International Journal of Engineering Research and Development
e-ISSN: 2278-067X, p-ISSN: 2278-800X, www.ijerd.com
Volume 13, Issue 4 (April 2017), PP.22-26

Design and Development of Transonic Axial Flow Compressor Rotor Blade

K Balaji[1], S Emilon Zeon Rajan[2], C Ganesan[3,] M Jananee[4], S Sanjay Kumar[5]
[1] Assistant Professor, [2,3,4,5] Students
Department of Aeronautical Engineering Karpagam Institute of Technology, Coimbatore.

Abstract:- *This paper is about a new computational fluid dynamics developed for the transonic flow in a compressor rotor. Due to 3-Dimensional blade modification the arrangements satisfying the required boundary condition. Engine compressor towards distorted inflow has to be taken in account which is already in the design phase. Flow separation over the blade surface reduction and elimination can improve better aerodynamic, performance, efficiency and stall margin. NASA transonic rotor tip critical in baseline blade rotor performance energizing the low momentum boundary layer, controlling the inception of stall. A Profile generator are attached on the inner casing of the rotor ahead to the loading edge of the rotor and it is influenced on the overall performance which has been studied.*

Keywords: *Compressor rotor blade, Numerical investigation, Flow separation, Vortex generators, Stall margin, Efficiency.*

I. INTRODUCTION

Rotating blades are usually attached with hub of the compressor. The blades are tilled with a correct series with coincidence. This shape is full fill in NASA 0037. Tip span are usually aligned and attached in starting from rotating stall. A jet engine of high pressure compressor to investigation the effect of inlet total pressure distortions on rotating stall. It will be occurred in spike type stall. A low momentum area near rotor tip leading edge causes the flow spillage and leads to stall inception. Turbo machinery and transonic axial flow compressors have played important role to attain higher pressure ratio at every single stage of compressor. A high level of aerodynamic performance must be maintained over a wide range of mass flow rate and speed. The successful operation and reliability of the axial flow compressor is greatly affected by its stability margin. In a compressor specific speed there is a limit for both higher and lower mass flow which is called its operating range, the operating range is between the chock limit and stall point. Stall margin of the compressor can be increased by using various passive and active control techniques.

II. LITERATURE REVIEW

[1] KAMARUL A.A predicted the reduction of flow separation over blade surface can improve better aerodynamic performance. The design cases studied comprised of compressor operation at 80%, 90%, 105% on design rotational speed. 3D flow visualization and performance parameters were detailed to study the relative blade, Mach number distribution created by the model**.**

[2] AVINASH KUMAR. Found the boundary layer separation and tip leakage flow between wall and the blade in axial flow compressor. The modeling effect is investigated to vortex generators in transonic axial flow in compressor stage. The rotor blade performing to energizing the low momentum boundary layer and controlling the stall. The profile will generate the vortex will be retrofitted.

[3] JIAYE GAN tested jet engine high pressure compressor rotor to investigate the effect of inlet total pressure distortion on rotating stall which usually starts from tip span**.**

III. METHODOLOGY

In this work to study the influence of different profiles on airflow in compressor rotor blade using CATIA and ANSYS fluent for strength following procedure is adopted 1) Modeling and Mesh generation.2) Flow analysis.

3.1Modeling and Extraction of flow domain

The data points for the rotor model and the computational domain were collected. The 3D model of the rotor blades are designed by using CATIA. Figure 1 show the circular jets are decided to place on the 3D model of the rotor blade. Figure 2 shows the 3D model of the rotor blade along with the flow domain in the entire compressor. The values for the generated mesh near the wall zones are maintained at too which is as per the turbulence modeling requirements. A single passage approach is modeled and numerically solved with the assumption that flow is periodic within the passage.

3.2Mesh Generation

Meshing of the computational domain is made using CFD as shown in Figure 3.Thus, the discretisation of the Hybrid structured grid elements were generated.

