Published by :
http://www.ijert.org

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 01, January-2016

Integrated Approach for the Flow Analysis in Axial Turbine Stages using Computational Fluid Dynamics (CFD)Tools

Deepthi K R
Assistant Professor,
Mechanical Engineering Department
New Horizon College of Engineering,
Bangalore – 560103, Karnataka, India

Dr. M S Ganesha Prasad
Dean – Student Affairs & HOD - ME,
New Horizon College of Engineering
Bangalore, Karnataka, India

***Abstract* -The paper analyzes the flow through axial turbine stages and the results of two numerical simulation of steady and unsteady flow are presented. Generally analysis of turbines is carried on individual blade rows i.e nozzle guide vanes (NGV) and rotor blades. It doesn't take the non-uniformity of the exit conditions of the upstream blade vanes. Hence, turbine stage analysis must be carried out to predict the stage performance. In this study an integrated approach to analyze the turbine stages by computational fluid dynamics (CFD) tools using Fine /TURBO is carried out.**

Keywords: Comparison, Model Turbine, Experiment

1. INTRODUCTION

The general purpose of rotating machines is to exchange mechanical energy with a flowing stream of fluid. The design of these devices is one of the most critical and difficult steps in the engine development process, for no progress with real hardware can be made until all of the rotating components are in working order [1].

The gas flow inside turbine stages is highly complex and major design requirements completely rely on greater extent on its internal aerodynamics .Turbine stage design details are generally not available owing to its proprietary nature. It is predominantly guided by experimental methods and past experience. These experimental methods are inherently slow and also very costly especially at engine operating conditions. These are its drawbacks and growing need to understand the complex flow field phenomenon involved, has led to the development of numerical methods for predicting flows in turbine stages. These models are usually used to optimize the design of the turbine component and reduce the volume of the experimental work and cost involved.

A comprehensive model is needed in order to simulate the real flow situation in a turbine stages which should incorporate all these factors and are expressed in the form of governing differential equations. The solution of these differential equations is time consuming. However, with rapid development in computer memory and speed, more realistic flow simulations are increasingly attempted now. In the last few decades significant attempts have been made to develop numerical procedures for predicting flow in the turbine stages. Accurate predictions of such flows are required to design a high performance and reliable turbine. A few examples of steady and unsteady multistage turbomachinery flow prediction capability include those developed by J. Swirydczuk [2,3], J. Yao[4,5], S. H. Chen[6], Maciej Karczewsi [7].

The present project is based on the computational analysis of flow through a multistage turbine which is employed in the propulsion system of Gas turbine engine.

Design and the analysis of the turbine stages is important for the overall performance of the engine. So the study of flow characteristics in the turbine stages through CFD simulation is undertaken.

The main objectives are as follows:

- Flow analysis by steady conditions in 1-1-1 passage.
- Flow analysis by unsteady conditions in 6-7-6 passage.
- To check the non-uniformity in the exit conditions of the upstream blade vanes.
- Study the performance of the stages through pressure loss

The main numerical tool used in this analysis was the code FINE Turbo of NUMECA .This tool allows the achievement of complete simulations of 3D internal and external flows from the grid generation to the visualization, without any file manipulation, through the concept of project

The geometry of the analyzed turbine is based on the 1+1/2 stage model turbine. This analysis deals with the analyzing the variation of flow properties with in stages such as velocity vectors, Temperature, Mach number and mass flow rate.

A Steady flow analysis was performed using an rotor-stator interaction of conservative coupling by pitch-wise row for 1-1-1 passage of 1.5 axial turbine stages. Two unsteady flow calculations were performed with 1-1-1 passage and 6-7-6 passage of 1.5 axial turbine stages. Nonlinear harmonic method (NLH) was used to simulate flow in 1-1-1 passage. Domain scaling method with frozen rotor technique was used to simulate flow in 6-7-6 passage. The results obtained from the CFD analysis over these grids have been compared in-terms for comparison with time averaged flow parameters along with inlet flow conditions and pressure loss was also determined by comparing with steady flow analysis.

(This work is licensed under a Creative Commons Attribution 4.0 International License.)

