ISSN(Online) : 2319 - 8753
ISSN (Print) : 2347 - 6710

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 3, March 2015

Computational Analysis of Compressor Blade

Sheik Ghouse .M, P. Manivannan

P.G Scholar, M.Tech (Aeronautical Engineering), Hindustan University, OMR, Padur, Chennai, Tamil Nadu, India.

Professor, Department of Aeronautical Engineering, Hindustan University, OMR, Padur, Chennai, Tamil Nadu, India.

ABSTRACT: This computational work aims at analyzing the flow behavior through a linear cascade of compressor blades with the help of Computational Fluid Dynamics using the FLUENT software. An attempt has been made to study the boundary layer behavior on the blades at various flow incidence angles and Reynolds numbers, based on flow inlet velocity and blade chord. A Computational Fluid Dynamics based computational model is created with the reference to an experimental work already carried out. This study will bring out the methodology for solving the problem using Computational Fluid Dynamics tools. Particularly, the capability of turbulence model to predict laminar, turbulent and transition in the boundary layer flows has been made. The geometric modeling and meshing of the flow domain has been carried out using the software's CATIA and GAMBIT respectively.Development of boundary layer, point of flow separation, flow reattachment and pressure coefficient over the blade surfaces have been found and compared with the experimental results.

KEYWORDS: Pressure coefficient, velocity effect over the blade

I. INTRODUCTION

Gas turbine engine manufacturers are constantly competing to produce more efficient , higher thrust and lighter engines. Correspondingly, the research and development of the turbo machinery components is directed for higher efficiency, higher power output and less weight. To achieve these goals, improvement in compressor blade design is essential. Generating qualified blade profiles is a prerequisite to developing high-performance compressors. So we are going to design a compressor blade in CATIA. after finish the blade we are create a domain in GAMBIT and finally we are going to analysis in FLUENT software these are the software packages we are using in this project for only the two dimension analysis.In three dimension we are going to mesh in ICEM and analyzing in CFX these are the software packages we are going to use in this project.from these methods we can see various pressure distribution, velocity distribution and boundary layer separation of the linear compressor cascade.

The basic function of the blades is to turn the air to the required angle. Along this process, undesired loss (entropy generation) results. Therefore, the goal of the blade design is to achieve the desired flow turning with minimum losses, within the constraint of geometric orientation of the blade row required by the overall compressor design. Unlike an isolated airfoil for external flow application, blades of a turbo machine (including compressor and turbine) are used in a row and referred to as a "cascade" in the research. Definition for the cascade and the cascade nomenclature description can be found in the references. The aerodynamic performance of a compressor cascade is basically determined by five quantities as listed below

The shape of the blades
Cascade stagger angle (γ)
Cascade solidity ($\sigma = C/S$)
Inlet flow angle ($\beta 1$)
Inlet flow Mach number (M1)

During the blade inverse design process, all the above quantities are chosen from the overall compressor design requirement, except for the blade shape. While it is the design goal to generate blades which meet the geometrical and flow turning requirements of the compressor, of equal importance is how much losses are generated. Unlike a turbine cascade through which the flow is accelerated, the adverse pressure gradient due to the flow diffusion in a compressor cascade imposes unfavorable force on the boundary layer. Correspondingly, it is more difficult to achieve a thinner

