International Journal of Research in Engineering and Science (IJRES)
ISSN (Online): 2320-9364, ISSN (Print): 2320-9356
www.ijres.org Volume 9 Issue 5 ǁ 2021 ǁ PP. 38-44

Investigation on the Effect of Holes in the Axial Compressor Blades

Abinaya.R[1], Charan Sibi.S[2], Ramkumar.P[3], Shri Arravindar Ayyasamy.L[4]

*[*1]Assistant Professor, Department of Aeronautical Engineering, Bannari Amman Institute of Technology, Sathyamangalam.*
[2]UG scholar, Department of Aeronautical Engineering, Bannari Amman Institute of Technology, Sathyamangalam.
[3]UG scholar, Department of Aeronautical Engineering, Bannari Amman Institute of Technology, Sathyamangalam.
[4]UG scholar, Department of Aeronautical Engineering, Bannari Amman Institute of Technology, Sathyamangalam.

Abstract:
*Compressors deliver the determined mass flow at a specific pressure ratio to the combustion chamber. Axial compressors are mainly used in subsonic/transonic commercial aircrafts. Axial compressors have stages which include a rotor and stator. In this paper the effect of holes incorporated in the rotor blades are analysed for the performance of pressure ratio. To verify the preliminary design modelling is done in Bladegen, Meshing in Turbogrid and flow is simulated in CFX flow solver ANSYS 18.1. In this study, total pressure at inlet and static pressure at outlet is used. The blades with holes are designed in CATIA V5. The blade without holes are analysed in TURBOGRID and post processing in CFX solver is done and with holes are analysed in FLUENT model for post processing. The same boundary conditions are applied for more accurate results. The compressor blades with holes are anticipated to help in increasing the pressure ratio. The rotor blades with holes are designed in catia and analysed in fluent. The holes have the size diameter 1.5mm and 6*10 patterns along the leading edge of the blade. The results are compared and are discussed..*

Keywords: *Compressor Blade, Bladegen, CFX, Pressure Analysis ,Blade with holes*

Date of Submission: 27-04-2021 Date of acceptance: 11-05-2021

I. INTRODUCTION

The main motive of the project is to incorporate holes in the axial compressor blade and check its pressure characteristics and find the effectiveness of the holes with the pressure ratio. The comparison is done between the normal conventional compressor and the compressor rotor blade with holes.The aim and objective of the project is to determine the Pressure ratio across the blade with holes incorporated in them along the leading edge of the compressor blade and reducing the weight of the blade.Today researchers are trying to increase the efficiency of the compressor. In this project we have modified the blade of the compressor by incorporating holes in it and checking its pressure characteristics.We have designed it using ANSYS BLADEGEN tools which is majorly used in designing the turbomachinery parts and analysed in TURBOGRID and CFX.The holed blades are designed in CATIA V5 and then imported to Ansys ICEM for meshing and checked in FLUENT. As a result, we are going to compare the pressure ratio across the conventional compressor without hotels with compressor rotor blade holes.

II. DESIGN AND ITS SPECIFICATION

2.1 INITIAL CONDITION

We have used bladegen for the designing of the compressor rotor blade and stator blade. The conditions are given in the initial input data and blades are created in bladegen with the condition. We have used the same for rotor stage and stator stage.

2.1.1 BLADEGEN DESIGN

After the initial conditions are given the blade gen creates the blade for a normal axial compressor and we have to create for both the rotor blade and stator blade

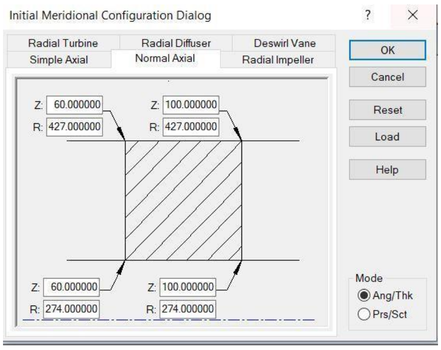

Fig 1.1 Initial condition

2.2 BLADEGEN DESIGN

After the initial conditions are given the blade gen creates the blade for a normal axial compressor and we have to create for both the rotor blade and stator blade.The modelling is done in Bladegen and meshing in Turbogrid and simulation in CFX with Ansys 18.1. The industrially accessible programming bundle ANSYS 18.1 gives BladeGen module which is utilized for the 3D design part of blade for either new or existing blade. The blade angles at hub and shroud should be maintained to keep the required shape. The calculated values of the compressor are inserted in the dialog box shown in fig 1.1.The blade gen provides two various modes of operation one is Angle/Thickness (Ang/Thk) mode and the other is Pressure Side/Suction Side (Prs/Sct). The former is used for compressors and the latter one for turbines.