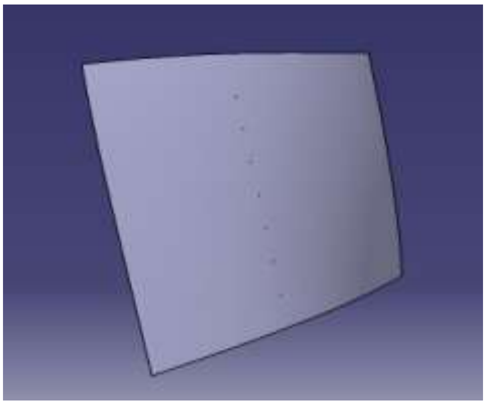


Figure1. Rotor Blade with Circular Jet Enclosure

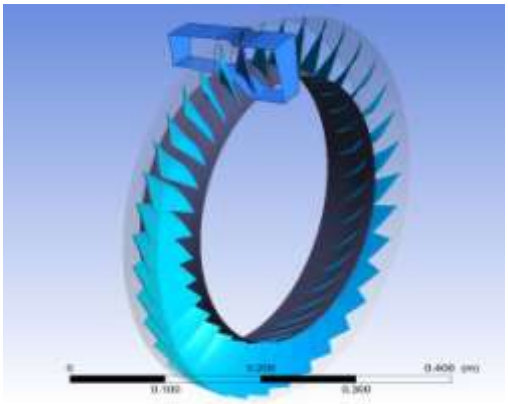


Figure 2. Wheel Rotor Assembly

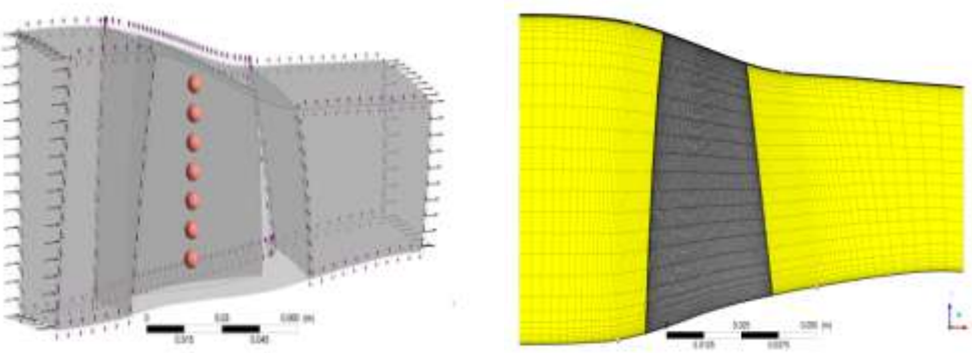


Figure 3. Mesh Rotor Blade

III. FLOW ANALYSIS

The flow analyses were carried out using a commercial CFD package. The flow fluid was assumed to be an ideal gas, Shear Stress Transport (SST) turbulence model was equipped for the present study. To conform that results are independent of the mesh, grid independency studies were carried out and total 1,800,000 cells were created in the final flow domain. The wall boundaries of the blade, rotor and stator, hub and casing are specified with no slip condition. It indicates that the fluid sticks to the wall and moves with the same velocity as the wall. The design speed of the rotor (1800 rad/s).The total pressure and total temperature were imposed at the inlet (p_{inlet} = 101330 pa, T_{inlet} = 298 k) to recreate the experimental boundary conditions .As an outlet condition, To find the Design mass flow and chocked mass flow conditions at on design speed. Figure 4.shows the boundary conditions applied to the flow domain.

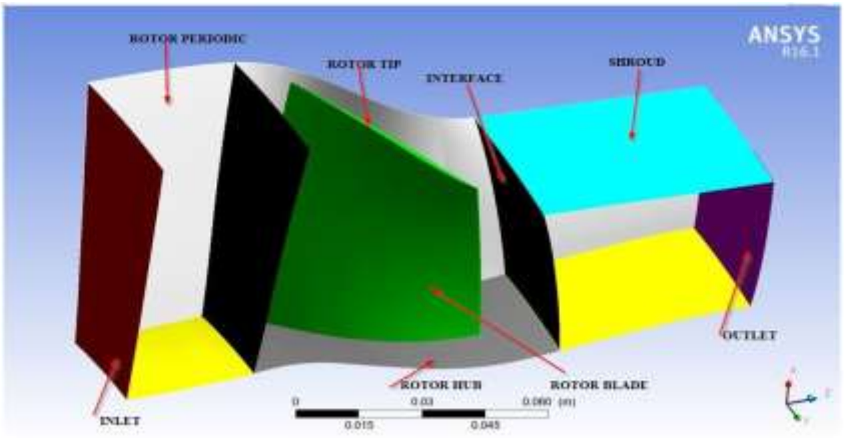


Figure 4. Boundary Conditions Applied to the Flow Domain

IV. RESULTS AND DISCUSSION

The comparative performance of the baseline compressor and injection compressor at on- design speed and off-design speed. It is monitored that among the off/on design conditions, the flow follows the consistent patterns and the highest efficiency, stall margin of the compressor stage is improved by the compressor design speed. The flow physics of the compressor rotor for different design conditions were studied using velocity vectors, Mach number contours and velocity streamlines as mentioned below.