ISSN(Online) : 2319 - 8753
ISSN (Print) : 2347 - 6710

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

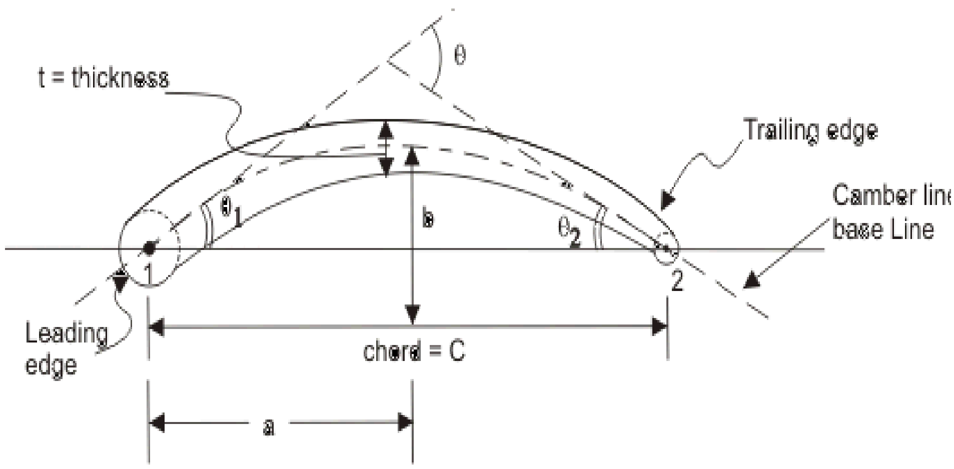
Vol. 4, Issue 3, March 2015

boundary layer and/or control the boundary layer from separation, which is the major factor determining the aerodynamic losses of the blades. In particular, when the flow speed is elevated such that transonic flow is formed in the blade passage, shock loss is introduced; and the shock boundary layer interaction makes it more difficult to control the boundary layer development. Therefore, the primary design challenge for a compressor blade is to Generate a blade shape with losses as low as possible.

II. BLADE AND CASCADE NOMENCLATURE

Since airfoils are employed in accelerating and diffusing the air in a compressor, much of the theory and research concerning the flow in axial compressors are based on studies of isolated airfoils. The nomenclature and methods of describing compressor blade shapes are almost identical to that of aircraft wings. Research in axial compressors involves the inter effect of one blade on the other; thus, several blades are placed in a row to simulate a compressor rotor or stator. Such a row is called a cascade. When discussing blades, all angles which describe the blade and its orientation are measured with respect to the shaft (Z axis) of the compressor.

The airfoils are curved, convex on one side and concave on the other, with the rotor rotating toward the concave side. The concave side is called the pressure side of the blade, and the convex side is called the suction side of the blade. The chord line of an airfoil is a straight line drawn from the leading edge to the trailing edge of the airfoil, and the chord is the length of the chord line as seen in figure 6. The camber line is a line drawn halfway between the two surfaces, and the distance between the camber line and the chord line is the camber of the blade. The camber angle θ is the turning angle of the camber line. The blade shape is described by specifying the ratio of the chord to the camber at some particular length on the chord line, measured from the leading edge. The aspect ratio AR is the ratio of the blade length to the chord length. The term “hub-to-tip ratio” is frequently used instead of aspect ratio. The aspect ratio becomes important when three-dimensional flow characteristics are discussed. The aspect ratio is established when the mass flow characteristics are discussed. The aspect ratio is established when the mass flow and axial velocity have been determined.



III. METHODOLOGY

Compressor blade cascade is a critical component to design has effects on the overall performance of the compressor. The role of the compressor blade cascade is described followed by a description of compressor blade types.The grid independence test is done which involves transforming the generated physical model into a mesh with number of node points depending on the fineness of the mesh. The various flow properties were evaluated at these node points. The extent of accuracy of result depended to a great extent on the fact that how fine the physical domain was meshed.

ISSN(Online) : 2319 - 8753
ISSN (Print) : 2347 - 6710

International Journal of Innovative Research in Science, Engineering and Technology

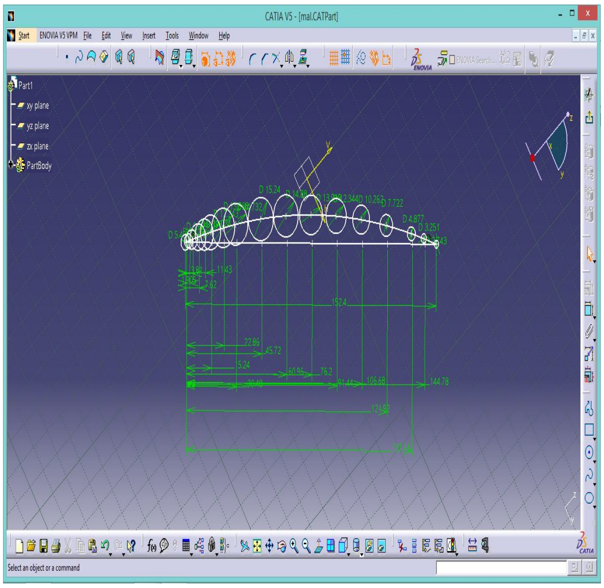
(An ISO 3297: 2007 Certified Organization)

