International Journal of Engineering Research and General Science Volume 3, Issue 4, July-August, 2015
ISSN 2091-2730

Numerical Investigation of Fluid Flows over a Rotor-Stator(Stage) in an Axial Flow Compressor Stage

Mr Vamsi Krishna Chowduru, Mr A Sai Kumar, Dr Madhu, Mr T Mahendar

M.Tech (Thermal Engineering), MLR Institute of Technology, +919505109726

Abstract:Although compressor blades have long been shrouded for aerodynamic and structural reasons, the importance of the pressurization over the cascade in the axial compressor stage is being investigated. However, the effects of the leakage tangential velocity variation on the blade passage flow are unknown. An experimental investigation of the loss and flow turning in the blade passage in shrouded axial compressor cascades subject to the variation of the leakage tangential velocity. First, increasing the leakage tangential velocity reduces overall loss. Second, increasing the leakage tangential velocity spreads loss core in the pitch-wise direction so loss core becomes more two-dimensional. Third, increasing the leakage tangential velocity makes the near hub passage flow more radially uniform. Our research predicts the streamline contour occurred at the Rotor Root-Stator Tip to predict the variation of flow. Research over selected series of airfoil at angle of 22° at the Root, 4° at the Tip and vice versa is being predicted using Computational Tools. The Aim is to identify the convergence criteria at lowest possible values of iterations. Based on the design data the approach is being done using ANSYS CFX to perform computational results.

Keywords— Compressor, Blades, Pressurization, Velocity, Stream Lines, Losses, Rotor-Stator, Stage, Computational, Converngence.

Introduction

The compressor is one of the three primary components of a gas turbine engine along with the combustor and turbine. Of these components, the compressor has certain aerodynamic limits which usually set the range of operation of the engine. The compressor is limited by choking at higher flow rates and by stall or surge at lower flow rates. Here in this thesis our main concentration is on single stage axial flow compressor. An axial flow compressor is one in which the flow enters the compressor in an axial direction (parallel with the axis of rotation), and exits from the gas turbine, also in an axial direction. The axial flow compressor compresses its working fluid by first accelerating the fluid and then diffusing it to obtain a pressure increase. The fluid is accelerated by a row of rotating airfoils (blades) called the rotor, and then diffused in a row of stationary blades (the stator). The diffusion in the stator converts the velocity increase gained in the rotor to a pressure increase.

Typical Axial flow compressor characteristics are tabulated table

Type of application	Type of Flow	Inlet relative velocity Mach number	Pressure Ratio per Stage	Efficiency per Stage
Industrial	Subsonic	0.4-0.8	1.05-1.2	88%-92%
Aerospace	Transonic	0.7-1.1	1.15-1.6	80%-85%
Research	Supersonic	1.05-2.5	1.8-2.2	75%-85%

So it is important to optimize the Stage pressure ratio, efficiency and operating range of the transonic axial flow compressor. The design and analysis of axial flow compressor has become core area of interest to many researchers due to its wide applicability in areas like aerospace, marine, power generation etc. Many analytical and experimental techniques are developed to design and analyze the axial flow compressors. Numerous mathematical optimization techniques are developed to optimize the design parameters of axial flow compressor stage. Axial compressor can elaborate a higher flow than the radial, which has a higher pressure ratio per stage, this means that for the same flow rate the firsts will have a smaller diameter, but it will need more stages to reaches the same pressure ratio.

Another aspect to consider is the efficiency, which reaches better values in the axial one, because the flow withstand less changes of direction along the stages, with minor perturbation through each blade row. For the same mass flow and pressure ratio radial compressor are cheaper than the other, furthermore they are more resistant in case of damage caused by external object.
It possible to see the behaviour of both compressors in relation to velocity and pressure ratio, is it clear that radial compressors have more margin to the surge, and axial compressor should be used only at high speed. The main character of this thesis is the axial compressor, which is become the main choose for the most of the application from gas turbine for electric energy production, because of the growth of turbogas plant, to engine for aircraft.