Mach number

Figure 5 shows the comparative Mach number contours computed along the on/off design speed conditions. Concerning the calculated flow field, the plots were computed at span blade to blade Mach number. It is observed that a strong shock front was created and flow separation was combined along the flow at the suction side of the rotor blade at design speed. While in the formation of shock and flow separation region was much weaker compared to previous flows.

In design speed, the flow was much smoother and flow separation was almost unnoticed. The reduction in shock strength is seen to directly affected the pressure ratio reduction .which disturbs the efficiency of the compressor rotor blade. It also indicates that the blade boundary layer and its interaction have been properly captured.

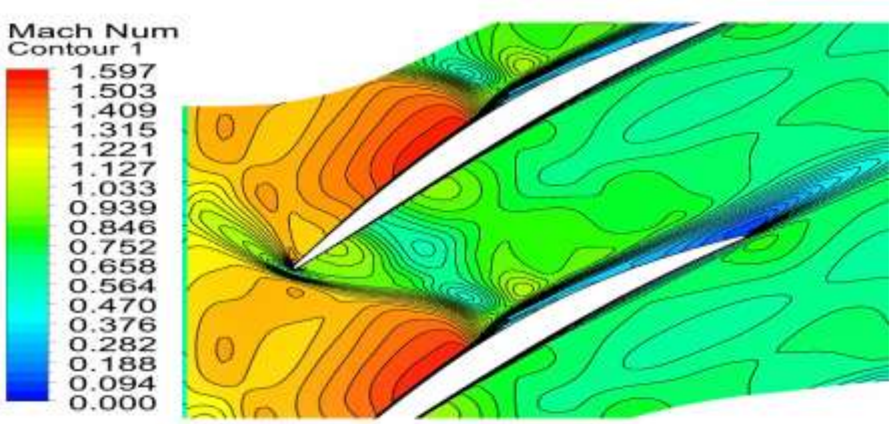
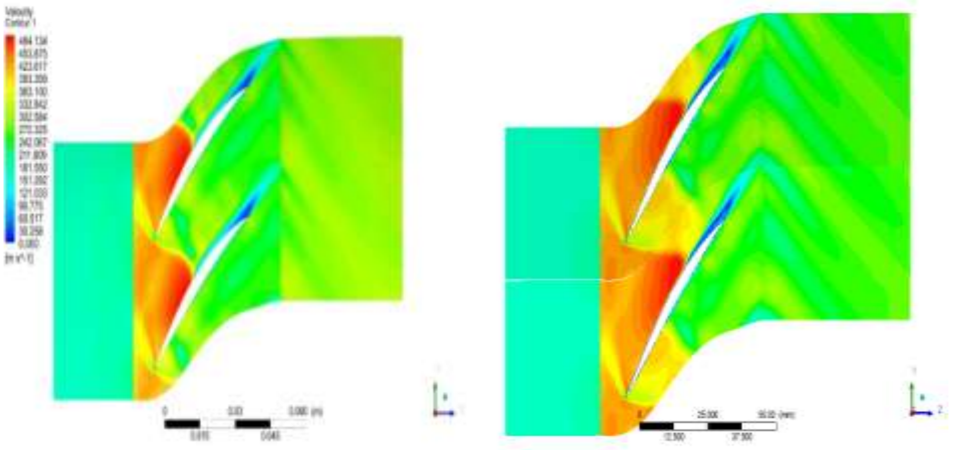


Figure 5. Comparative Mach Number Contours at Various Design Conditions

Velocity

Figure 6 displays the comparative velocity vector plots for peak efficiency condition at baseline rotor's on/off design compressor speed. Vector plots are plotted at the span of blade to blade view for the velocity distribution over the operating conditions. As shown in the contour, high velocity is observed at the inlet condition which produces shock front where it gets reduced from the blade surface at trailing edge because of the flow separation. Though there is a difficulty to produce the well-defined tip clearance vortex pattern, its shock interaction with the flow is captured reasonably well.