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 01, January-2016

2. AXIAL TURBINE GEOMETRY

The geometry of the analyzed model turbine bases on 1.5 stage model. The turbine stages consists of 3 rows in which 1st row is stator or nozzle guide vane, 2nd row is rotor and 3rd is stator or guide vanes. Two stator rows in this turbine are constructed using 36 blades. The inlet flow to each stator is axial. The rotor row consists of 41 blades. The tip clearance in the rotor row is equal to 0.0004m.The rotational speed of the turbine rotor is 3500 rev/min. The schematic of the turbine stages is shown in figure 1

3. GRID GENERATION & BOUNDARY CONDITIONS

The code Fine TURBO uses the grid of HOH type. In the present project,2 different type of configuration is used based on different passage counts per blade row: the 1-1-1 (36-41-36) and the 6-7-6 (36-42-36) have been generated using AUTOGRID software.
All the intricate features are meshed as per the geometry viz.,1st vane/stator,rotor,2nd stator or vane, hub and shroud.. Hexahedral element is used for the flow path simulation.
The details of mesh generation process for each component of turbine stages are shown below for 2 different passage configurations. The grid refinement in the boundary layer secured obtaining y+ level of an order of 25 in all directions.

1-1-1Passage (36-41-36)
Figures 2 (a) and (b) show the blade to blade mesh(2D mesh generation- it is used to control the mesh topology, grid clustering and mesh orthogonality) and then 3D mesh which combines meridional flow path and 2Dmesh to create the mesh on surface of revolution for 1-1-1 passage. The approximate number of grid nodes for all rows were equal to 2.4 million

6-7-6 Passage (36-42-36)
Figures 3 (a) and (b) show the blade to blade mesh(2D mesh generation- it is used to control the mesh topology, grid clustering and mesh orthogonality) and then 3D mesh which combines meridional flow path and 2Dmesh to create the mesh on surface of revolution for 6-7-6 passage. The approximate number of grid nodes for all rows were equal to 14.5milllion.

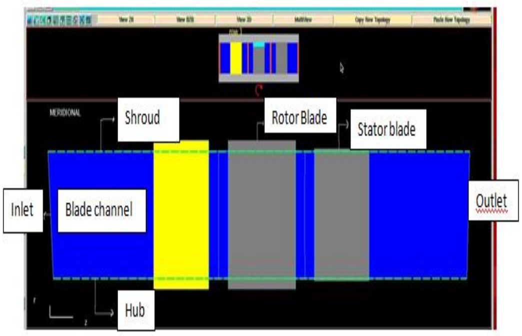


Fig1: Schematic of Turbine stages

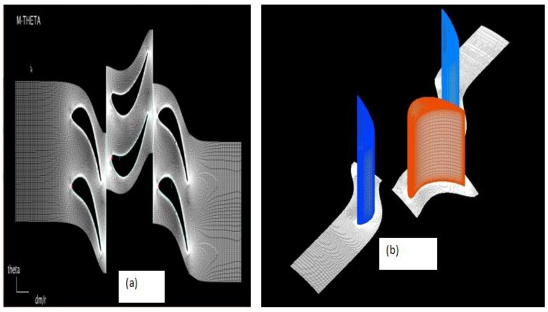


Fig 2: (a) Blade to Blade mesh (b) 3D hexahedral

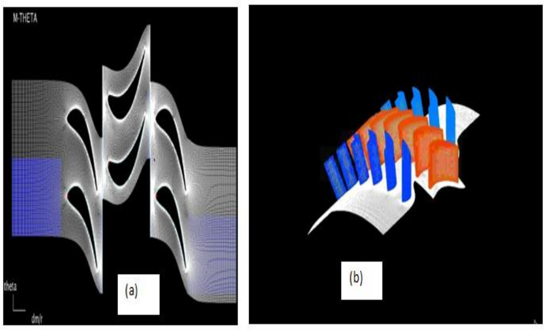


Fig 3:(a) blade to blade mesh (b) 3D hexahedral

The thermodynamic data assumed in the calculations based on this turbine are given in table 2.In the inlet plane these data are in the form of the distributions of total pressure ,total temperature along the turbine passage radius R and the parameter assumed at the turbine exit is static pressure is defined.

Table 1:Boundary conditions of 1-1-1 &6-7-6 passage

Inflow (Infront of 1st Stator)	Outflow(Behind 2nd stator)
MeanP_{tot}=1.6938bar MeanT_{tot} = 307.889K Tu=2%	Mean P_{stat} 1.1125bar

The Fine TURBO calculations were performed using Spallart Allmaras single equation differential model for turbulence.