The increase of efficiency in gas turbine has been obtained from the increase in pressure ratio in the compressor and the increase in firing temperature in the combustion chamber; in the axial compressor the total pressure ratio is due to the sum of the increase obtained in each stage, which is limited to avoid high diffusion.

BLADE NOMENCLATURE:
Generally an axial compressor is compose by a variable number of stages where each follow one other, a single stage is made up by a rotor and a stator; both of them present blades disposed in a row, called cascade.

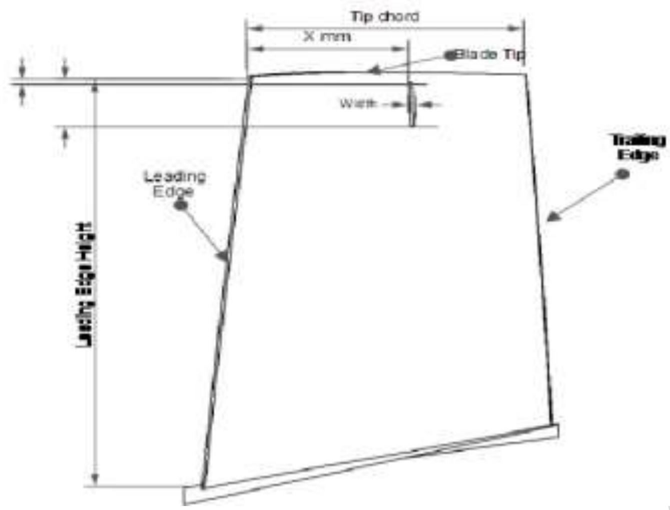
A blade has a curved shape, convex on one side, called suction side, and concave on the other, called pressure side, the symmetric line of the blade is the *camber line*, whereas the line which connect directly the leading and trailing edge is the *chord line*, the distance between these two line is the camber of the blade.

The turning angle of the camber line is called *camber angle, ϑ,* and the angle between the chord line and the axial direction is the *stagger angle*, γ.

Chord Length	150mm
Pitch-to-chord ratio (S/C)	0.52
Aspect ratio (H/C)	1.56
Outlet blade angle	4 degree
Stagger angle	22 degree
Number of blades	3+3
Operating Pressure	1 atm
Temperature	300 K
Velocity	20 m/s
Fluid	Air
Blade Height	17.7
Blade Tip Chord Length	7.7

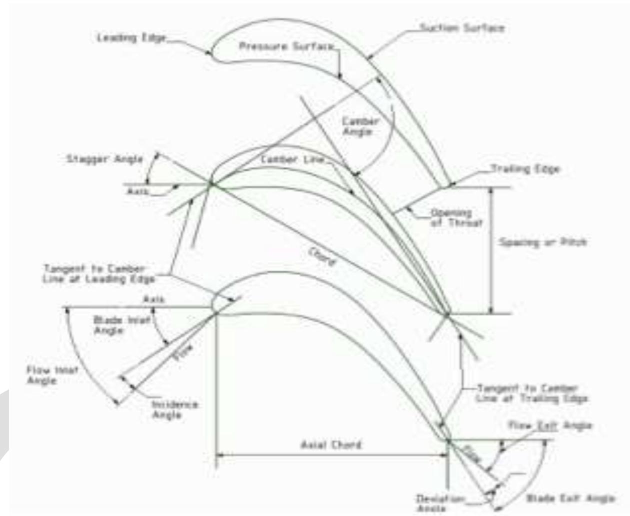
ISSN 2091-2730

ROTOR DIMENSIONS



ROTOR HUB	
Chord(mm)	150
Thickness (%)	100
Pitch(deg)	22
ROTOR TIP	
Chord(mm)	77
Thickness (%)	100
Pitch(deg)	4

STATOR DIMENSIONS



STATOR HUB	
Chord(mm)	150
Thickness (%)	100
Pitch(deg)	4
STATOR TIP	
Chord (mm)	77
Thickness (%)	100
Pitch (deg)	22

THEORY & MODELS:

A review of the modelling concerned turbomachinery, starting from Euler work equation until CFD model, passing throughout bi-dimensional and three-dimensional flow.

