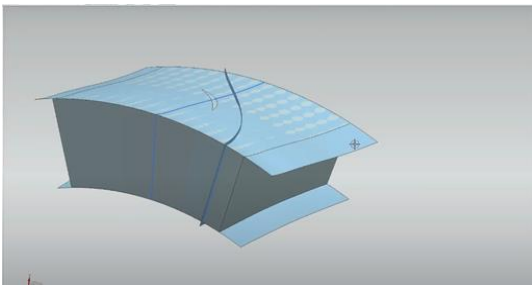


Fig.3. Periodic surfaces around the blade.

C. ANSYS 18.1:-

- After the completion of the required design we have imported the geometry along the ZX axis. In the next step the solid body has been selected and we have created the pattern of about 21 blades in a circular formation about ZX axis
- Since the generated bodies are still considered as an individual body we have unite them as single body by opting the Boolean command, the Boolean formation is very important for the upcoming command because sketches among the top and the bottom surfaces has to be generated. After uniting the 22 bodies using the Boolean command then we have selected the XZ plane to create the sketches like (Shroud, Hub, Inlet and the Outlet) of the blade.
- Face split option has been used to create a required flow path for the fluid or air flow. In the Face split 1 we have selected the two surface where the sketch 2 is the target body and the upper surface of the shroud is the other surface. In the Face split 2 command we have selected the sketch 2 surface as the target surface and the other surface is considered as the hub surface of the body.
- While defining the flow path we have selected the individual sketch planes to draw the required line and spline geometry. The lines were used to define the Inlet and the outlet, whereas the splines are used to define the shroud and the hub surface of the body
- The export points are the most important part of the analysis, the export points are created by selecting the flow path of the blade and both Front & Back surfaces of the blade.

- By creating the export points we can able to export the blade CAD geometry files. In the below diagram the green lines indicates the flow path of the air through the blade domain.

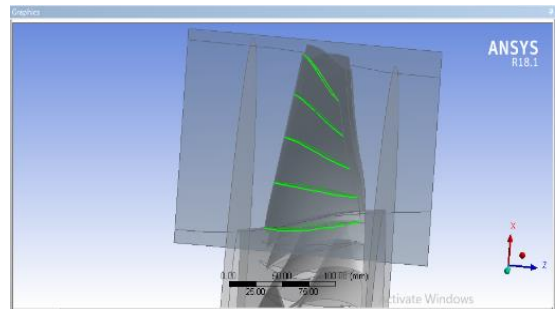


Fig.4. Flow path over the blade

D. ANSYS TURBOMESH:-

After the successful completion of the commands like pattern, Boolean, Facesplit 1 & 2, Flow path and Export points now we have to work in the Turbo grid to get the required mesh conditions along the regions of Blade, Shroud, Hub, Inlet and the Outlet. The below given figure shows the meshed surface.

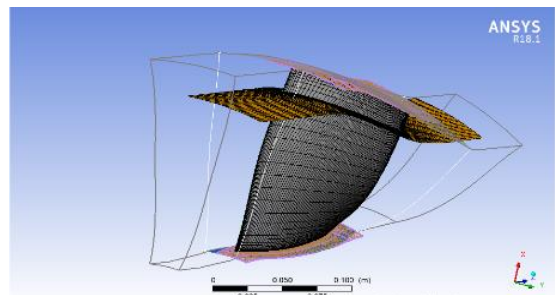


Fig.5. Meshed region of the blade domain.

E. ANSYS CFX SOLVER:-

- The meshed geometry now has been transferred to the CFX solver. Here we have to give the respective Conditions like component definition where the compressor is considered as the rotational part and the revolution value is given as -16000 revolutions per min and the expression's like pressure ratio, temperature ratio etc.. across the blade sn frame.

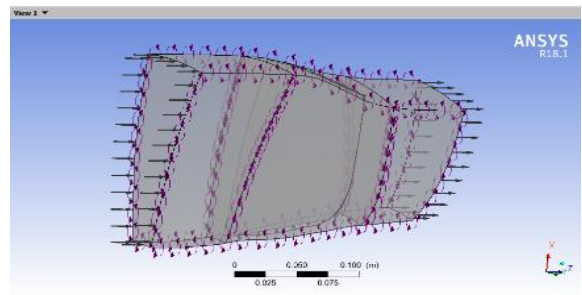


Fig.6. Turbo grid domain

III. CIRCUMFERENTIAL GROOVE CASING SPECIFICATIONS:-

A. UNIGRAPHICS NX:-

The Unigraphics NX software has been used to design the required Groove design above the shroud surface of the compressor. To create the required geometry we have selected the shroud curve so that we can able to draw the required sketch by keeping it as the reference. Then the groove geometry has been saved and exported as the para solid part so that it can be imported in the ANSYS ICEM.