4. RESULTS AND DISCUSSION

The comparison of models can also be made by observing the contours of the various flow parameters along the mid-span of modelled turbine for 1-1-1 passage with steady and Non linear harmonic (NLH) flow analysis and 6-7-6 passage. The relative velocity contours are compared for steady and unsteady analysis for 1-1-1passage and 6-7-6 passage.

Figure 4 shows the plot of relative velocity distribution in the mid plane for the entire turbine stages from inlet to exit channel.
The flow accelerates from v=0 to about v=150m/s at the outlet of stator1,further it decelerates in rotor stage. At the outlet of stator2 a flow velocity of v=200m/s is achieved.

(This work is licensed under a Creative Commons Attribution 4.0 International License.)

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 01, January-2016

It can be seen that the recirculation zone formed near the end of stator 2 is well defined in 6-7-6 passage compared to that of 1-1-1 passage modelled turbine.

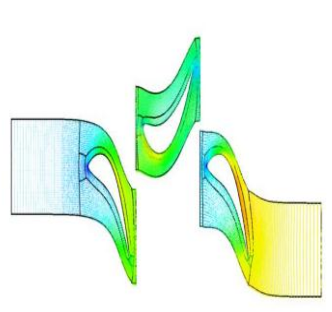
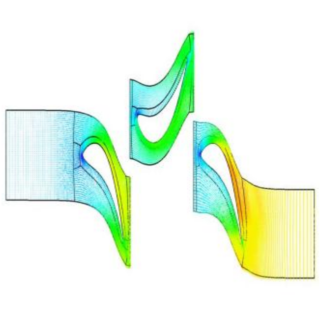
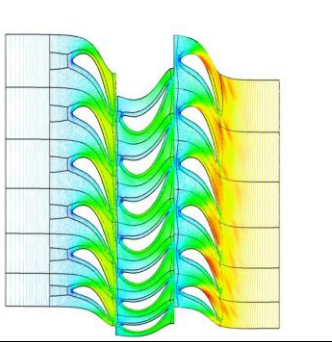
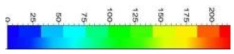
Relative velocity 1-1-1 passage (Steady analysis)	
Relative velocity1-1-1 passage (NLH method)	
Relative velocity 6-7-6 passage (Unsteady)	
W_{xyz}	

Figure 4: Relative velocity contours at mid span

The below figure 5 shows the contour plots of varied total pressure along the length of turbine for 1-1-1 passage. The value of pressure decreases from 1.698 bar to 1.4bar at the outlet of stator2. The absolute total pressure contour plots are compared for each of the steady and unsteady analysis of 1-1-1 passages

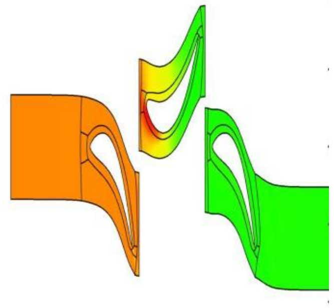
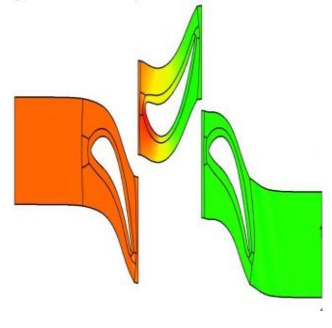
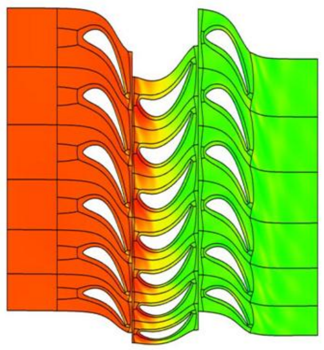
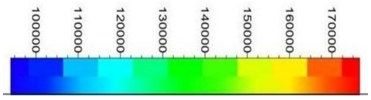
Total Pressure 1-1-1 passage (Steady analysis)	
Total Pressure 1-1-1 passage (NLH method)	
Total Pressure 6-7-6 passage (Unsteady)	
Total Pressure[Pa]	

Figure 5 Total pressure contours at mid span

Figure 6 shows the plot for total temperature at midsection. The inlet total temperature at the Stator1 is 305.778[K] and it remains constant upto outlet of stator1 further it reduces in rotor to 290K.At the outlet of stator 2 it remains constant of nearly 290K.The contour plots are compared with 1-1-1 and 6-7-6 passage for steady and unsteady analysis

(This work is licensed under a Creative Commons Attribution 4.0 International License.)