FUNDAMENTAL LAWS

It is possible to write the elementary rate of mass flow like

$$d\dot{m} = \frac{dm}{dt} = \rho c dA_n$$

where $\boldsymbol{dA_n}$ is the element of area perpendicular to the flow direction, c is the stream velocity and $\boldsymbol{\rho}$ the fluid's density.

In one dimensional steady flow, where we can suppose constant velocity and density, defining two consecutive station, 1 and 2, without accumulation of fluid in the control volume, it is possible to write the *equation of continuity:*

$$\dot{m} = \rho_1 c_1 A_{n1} = \rho_2 c_2 A_{n2} = \rho c A_n$$

The fundamental law used in turbo machinery field is the *steady flow energy equation*:

$$\dot{Q} - \dot{W} = m\left[(h_2 - h_1) + \frac{1}{2}(c_2^2 - c_1^2) + g(z_2 - z_1)\right]$$

but, some observation can be do, first of all flow process in this field are adiabatic, so it is possible to consider $\dot{Q}$ equal to zero, than the quote different $(z_2 - z_1)$ is very small and can be ignored, thus, considering that compressors absorbed energy we can write

$$\dot{W} = m(h_{02} - h_{01})$$

h_0 is called stagnation enthalpy and is the combination of enthalpy and kinetic energy:

$$h_0 = h + \frac{1}{2}c^2$$

For a compressor the work done by the rotor is

$$\tau\Omega = \dot{m}(U_2 c_{\theta 2} - U_1 c_{\theta 1})$$

Where τ is the sum of the moments of the external forces acting on fluid, U is the blade speed and $c_{\theta 2}$ is the tangential velocity. So the specific work is

$$\Delta W = \frac{\dot{W}}{\dot{m}} = U_2 c_{\theta 2} - U_1 c_{\theta 1}$$

also called *Euler work equation*.

Combining the above equations it is possible to obtain the relation between the two stations, which in our case are the inlet and the outlet of the rotor and the stator:

$$h_2 + \frac{1}{2}c_2^2 - U_2 c_{\theta 2} = h_1 + \frac{1}{2}c_1^2 - U_1 c_{\theta 1}$$

those two terms are known as rothalpy I, which is constant along a single streamline through the turbomachine; it is also possible to refer it at the relative tangential velocity becoming

$$I = h + \frac{1}{2}(w^2 + U^2 + 2Uw_\theta) - U(w_\theta + U) = h_{0rel} - \frac{1}{2}U^2$$

having define the relative stagnation enthalpy as

$$h_{0rel} = h + \frac{1}{2}w^2$$

In the turbomachinery field is not possible to consider the fluid incompressible anymore, due to the Mach number that is bigger than 0.3; using the local value of this parameter we can relate stagnation and static temperature, pressure and density

$$\frac{T_0}{T} = 1 + \frac{\gamma-1}{2}M^2$$
$$\frac{p_0}{p} = \left(1 + \frac{\gamma-1}{2}M^2\right)^{\frac{\gamma}{\gamma-1}}$$
$$\frac{\rho_0}{\rho} = \left(1 + \frac{\gamma-1}{2}M^2\right)^{\frac{1}{\gamma-1}}$$

Combining these three equations and the continuity one, non dimensional mass flow rate is obtained:

$$\frac{\dot{m}\sqrt{C_p T_0}}{A_n p_0} = \frac{\gamma}{\sqrt{\gamma-1}} M\left(1 + \frac{\gamma-1}{2}M^2\right)^{\frac{1}{2}\left(\frac{\gamma+1}{\gamma-1}\right)}$$

also known as flow capacity.

NUMERICAL INVESTIGATION OF AXIAL FLOW COMPRESSOR STAGE:

METHODOLOGY

Axial flow compressor blades are generated by using the CATIA V5 R20 modeling software as shown in the figure. Then the CFD simulations for the available axial flow compressor are carried out and the results of velocity streamlines and pressure at outlet are plotted. and analysis is done using ANSYS solver, the simulations are carried out for axial flow compressor blades at Velocity of 20m/s.