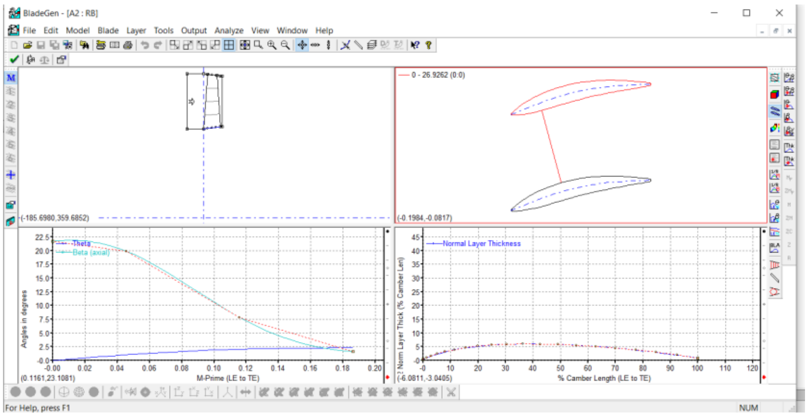

Fig 1.2 Bladegen design

2.3 MESHING

A Mesh is a network that is formed of cells and points. Meshing is a part of the engineering simulation manner where complex geometries and models are divided into simple elements that can be used as discrete local approximations of the larger domain. The Mesh can be resembled into any shape and size depending upon the geometry and is used to resolve Partial Differential Equations.Proper meshing is an important step to solve the fluid domain. The meshing of the bladegen model is done with the Turbogrid of Ansys 18.1. Turbogrid is a powerful tool that provides high quality mesh for turbomachinery products while preserving the geometry.While discretizing the model proper topology is important and in this model mesh ATM type is used for the flow passage to connect with the passage easily. The same is being created for the stator blade. The mesh was created with tip clearance for the rotor blade. Hexahedral mesh element is used for the entire volume of the blade passage. The 3D mesh is shown in fig 1. After doing the grid independency the mesh element of about 300k is sufficient for pressure ratio and efficiency.

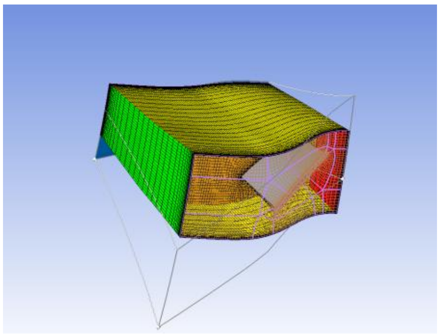

Fig 1.3 Meshing of compressor

2.4 NUMERICAL MODELLING

The conventional part of the compressor analysis, the numerical modelling is done in the CFX solver of the ANSYS18.1 package. Here problem includes the compressor blade passage 1 and passage 2 to attain solution to this problem , the equations to be solved are mass, momentum, turbulence and energy. The same condition is used for both the passage 1 and 2.

2.4.1 Boundary Conditions and Solver Selection

Total pressure at R1 Inlet : 1 bar
Static pressure at S2 Outlet : 1.6 bar
Rotational speed: 8933 rev/min
Air : ideal gas
Turbulence Model : SST
No of nodes : 445885
No of elements : 1952608
We have created the two domains one for rotating blade and other stationary blade. There are two turbulence models in CFX solver : k-ε and Shear Stress Transport (SST). The k- ε model is employed for low Reynolds number flow. The SST model is a combination of k-ω models and k- ε models. In this model the Reynolds number is high so it is used. To solve the problem the total temperature and pressure at inlet and outlet is given. Different flow domains are defined and shown in fig 1.4 and boundary condition in fig 1.5 .For boundary conditions we have applied a turbo mode tool which is unique in cfx solver setup. The turbo mode tool is one which sets up its boundary condition according to our conditions and is very much useful in time saving.