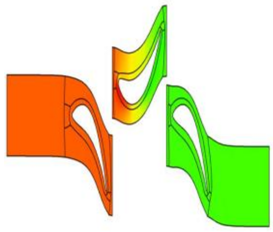
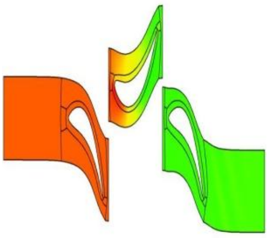
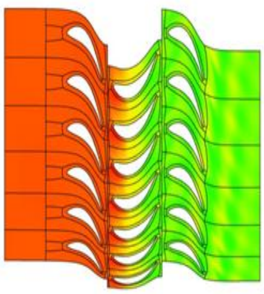
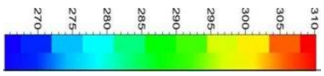
Total Temperature 1-1-1 passage (Steady analysis)	
Total Temperature 1-1-1 passage (NLH method)	
Total Temperature 6-7-6 passage (Unsteady)	
Total Temperature[K]	

Figure 6 Total temperature at mid span

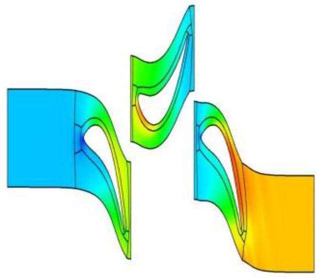
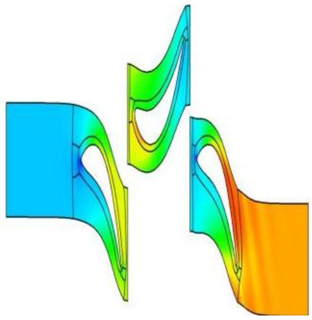
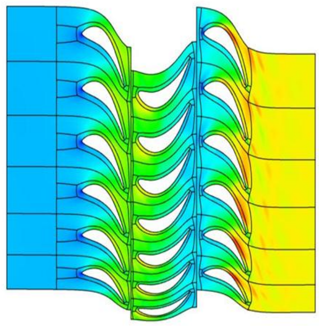
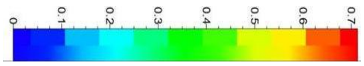
Relative Mach number1-1-1 passage (Steady analysis)	
Relative Mach number1-1-1 passage (NLH method)	
Relative Mach number 6-7-6 passage (Unsteady)	
Mach number	

Figure 7:Relative Mach number at mid span

Figure 7 shows the variation absolute Mach number along mid span of section for 1-1-1 and 6-7-6 passages. The uniform flow at the inlet of turbine is turned and distorted as it goes through the three blade rows. At the outlet of the last blade row, the Mach number contours reflect all of the flow patterns. At the inlet of stator 1Mach number is nearly equal to 0.1, then reduces to 0.45 at the outlet of stator 1.

In rotor inlet the Mach number 0.45 reduces to 0.12.further at stator2 exit ,the Mach number increases nearly upto 0.65 for 1-1-1 passages, but for 6-7-6 passage the Mach number at the outlet of stator2 is 0.6 to 0.48.

4.2 Time –Averaged Performance Parameters

The complete details of the time averaged overall performance parameters are discussed in this section. These time-averaged results together with the results of steady flow analysis are compared with inlet data at the inlet condition of all the blade rows

4.2.1 Vane-1 Exit

Figure 8 shows the comparison between the inlet condition of Stator/vane1 with the predicted circumferentially averaged absolute total pressure at the exit of vane 1.The inlet total pressure profile was used in all of the simulations as an upstream boundary condition is also included as a reference. The total pressure profile at the exit of the 1st vane is flat across most of the span as there is no upstream blade rows to distort the flow.