Analysis

CFD Analysis and study of results are carried out in 3 steps: Pre-processing, Solving and Post-processing.

Pre-Processing

Geometry of the CAD model is prepared in CATIA V5 R20 modeling software, and imported into the ANSYS Workbench by using the STEP format. Meshing is carried out using the ANSYS MESHER TOOL by defining the element size for the fluid domain and giving finer surface mesh on the regions of interest like inlet, outlet and wall.

Meshing

Mesh Information for Case CFX

Number of Nodes: 488410

Number of Elements: 2752198

Tetrahedral: 2752198

Maximum Face Angle for Case CFX

Min: 53.9666 [degree]

Max: 128.593 [degree]

Edge Length Ratio for Case CFX

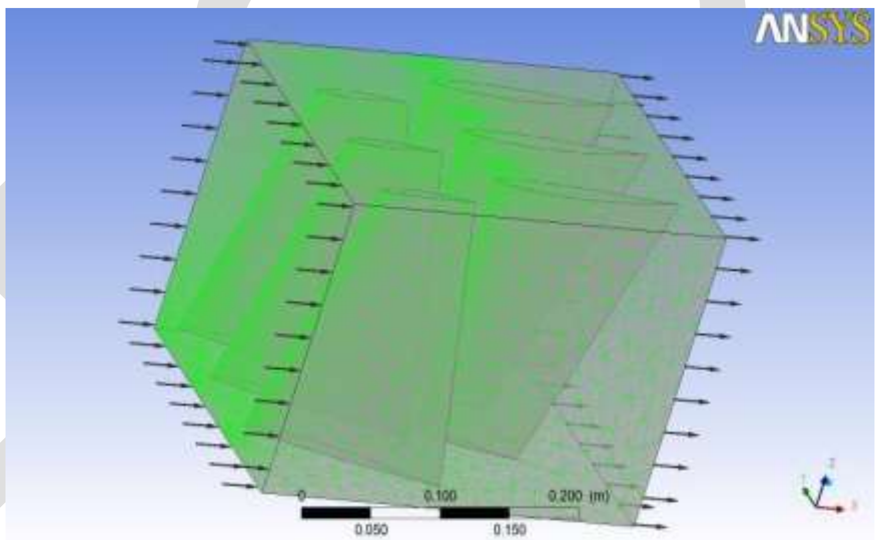
Min: 1.07186

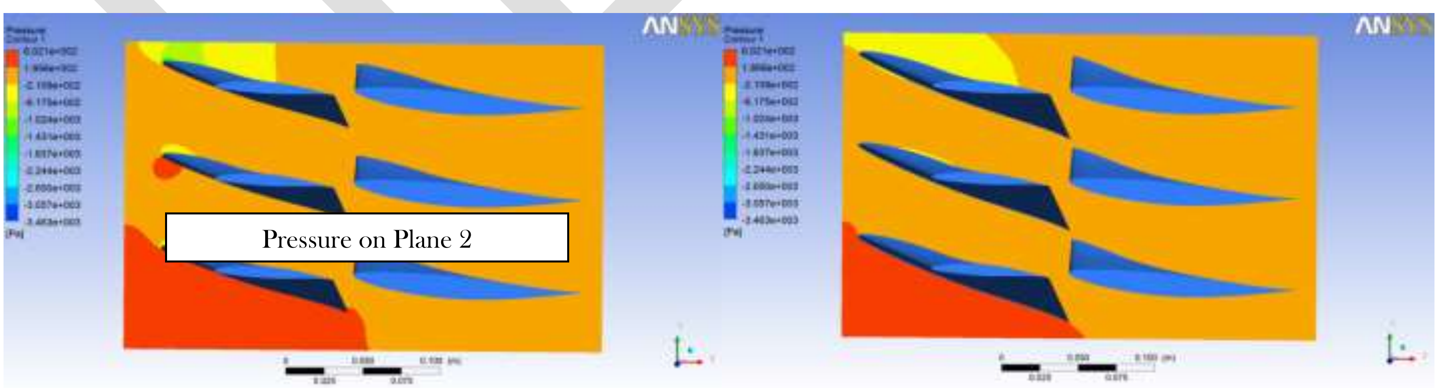
Max: 12.3327

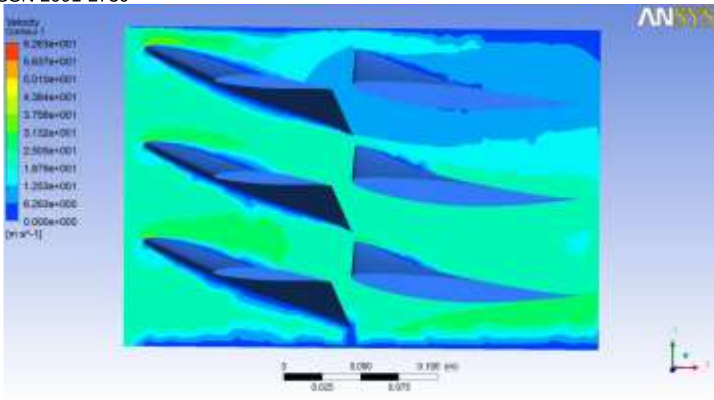
Element Volume Ratio for Case CFX

Min: 1.10976

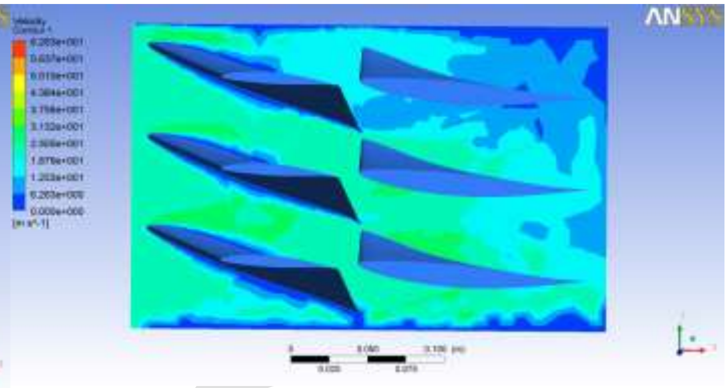
Max: 31.5171





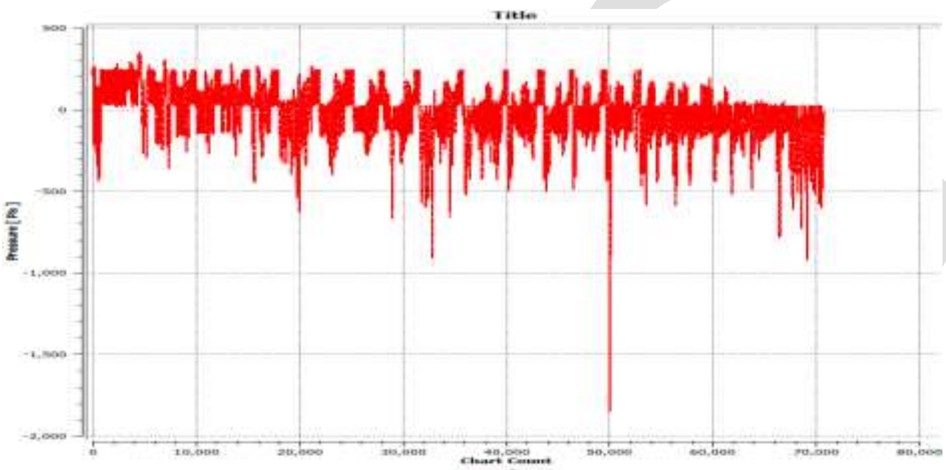


Velocity on Plane 1

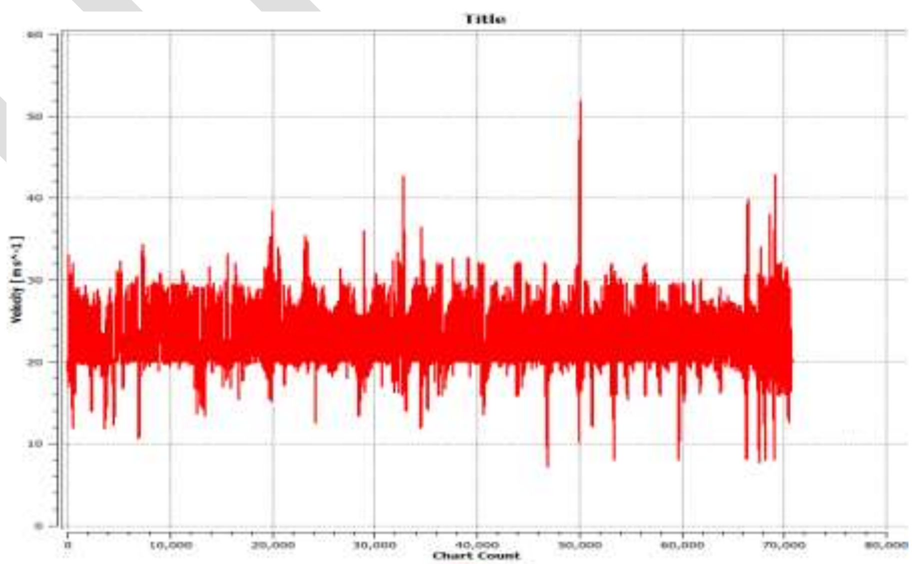


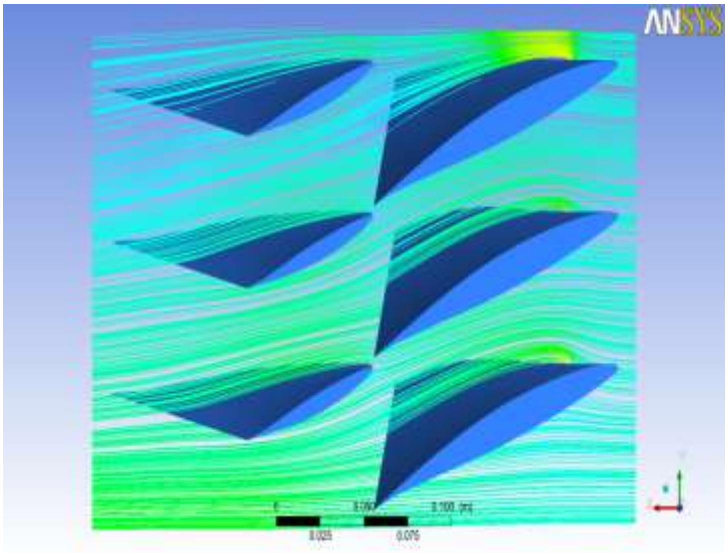