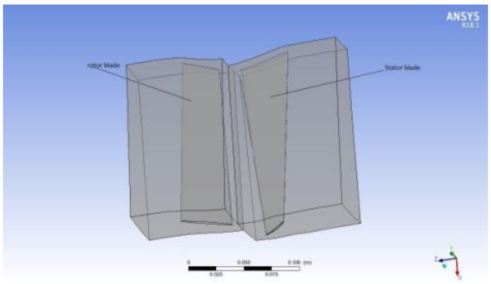

Fig 1.4 Flow domain of whole stage

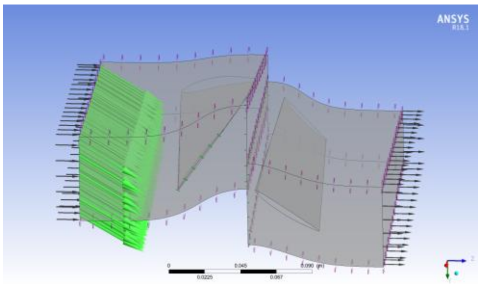

Fig 1.5 Boundary condition

2.5 CATIA V5

We have used Catia v5 version for incorporating holes in our compressor blades and the blade which is chosen is the same blade that we have created in Bladegen software. After importing the iges file from the bladegen the blade is extracted from the file and holes are incorporated in the blades as shown in fig 1.6. The holes which we have incorporated have the diameter of about 1.5 mm in diameter and has a pattern of 6*10 and of the total 60 holes are imposed in the blades. These blades are through blades which are through top to bottom.

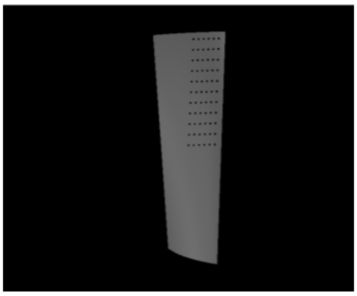

Fig 1.6 Rotor blade with holes

2.5.1 DOMAIN CREATION

As we have already known that solving the modified compressor blade cannot be done in the Turbogrid model since it solves only the 2D model of the blade we have to create a new domain (shown in fig 1.7) for our blade so it is created separately and then meshed in ICEM for more accurate meshing.

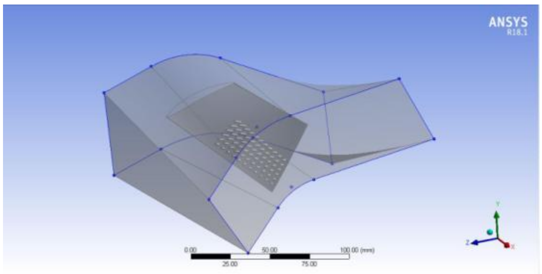

Fig 1.7 Domain for blade with holes

2.5.2 ICEM MESHING

ICEM is used to generate the mesh along the blade so that we are able to get the accurate flow and results across the domain. Then our file is imported into Fluent solver for further analysis as shown in fig 1.8.

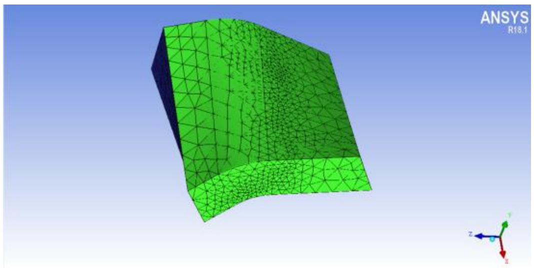

Fig 1.8 ICEM Mesh of blades with holes

2.5.3 NUMERICAL MODELLING:

The numerical modelling for the modified blade with holes is done in the Fluent Solver package in Ansys 18.1. Here the rotor blade is alone used for the analysis. The model is transferred and same boundary conditions are applied to validate the results with conventional compressor.