(This work is licensed under a Creative Commons Attribution 4.0 International License.)

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 01, January-2016

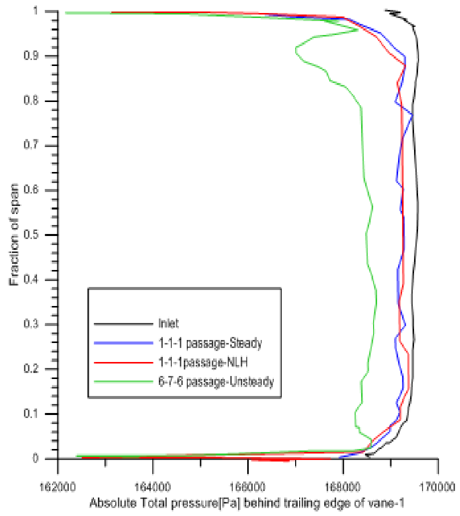


Figure 8: Absolute total pressure comparison behind trailing edge of vane-1

The differences between the various predicted results are clearly shown in figure 9.The secondary flows near both the end wall distort the total pressure profile at 8% span near the hub and 90% span near the tip.

The configuration 6-7-6 clearly improves the prediction of this vortex core and the total pressure profile as well. There is strong total pressure loss around 8%span caused by the hub passage vortex. This vortex is captured in all the simulations in different intensity.

The total pressure peak near 4%span is due to the fact that the hub passage vortex collects the low total pressure fluid in the inflow near the hub and pushes this high energy towards endwall and forms the edge of endwall boundary layer. Unsteady analysis predicts this well, while steady analysis gives thicker boundary layer which leads in peak moving radially upwards.

Figure 9 shows the comparison of absolute total temperature behind the trailing edge of vane-1 for 1-1-1 passage with steady and NLH analysis & 6-7-6 passage unsteady analysis. it is mainly compared with the inlet of flow channel condition

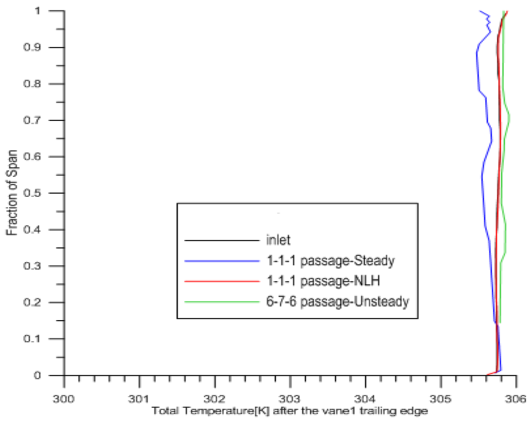


Figure 9: Absolute total temperature behind the trailing edge of vane-1.

Above figure 9 clearly depicts the comparison of total temperature for different analysis. There is no much variation in the total temperature with inlet condition, both the unsteady analysis for 1-1-1 passage and 6-7-6 passage prediction matches with inlet condition. There is bit deviation in 1-1-1 passage for steady analysis from 0.4 to 0.9% span.

4.2.2 Blade Exit

The comparison of absolute total pressure behind the trailing edge of rotor for different analysis is shown in figure 5.15.The absolute total pressure at this station for different solutions are unpredictable due to upstream flow. Even there maybe some discrepancy due to experimental results. The predicted profile for spanwise trends are consistent.

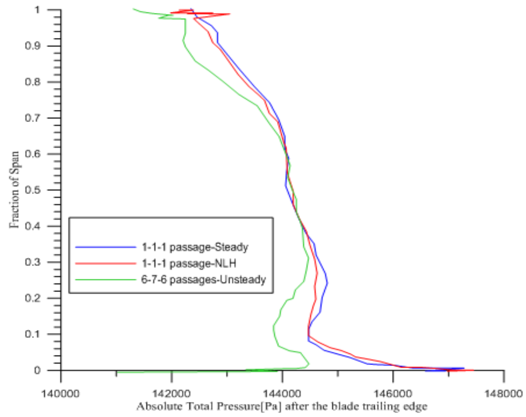


Figure 10 : Absolute total Pressure behind trailing edge of blade

The total pressure drop for each of the analysis do not capture the losses at the peak of 5%span.however,where the rotor hub passage vortex exist.