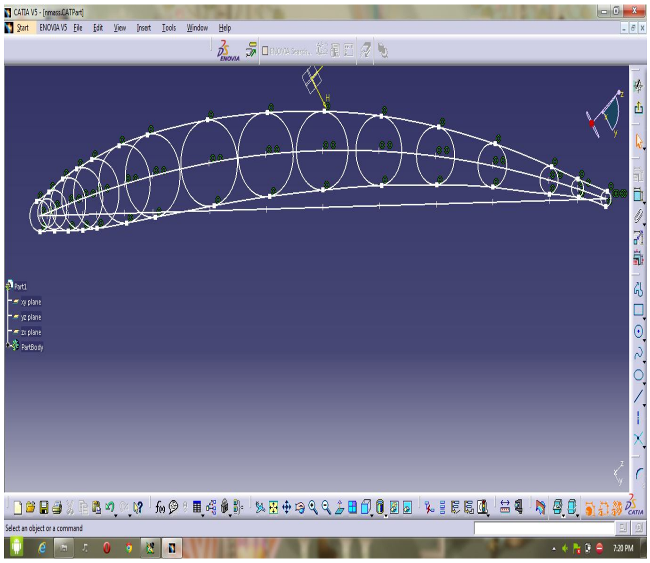
Vol. 4, Issue 3, March 2015

After a particular refining limit the results changes no more. At this point it is said that grid independence is achieved. The results obtained for this mesh is considered to be the best. This mesh formation was done with GAMBIT.The 2D modeling scheme was adopted in GAMBIT and it was analyzed using FLUENT. A compressor cascade model with zero degree flow angle of incidence was designed. The design should be in the basis of the airfoil database with the airfoil co ordinate points

IV. MODELLING

Geometry creation in CATIA is done with the required commands from the geometry creation tool pad. The geometry creation tool pad contains specification of compressor blade camber line. To design a few modes of compressor blade with different specifications.
To create geometry by using software CATIA.





V. GRID GENERATION

Introduction
Meshing creation in gambit is done with the help of required commands from the meshing creation tool command buttons that allows performing operations which include creating edge meshing, face meshing and boundary conditions. For the numerical study, inlet geometry parameters such as inlet ramps angles, length, number of ramps, cowl deflection and contraction ratio are varied. Axisymmetric inlets with sharp and rounded leading edge also meshing with rectangle domain can be create in this Chapter

Computational domain
The 2D modeling scheme was adopted in GAMBIT. The structured grids were generated using ANSYS Gambit meshing tool.
Meshing can be done in forms namely edge meshing, face meshing.
Meshed edge, faces can be copied, moved, linked or disconnected from one another.
Structured grid cells are used for entire domain. Cells are clustered at the region.
Grading schemes includes successive ratio. Double sided grading also can be performed. The interval count can be specified for the starting mesh based on the model. In face or 2Dmeshing the following parameters can be specified. Meshing schemes mesh node spacing and face meshing options.

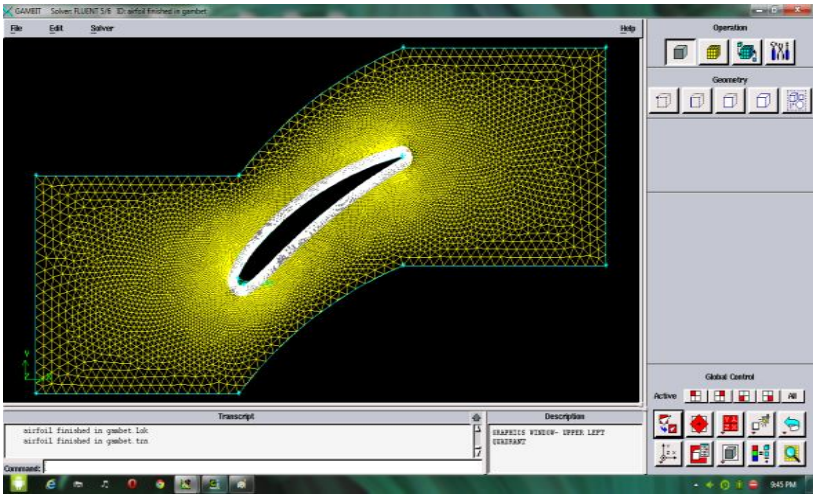
ISSN(Online) : 2319 - 8753
ISSN (Print) : 2347 - 6710

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 3, March 2015

The meshing schemes include the elements and the types. Quadrilateral can be used as the elements. The meshing type pave are used



Meshing
Total no. of elements – 45968
Total no. Of nodes – 37507
Element – tri
Type – pave

Boundary condition
Inlet – velocity inlet
Outlet – pressure outlet
Upper wall
Lower Wall
Blade – wall

The grid for 2D models generated using the software GAMBIT and the other specification discussed. Grid independence study results in formation of fine grids to obtained desired results. Separated domains was selected based on several iterations were chosen. The initialize boundary condition for the compressor blade is given.