Velocity on Plane 2

Pressure Contours

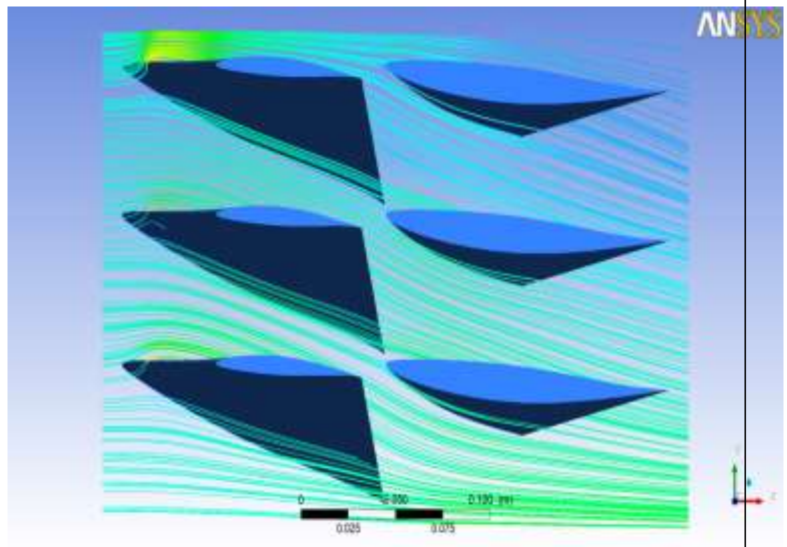


Velocity Contours





Stream Line Contours Plane 1



Stream Line Contours Plane 2

ACKNOWLEDGMENT

First of I would like to thank Dr Madhu Head of the Department Mechanical Engineering, MLR Institute of Technology, Mr T Mahendar, Assistant Professor for continuous guidance and support towards my project. My Sincere thanks to Mr A Sai Kumar Assistant Professor MLR Institute of Technology for Technical Support for successful completion of my work.