Figure 11 shows the comparison of absolute total temperature behind the trailing edge of the rotor blade for steady and unsteady analysis of 1-1-1 passage and 6-7-6 passages.

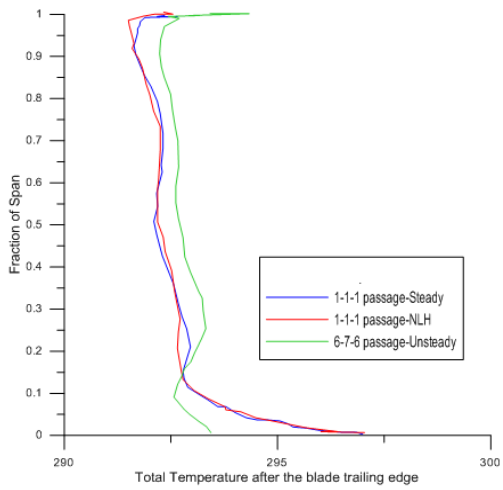


Figure 11: Absolute total temperatures behind the trailing edge of blade

(This work is licensed under a Creative Commons Attribution 4.0 International License.)

The unsteady flow analysis of the 6-7-6 passage shows better results than that of 1-1-1 passage for absolute total temperature analysis

4.2.3 Vane-2 Exit

The geometry of second vane/stator is same as of first vane, but this stator/vane2 receives a much more distorted inflow resulting from the effect of the vane-1 and rotor.

Figure 12 shows the comparison of the absolute total pressure behind the trailing edge of the stator/ vane-2.For comparison purposes, the total pressure profile at the inlet of vane-2 is also added in this figure. Near the end walls the pressure loss for all the blade rows results from the accumulation of loss. The 6-7-6 passage analysis improves the prediction near the end walls (mainly near the hub) over the 1-1-1 passage Non linear harmonic analysis and steady analysis of 1-1-1 passage due to the modeling of unsteady flow physics.

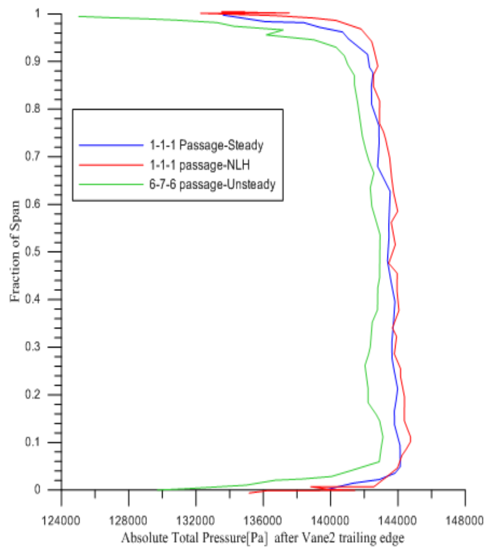


Figure 12 Absolute Total pressures behind the trailing edge of vane-2

Figure 13 shows the comparison of total temperature at the exit of the vane-2 for steady and unsteady analysis of 1-1-1 passage and 6-7-6 passage. No much variation in total temperature at the exit of the of vane-2 for steady and unsteady analysis.

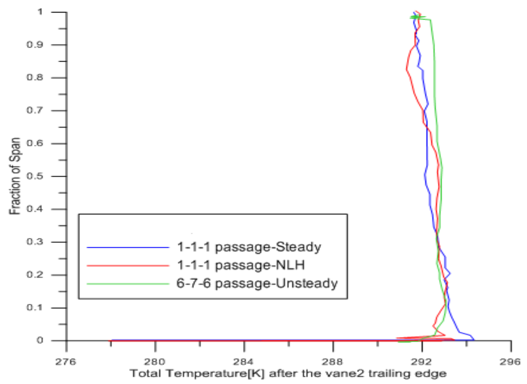


Figure 13 Absolute Total temperatures behind the trailing edge of vane-2

The uniform flow at the inlet of vane-1 is turned and gets distorted as it goes through all three blade rows. At the outlet of vane-2,The Mach number comparison for the steady and unsteady analysis of 6-7-6 passage and 1-1-1 passage is shown in figure 15. Comparisons are made for different passage and analysis and the solution for 6-7-6 passage almost accurately measures the various losses in the flow over the 1-1-1 steady and non linear harmonics analysis.