VI. ANALYSIS OF A COMPRESSOR BLADE

The two dimensional meshed file have been imported in FLUENT software then it is created two periodic after that it has been analyzed in grid or mesh mode and finally it has been analyzed in velocity vector and static pressure approach so we can plot various graph by using the software.

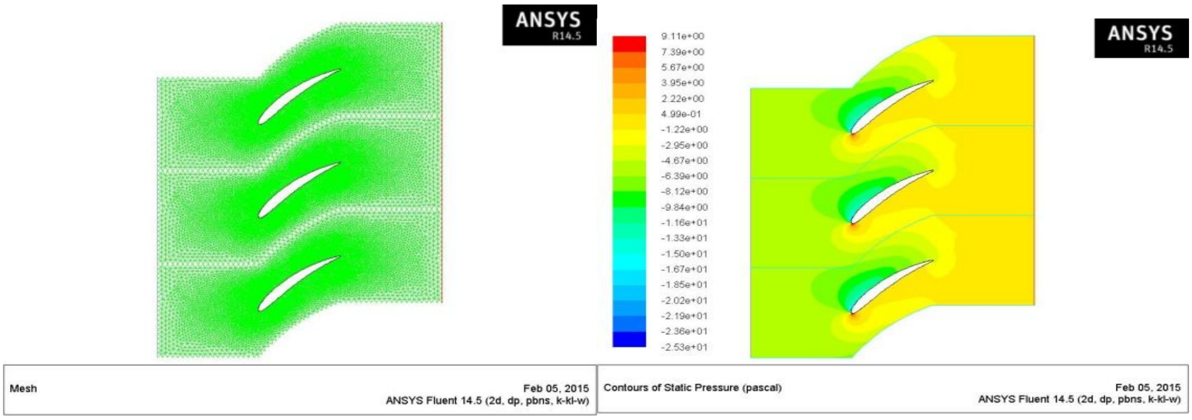
Meshing results in fluentContours of static pressure

ISSN(Online) : 2319 - 8753
ISSN (Print) : 2347 - 6710

International Journal of Innovative Research in Science, Engineering and Technology

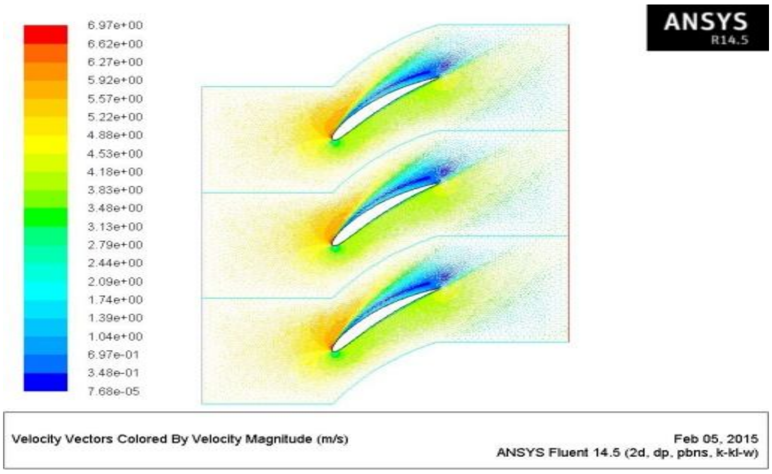
(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 3, March 2015



After finish the mesh in GAMBIT we have import the design in FLUENT software here we are going to create two periodic walls to ensure the flow will be similar in both the blades are not. After the completion of the grid or mesh the test section we analyzed the blades in FLUENT software in that the velocity inlet values will be given with respect to the Reynolds number.