2.5.3.1 Boundary conditions

Total pressure at Inlet :1 bar
Static pressure at outlet : 1.6 bar
Air : ideal gas
Turbulence model : SST
No of nodes :469109
No of elements : 1675504

We have used the same boundary conditions and defined the k- ε model and standard reliable for the boundary conditions. The inlet and outlet are given static pressure inlet and pressure outlet to determine the results.

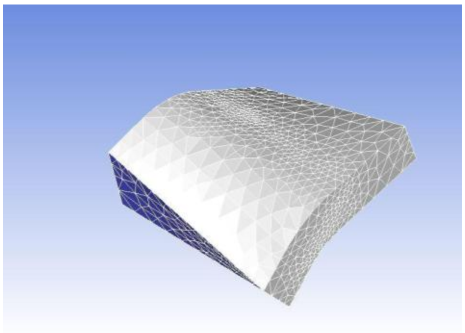

Fig 1.9 Boundary condition of blade with holes

III. EXPERIMENTAL WORKS

The blade is being designed in the BLADE GEN software in ansys 18.1. The conditions are given from the theoretical form and. The blade was created according to the theoretical data and used the NACA 4 series and modified the blade according to the data. The initial design is done by assuming constant mean diameter of the blade. The rotor blade consists of 45 blades and the stator blade consists of 49 blades. The no of blades are considered according to the pitch to chord ratio which should be 0.8 to 1.2. The blade with holes are done by CATIA V5 which is imported from bladegen and modified along it. The blade after modification, is imported to ANSYS and meshing is done in ICEM and post processing in FLUENT. After both the analysis is done the results are validated with theoretical data and compared blades without holes and with holes is carried on.

IV. RESULTS AND DISCUSSION

4.1 CONVENTIONAL COMPRESSOR

The results are taken and comparison is done with the results obtained from the conventional model and blades modified model. The conventional compressor that we have analysed with each other. The pressure increases gradually from inlet to out and the results are discussed below. As we are comparing it with the blades modified with holes those results are also shown below.The fig 1.10,1.11,1.12 has shown that our compressor analysis of conventional blades has shown positive results and then it has shown an increase in the pressure from inlet to outlet.

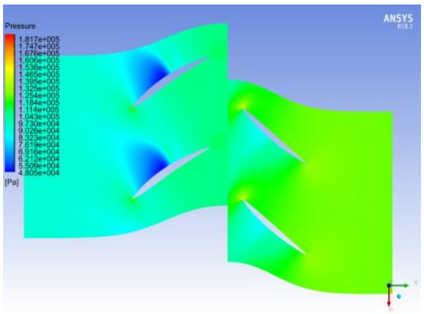

Fig 1.10 Pressure contour

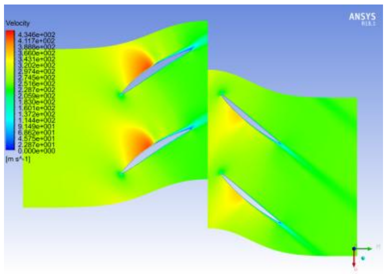

Fig 1.11 Velocity contour

The above fig 1.10,1.11,1.12 has shown that our compressor analysis of conventional blades has shown positive results and then it has shown an increase in the pressure from inlet to outlet.

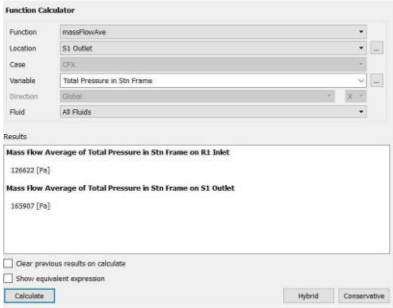

Fig 1.12 Pressure value of conventional

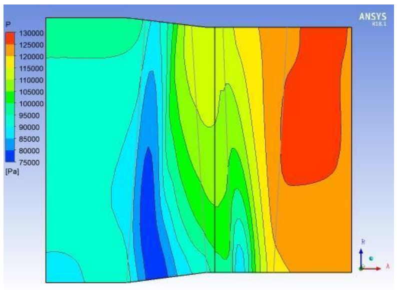

Fig 1.13 pressure compressor

4.2 BLADES WITH HOLES

The blades with holes are analysed in Fluent post solver and the results are shown below. The results show that the model with holes tends to decrease the pressure along the blade from inlet to outlet. There is pressure drop across the blade due to flow separation along the surface.