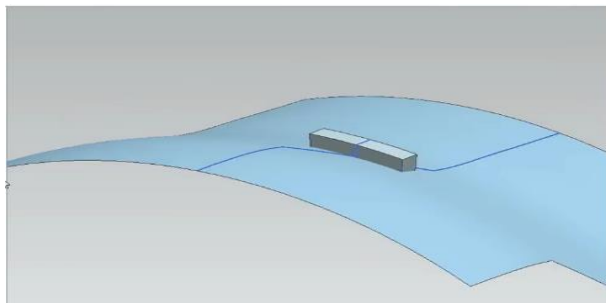


Fig.7. Groove casing

B. ANSYS ICEM:-

- ICEM is used to generate the mesh along the groove surface so that we can able to get the accurate flow and results across the groove. Then we have to save the file in ICEM CFD format so that we can import them in the turbo grid.

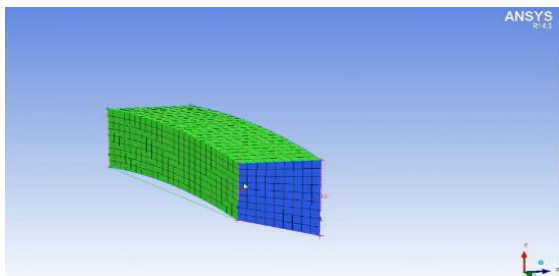


Fig.8. Meshing of Groove in ICEM

C. ANSYS CFX SOLVER:-

- After completing the meshing in the ICEM, we have imported the domain of the NASA Rotor 67, since we are directly importing the file we have given the specifications in the turbo mode like axial compressor and the body should be rotational. We have given the revolution as -16000 revolutions per mins.

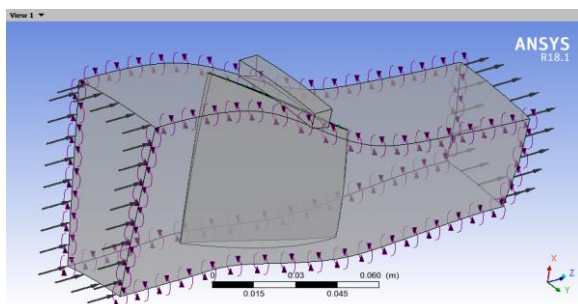


Fig.9. Groove with Blade Domain

D. GROOVE WALL DEFINITION:-

In the groove wall definition we have to set the conditions as wall. Then we have given the new domain interface as the groove interface, in that the Interface has to be given as groove surface along with the PERI 1 2 and the PERI 2 2. The type of domain interface model is rotational since it is a compressor.

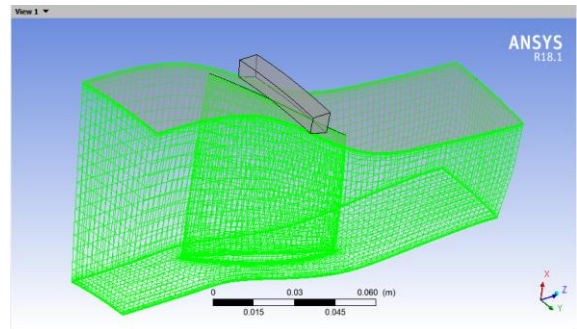


Fig.10. Groove wall definiton

E. REQUIRED MESH ON THE GROOVE:-

Though the Groove casing and the shroud surface are connected together but there is no mesh region created among those connected bodies so that we have to hide the other meshed regions of the Blade, hub, inlet, and outlet. The flow passage along the domain is also hided so that we can able to edit the mesh accurately. By clicking the edit meshing option along the shroud surface we have select the box select option, then we have to accurately select along the groove casing area. Then we have to select one more region along the same surface by opting the flood selection option we can merge the shroud surface and the groove's bottom surface by applying the changes we will get the required mesh over the geometry.

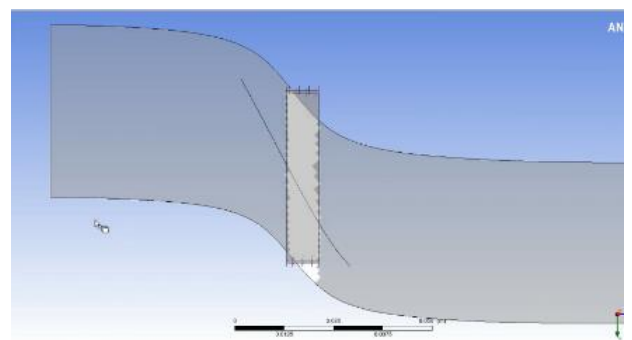


Fig.11. required mesh over the Groove

F. VELOCITY AND PRESSURE CONTOURS

From the previous analysis of the Rotor blade it was evident that the pressure across the blade tends to be increases where as the velocity tends to be increased at the inlet and slows down behind the blade. With the replacement of the compressor with the casing helps us to notice that there is an evidence of turbulence inside the groove.

V. RESULTS

A. PRESSURE ACROSS THE BLADE WITHOUT GROOVE CASING:-

From the pressure contour it is visible that the pressure is increased when the air passes the blade. In velocity contour at the entry of the inlet the velocity seems to be greater when the air passes the blade the velocity gets decreased which indirectly shows that there is an increase in pressure.