Velocity vector



The compressor blade has been analyzed in two dimensional so we can analysis the pressure and velocity distribution over the compressor blade. The blade has analyzed in various angle of incidence and various velocity inlet conditions

VII. DESIGN IN 3 DIEMENSIONAL

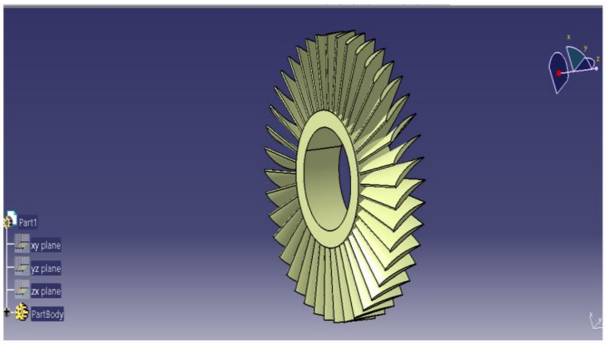
Design cascade blade in CATIA meshed in ANSYS(ICEM) and finally analyzed in CFX from these software's we will get various results like pressure distribution and velocity distribution over the cascade blades.
Design of a compressor cascade Meshing in ANSYS(ICEM)

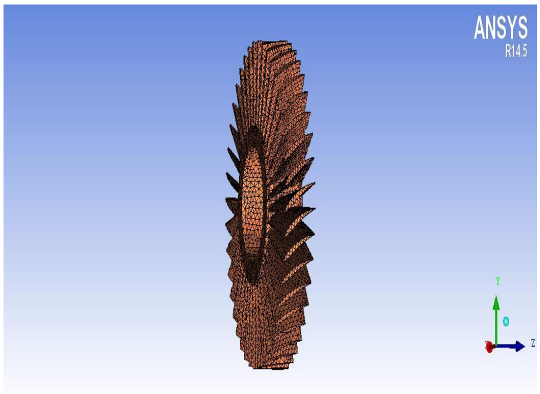
ISSN(Online) : 2319 - 8753
ISSN (Print) : 2347 - 6710

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 3, March 2015

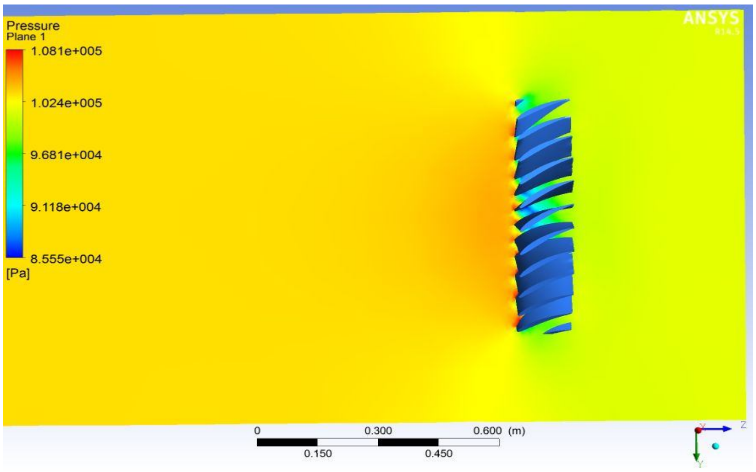




The compressor blades has designed in three dimension with various parameters the main thing in three dimension we have to consider twist angle and no of blades which we have used in the cascade form each blade has some angle of twist and pitch so we can get the fine mesh in the ICEM software by using various commands if the mesh rate is increased then the mesh will be fine and gives good accuracy. mesh we are used unstructured mesh in this project.

Analysis in CFX

Pressure distribution over the blade



In the pressure distribution we have used various commands and various pressure values over the blade in stable condition.the pressure and velocity may be vary in static and dynamic condition. the pressure will be increaseinfront when the blade is in static this pressure will be decrease when it is in dynamic condition.the presssure will be high infront of the blade section

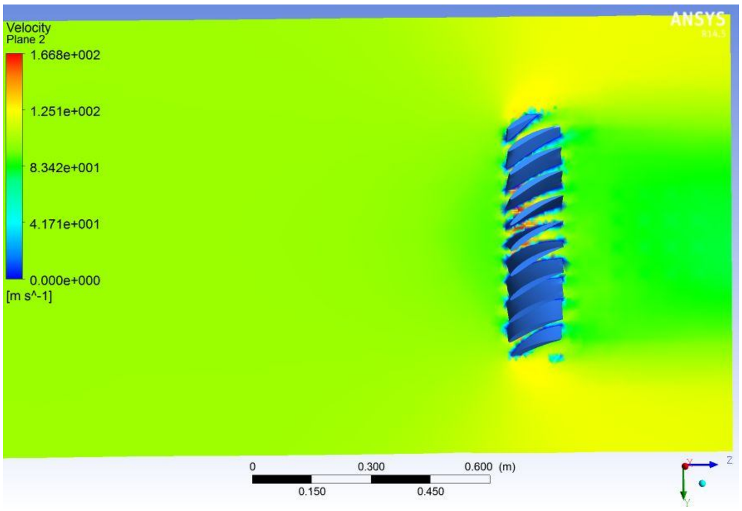
ISSN(Online) : 2319 - 8753
ISSN (Print) : 2347 - 6710

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 3, March 2015

Velocity distribution over the cascade



The velocity distribution over the cascade has been determined by the applying various velocity in the domain inlet so we can easily determine the velocity distribution and where the the velocity will be high in the cascade so these are all the velocity distribution over the cascade.