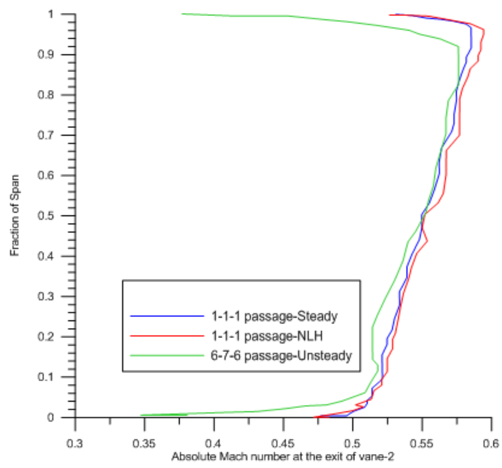


Figure 14 Absolute Mach number comparison

4.3 Turbine Performance Parameters

Turbine performance is evaluated in terms of the total pressure loss.

Total pressure loss across each component (%) = $\mathbf{P_0/P}$

Where Po is the circumferentially and radially mass averaged total pressure t inlet of a blade row and P is the total pressure of the point of interest.

The overall total pressure loss is calculated using a circumferentially and radially mass averaged total pressure at the exit of each blade row with the inlet pressure of each row. Table 2 shows the total pressure loss for each row for different analysis of 1-1-1 passage and 6-7-6 passage.

Table 2 Total pressure losses of each blade row

	Vane-1	blade	Vane-2
1-1-1-Steady flow	0.649%	0.6895:%	0.723%
1-1-1-NLH	0.6789%	0.6209%	0.921%
6-7-6 - Unsteady	0.4722%	0.4829%	0.943%

The losses of the steady flow analysis for 1-1-1 passage and non linear harmonics(NLH) analysis is greater than that unsteady analysis of 6-7-6 passage, but they still remain on same level.

(This work is licensed under a Creative Commons Attribution 4.0 International License.)

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 01, January-2016

5. CONCLUSION

The numerical analysis in the 1.5 axial turbine stages for 1-1-1 passage and 6-7-6 passage has drawn out following results.

•The secondary flows near both the end wall distort the total pressure profile at 8% span near the hub and 90% span near the tip for vane-1.
•Unsteady analysis predicts hub passage vortex well, while steady analysis gives thicker boundary layer which leads in peak moving radially upwards
•The absolute total pressure at rotor station for different solutions are unpredictable due to upstream flow
• The unsteady flow analysis of the 6-7-6 passage shows better results than that of 1-1-1 passage for absolute total temperature analysis.
•Stator/vane2 receives a much more distorted inflow resulting from the effect of the vane-1 and rotor.
•The 6-7-6 passage analysis improves the prediction near the end walls (mainly near the hub) over the 1-1-1 passage (Non linear harmonic analysis) and steady analysis of 1-1-1 passage due to the modeling of unsteady flow physics
•The losses of the steady flow analysis for 1-1-1 passage and non linear harmonics (NLH) analysis is greater than that unsteady analysis of 6-7-6 passage

6. REFERENCES

[1] Jack D. Mattingly, Aircraft engine Design ,2nd edition, American Institute of Aeronautics and Astronautics,2002
[2] Thomas.A.Ward,Aerospace propulsion Systems, John Wiley& sons,2010
[3] J.Swirydczuk, M. Szymaniak ,2011, “Numerical analysis of the flow structure in the RWTH Aachen model turbine”,ISSN 1727-0219
[4] J.Yao,A.Jameson,J.J.Alonso.2000,“Development and validation o a massively parallel flow solver for turbo machinery flows”,A00-16717.
[5] J.Yao,A.Jameson,J.J.Alonso ,2001, “Unsteady flow investigations in an axial turbine using the massively parallel flow solver TFLO,AIAA 2001-0529
[6] S.H.Chen and L.C.Lee ,1999. “Three-dimensional unsteady turbo machinery flow analysis”,ISABE 99-7283.
[7] Maciej Karczewski and Jaroslaw Blaszczak ,2008, “Performance of 3 turbulence model in 3d flow investigations for a 1.5 axial turbine”, tq312f-g/185

(This work is licensed under a Creative Commons Attribution 4.0 International License.)