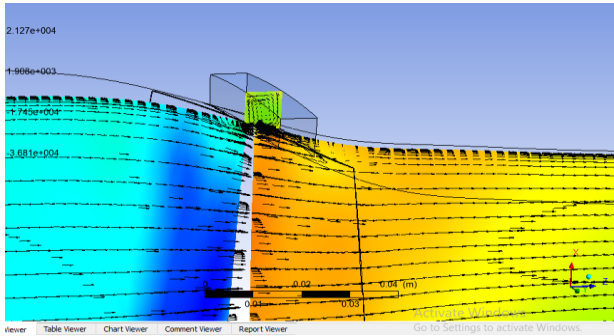


Fig.12. Pressure Contour

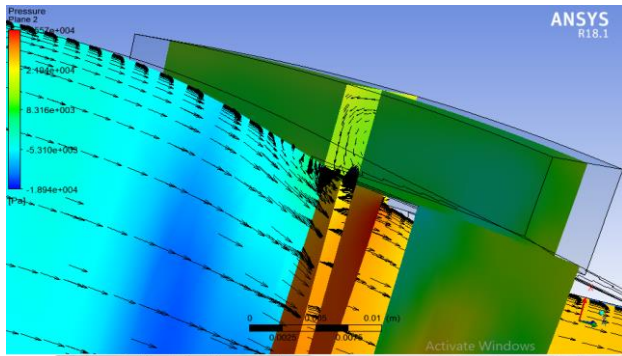


Fig.13. Pressure inside the Groove

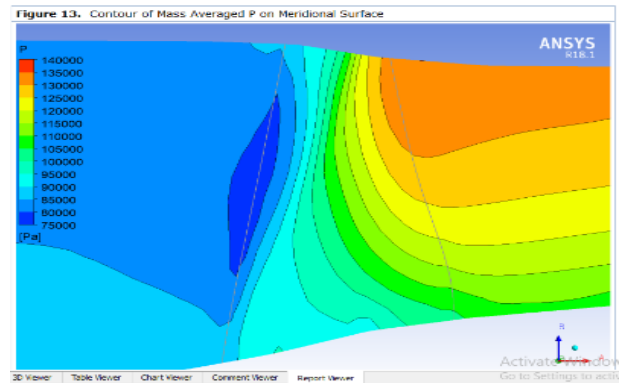


Fig.14. Pressure across the Blade without groove

B. PRESSURE ACROSS THE BLADE WITH CIRCUMFERENTIAL GROOVE CASING:-

It can be clearly seen that the pressure across the blade has been increased when the shroud region has been modified with the circumferential groove casing.

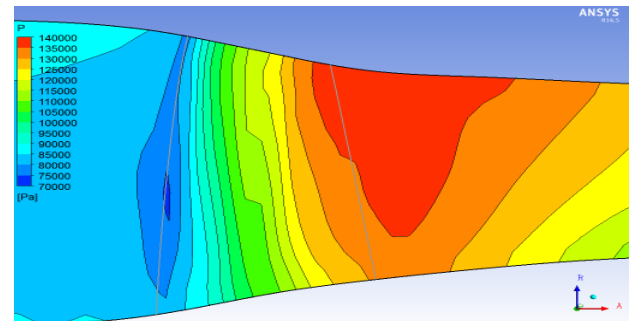


Fig.15. Pressure across the blade with circumferential groove.

IV. FIGURES AND TABLES

A. NASA ROTOR 67 SPECIFICATION:-

S. No	CHARACTERISTICS	VALUES
1	No of Blades	22
2	Design Rotational speed	-16,000 Rev/ Min
3	Tip Speed(m/s)	429
4	Inlet Tip Relative Mach Number	1.38
5	Design Mass Flow Rate(Kg/sec)	33.28
6	Design Pressure Ratio	1.66
7	Design Tip Clearance	1.01
8	Average aspect Ratio	1.56
9	Tip Solidity	1.29
10	Hub Solidity	3.1
11	Inlet Hub/Tip Ratio	0.37
12	Outlet Hub/Tip Ratio	0.47

VI. CONCLUSION

By comparing the [14] and [15] images we can able to evident that the pressure is higher across the blades where the shroud region is connected with the circumferential groove casing. The same images can also be compare the velocity across the blades groove casing, where there is a small turbulence area inside the groove which resulted in the higher pressure.

ACKNOWLEDGMENT

Thanks to Asst. prof Ms. Abinaya R for his devoted guidance and the work he has done –working in conjunction with Dr. Kalil Rahiman. The clarity of her comments in the designing and analysis of the Rotor Blade allowed me to complete the article with expected results.

REFERENCES

- [1]. Lakshya Kumar and Dilip Kumar, “Design and CFD Analysis of a Multi Stage Axial Flow Compressor,” (CSIR National Aerospace Laboratories), India, 2019
- [2]. Mohammed Sad aftab, Muhammed Aadil Khan, “CFD analysis of the five stage axial compressor”, (Department Of Aeronautics & Astronautics, Institute of space technology), Pakistan, 2018
- [3]. Haixin Chen, Xudong Huang, Mark ross, “Analysis of Axial Compressor with groove casing”, (Chinese Journal of Aeronautics), China .