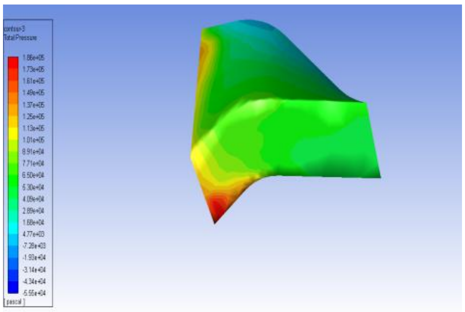

Fig 1.14 total pressure along blade with holes

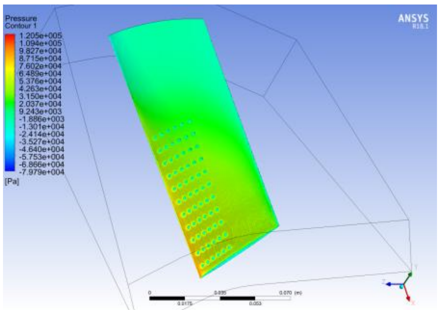

Fig 1.15 pressure along blade

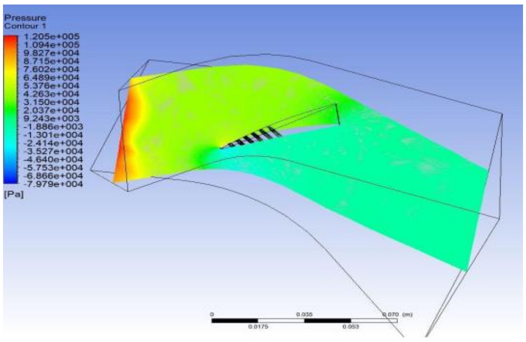

Fig 1.16 Pressure Contour Of Blade With Holes

V. CONCLUSION

In this project we have done the numerical analysis for the compressor blades with rotor blade holes and single stage axial flow compressor.The imbalance of flow between R1 inlet and S2 outlet was about negligible . The pressure characteristics is the main parameter that we are about to compare. As we can see that the conventional compressor has increased in pressure by about a pressure ratio of 1.3. The compressor rotor blade with holes did not increase the pressure as we have anticipated or expressed to. Thus we can conclude that the compressor blade with holes will not increase the pressure ratio or not have any influence in decreasing the velocity nor increasing the pressure. The pressure properties along the blade with holes is said to fluctuate with the change in shape of the blade and so the flow is getting disturbed and it did not behave as we expect.

REFERENCES

[1]. Lect.Iman Jabbar Ooda , prof.Dr.Arkan Kh.Al-Taie, 〝Axial Flow Compressor Blades Separation Control Using An Air Injection Holes〞, researchgate, vol. 331533190.

[2]. K. Bammer, R. Staude,"Optimization for Rotor Blades of Tandem Design for Axial Flow Compressors", Journal of Engineering for Power, vol. 102, 1980.

[3]. Jeetendra Kumara, Tarannum Mujawarb and Dr.S.A.Channiwala, 〝Aerodynamic Design and Numerical Analysis of Transonic Axial Flow Compressor〞, researchgate, vol. 335222171

[4]. M. S. Aftab, M. A. Khan, F. Ali and K. Parvez, "Design and analysis of a five stage axial flow compressor," 2017 Fifth International Conference on Aerospace Science & Engineering (ICASE), 2017, pp. 1-9, doi: 10.1109/ICASE.2017.8374248.

[5]. Mr Vamsi Krishna Chowduru, Mr A Sai Kumar et.al ,"Numerical Investigation of Fluid Flows over a Rotor-Stator(Stage) in an Axial Flow Compressor Stage",International Journal of Engineering Research and General Science Volume 3, Issue 4, July-August, 2015.

[6]. Ujjawal A. Jaiswal, Prof. S. J. Joshi "Design and Analysis of Stator, Rotor and Blades of the Axial flow Compressor ",International Journal Of Engineering Development And Research, IJEDR1301005, 2013.