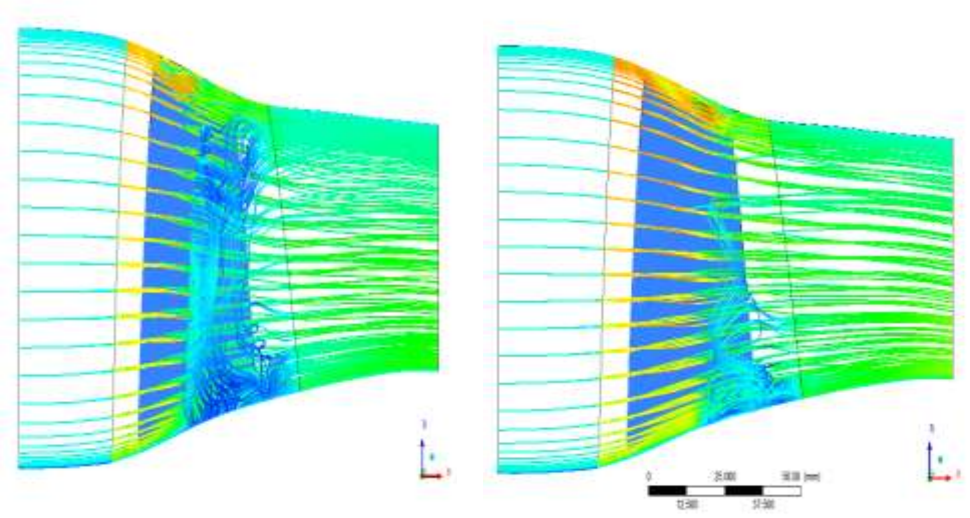


(a) (b)

Figure 6. Comparative Velocity Contours at Various Design Conditions

Streamline

Figure 7 shows the comparative detailed streamline patterns in suction side of the rotor for various design speed at maximum efficiency condition. The streamlines discharged from the leading edge and the separation lines are created at the suction side boundary. The 3D streamline patterns indicate fairly well the internal flow conditions created at suction of the rotor span. Forms of flow disruption were created because of the shock and the flow separation near the trailing edge which were also captured in the blade to blade Mach number contour as seen in Figure 5. Also for the off-design condition at maximum efficiency state, clearly reveals that there is less flow separation created compared with the design speed.



(a) **(b)**

Figure 7. Comparative Streamlines on the Maximum Efficiency Rotor State at Various Design Conditions

V CONCLUSION

A computational flow steady analysis for a transonic axial flow compressor rotor at on-design speed (1800 rad/s) and various design conditions was carried out. The flow pattern analysis for the Mach number distribution, streamline patterns and Velocity Vector Contours to provide a detailed conclusion. The purpose of this paper was to study the flow and performance parameters for better understanding over the design condition.It was found that the efficiency of the rotor has a direct impact from the shock front created. Thus, there is a reduction in efficiency due to the poorly produced shock front in design speed along with a big pressure ratio difference. Even though the velocity condition at design speed looks better than on-design speed (1800 rad/s), due to unstable flow condition along various mass flow rate, reduced efficiency was noted which negatively impacts the efficiency.

ACKNOWLEDGMENT

With great pleasure and deep gratitude, authors wish and express their sincere gratitude to beloved Principal **Dr. T. Ramachandran** for providing an opportunity and necessary facilities in carrying out this work and express sincere thanks to all the staff members of Aeronautical Engineering whose assistance played a big role in this work and have been of immense value.

REFERENCES

[1] Moore Royce, D., & Reid, L. (1982). Performance of Single stage Axial Transonic Compressor with rotor and stator), aspect ratios of 1.19 and 1.26 respectively and with design pressure ratio of 1.82. NASA Technical Paper, (November 1978), 105.]

[2] Chen, G. T., Greitzer, E. M., Tan, C. S., & Marble, F. E. (1991). Similarity Analysis of Compressor Tip Clearance Flow Structure. Journal of Turbo machinery, 113(2), 260.

[3] Freeman, C., & A., C. N. (1992). Method for the prediction of supersonic compressor blade performance. Journal of Propulsion and Power, 8(1), 199–208.

[4] Law, C. H., & Wadia, A. R. (1993). Low Aspect Ratio Transonic Rotors: Part 1—Baseline Design and performance. Journal of Turbo machinery, 115(2), 218–225.

[5] Suder, K. L., Chima, R. V., Strazisar, a. J., & Roberts, W. B. (1995). The Effect of Adding Roughness and Thickness to a Transonic Axial Compressor Rotor. Journal of Turbomachinery,117(4), 491.

[6] Suder,K.L.(1997).Blockage Development in a Transonic, Axial Compressor Rotor.

[7] Calvert,W. J., Emmerson,P. R., & Moore, J. M. (2003).Design, Test and Analysis of a High-Pressure-Ratio Transonic Fan. In Volume 6: Turbo Expo 2003, Parts A and B (pp. 417–427). ASME. http://doi.org/10.1115/GT2003-38302.

[8] Benini,E.,Biollo,R, & Ponza, R. (2011).Efficiency enhancement in transonic compressor rotor blades using synthetic Jets: A numerical investigation. Applied Energy, 88(3), 953–962.

[9] Biollo,R, & Benini, E. (2013). Recent advances in transonic axial compressor aerodynamics. Progress in Aerospace Sciences, 56, 1–18. http://doi.org/10.1016/j.paerosci.2012.05.002.