VIII. CONCLUSION

The two dimensional and three dimensional results have been made by using various software's. Finally we compared to experimental result with analytical result so the experimental result and analytical result in two dimensional will be the same at last the experimental and analytical results have compared

Experimental

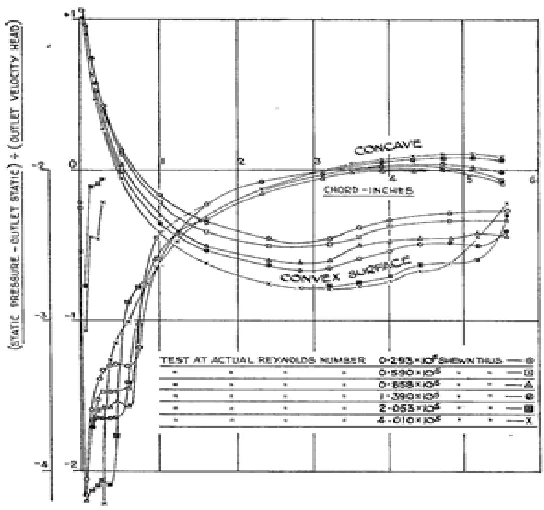
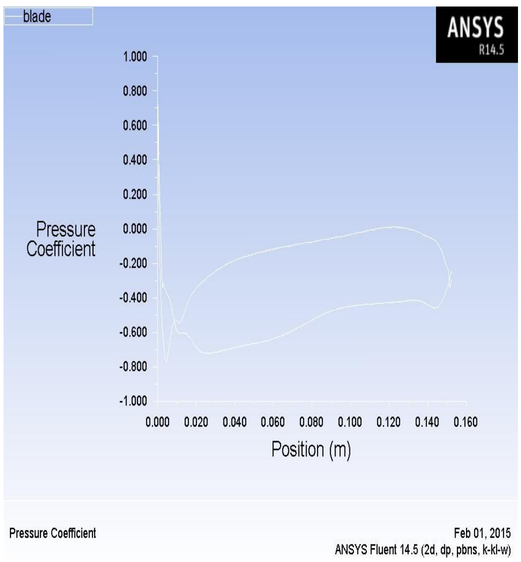


FIG. 33. Pressure distribution around central cross-section of middle blade. 30-deg camber cascade. Nominal α_1 = 35 deg. Incidence = — 16 deg.

Analysis



REFERENCES

[1]A. D. S. Carter, R. C. Turner, D. W. Sparkes and R. A. Burrows, 1960,the design and testing of an axial-flowcompressor having different blade profiles in each stage , *aeronautical research counctl reports and memoranda*

ISSN(Online) : 2319 - 8753
ISSN (Print) : 2347 - 6710

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 3, March 2015

[2] A.X. Lio and C.X. Lin, “Three BEM Schemes For The Calculating Of Subsonic Compressible Plane Cascade Flow”, Engineering Analysis with Boundary Elements 11; 1993 : 25-32.
[3] F.Bakhtar and K.S So, “A Study Of Nucleating Flow Of Steam In A Cascade Of Supersonic Blading By The Time-Marching Method”, International Journal Of Heat & Fluid Flow, Vol. 12, No.1,Butterworth-Heinemann,1991.
[4] J. Delery& G. Meauze, “A Detailed Experimental Analysis Of The Flow In A Highly Loaded Fixed Compressor Cascade: The Iso-Cascade Co-Operative Programme On Code Validation”, Aerospace Science & Technology 7,Elsevier;2009:1-9.
[5] Li Qiushi, Wu Hong, and Zhou Sheng, “Application Of Tandem Cascade To Design Of Fan With Supersonic Flow”, Chinese Journal of Aeronautics, Elsevier; 23(2010):9-14
[6] S.J. Andrews, B.Sc.(Eng), 1955, Tests related to the effect of profile shape and camber line on compressor cascade performance,*aeronautical research counctlreports and memoranda*