CONCLUSION

The Performance of the selected blade profile is clearly examined. The results are validated with the results of various blade profiles. It also observed that the Pressure over the Rotor Blades have transformed the Relative, Tangential and Normal Velocity. The reaction forces on the next consequent Rotor is being extended of further Research. The velocity contour have clearly satisfied for the Low Pressure region with a velocity of 20m/s. The Constant performance of the Pressure and Velocity contours display the exact convergence obtained by the Design. The solutions are being inspected at velocity of 20m/s at two planes as different instances.

REFERENCES:

[1]. M Javed hyder and M.Omair Khan," *Development of Novel method for the selection of Material for Axial compressor blade*", Conference on failure of Engineering materials and structures, MEDUET TAXILA, Feb 2007, pp: 73-78.

[2]. Niclas Falck, "*Axial compressor mean line design*", masters thesis, Lund University, Sweden, Feb 2008.

[3]. Hanoca P and Shobhavathy M T "*CFD analysis to investigate the effect of axial spacing in a single stage transonic axial flow compressor*" Symposium on Applied Aerodynamics and Design of Aerospace Vehicle (SAROD 2011) November 16-18, 2011, Bangalore, India.

[4]. MC Kenzie AB, "The *design of axial compressor blading based on tests of a low speed compressor*", proceedings of IMech (E), *Vol 194*, *issue 6*, 1980, pp: 103-111.

[5] H Cohen , GFC Rogers & HIH Sarvanamato, "*Gas turbine theory* ",Pearson education limited ,fifth edition,

[6] **Nikolay Ivanov, Valery Goriatchev, Evgueni Smirnov and Vladimir RIS**, 'CFD analysis of secondary flow and pressure losses in a NASA Transonic turbine cascade.' *Conference of modelling fluid flow*, *Hungary September 2003*.

[7] **J. Dunham** 'A review of cascade data on secondary losses in turbines'.

[8] **J.H. Horlock, J.D. Denton** 'A review of some early design practise using Computational fluid dynamics and a current perspective.' *Cambridge University Engineering Department.*

[9] **A. R. Howell, M.A.** 'Flow Dynamics of Axial Compressors.'

[10] **Johan Hjarne, Valery Chernoray, Jonas Larsson, Lennart Lofdahl** 'An Experimental Investigation of secondary flows and loss development downstream of a highly loaded low pressure turbine outlet guide vane cascade.' *Asme Turbo Expo 2006*.

[11] **Piotr Lampart, '**Investigation of end wall flows and losses in axial turbines. Part i. formation of end wall flows and losses.' *Journal of theoretical and applied mechanic 47, 2, pp. 321-342, Warsaw 2009*

[12] **Anderson, J. D.** Jr., 1995: *Computational Fluid Dynamics, The Basics With Applications, McGraw-Hill Inc., Singapore.*

[13]. **B. Lakshminarayan & J.H. Horlock** , Text book of turbomachinery

[14] **Yahya S.M.** 1998, *Turbines, Compressors and Fans*, Tata Mc-Graw-hill publishing company limited.

[15] **Dixon S.L.** 1998, *Fluid mechanics and thermodynamics of turbo machines*, Reed Educational and Professional Publishing Ltd.

[16] **Ghoshdastidar P**, *Computer simulation of flow and heat transfer* , TMH New Delhi publication

[15] **Fluent,** (1998) FLUENT 6 User´s Manual, *Fluent. Inc*

[16] **Gambit** user's manuals.