International Journal of Mechanical Engineering and Robotics Research

ISSN 2278 – 0149 www.ijmerr.com
Vol. 3, No. 4, October, 2014

Research Paper

COMPUTATIONAL FLUID DYNAMICS (CFD) ANALYSIS OF INTERMEDIATE PRESSURE STEAM TURBINE

Shivakumar Vasmate[1]*

Corresponding Author: **Shivakumar Vasmate ✉ shivavasmate@gmail.com*

Steam turbines play a vital role in power generation as a prime mover which converts kinetic energy of steam into mechanical energy. The turbine normally consists of several stages with each stage consisting of a stationary blade and a rotating blade. Many of the utility steam turbines are of three cylinder constructions, followed by first the high pressure (HP), intermediate pressure (IP) and last low pressure (LP) stage respectively, where high, intermediate and low describe the pressure of the steam. A typical intermediate pressure cylinder module is chosen to carry out the project work. To achieve the above objectives we need to model separately the bladed region and attach the hub/shroud seal region to it by General Grid Interface (GGI). IDEAS software is used for geometric modeling, CFX TURBO-GRID software is used for meshing the bladed region, ICEM-CFD software is used for meshing the hub/shroud region of the seals and CFX Software is used for physics definition, solving and analyzing the problem. Analysis has been carried out for the 8th stage with and without seal. The results are compared and found to be in close comparison with two dimensional (2D) experimental calculations.

Keywords: Steam turbine, Hub/shroud, General Grid Interface (GGI), IDEAS software, CFX TURBO-GRID software, ICEM-CFD software, CFX software

INTRODUCTION

BHEL is manufacturing a wide variety of turbines over the last 50 years to meet India?s growing need for power. Steam turbine plays a vital role in power generation as a prime mover, which converts kinetic energy of steam to mechanical energy.

Many of the utility steam turbines are three cylinder constructions i.e. high pressure cylinder in which pressure is maximum with minimum specific volume and so blade height is minimum, intermediate pressure cylinder in which pressure is intermediate so the blade height is intermediate and low pressure cylinder which has a minimum pressure level and maximum specific volume and hence LP cylinder blade height is maximum.

A typical intermediate pressure turbine of

[1] Assistant Professor, Department of Mechanical Engineering, LAEC, Bidar, Karnataka, India.

utility steam turbine is chosen to carry out the CFD analysis. The analysis requires solving of fluid problem in bladed region. This can be done in three approaches, analytical, experimental and numerical.

Analytical methods which assume a continuum hypothesis are more suited for simple problems and are not suited for complex fluid flow problems. Experimental methods are suited for complex fluid flow problems but the expenditure for carrying out the analysis is high. The other limitation is that the determination of the fluid characteristics in the interiors becomes complex and difficult. Hence, numerical approach is more feasible approach for analysis of a particular design because it overcomes the limitations of the two methods and it gives a close approximate for complex form of fluid problems too. Numerical approach involves discretization of the governing mathematical equations gives numerical solutions for the flow problems.

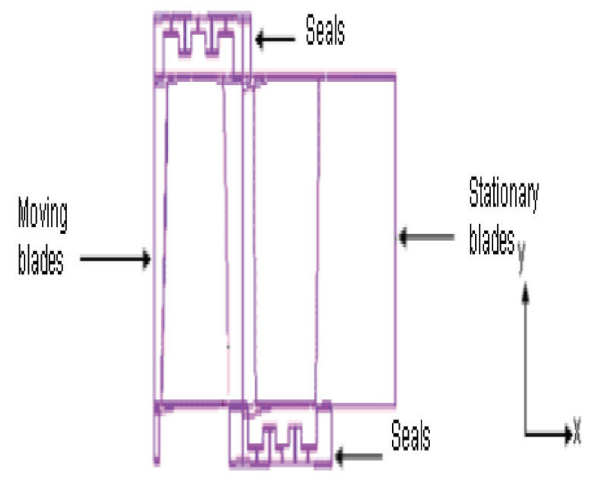
The analysis is carried out by identifying the flow domain. The domain is modeled discretized and governing equations are solved using commercially available software. The results are post processed and compared results which were obtained using 2 dimensional program developed and available at BHEL.

ELEMENTS OF STEAM TURBINE

The bladed region of steam turbine consists of the following as shown in Figure 1.

1. Stationary Blades
2. Moving Blades
3. Labyrinth Seals

Figure 1: Elements of a Steam Turbine



Aerofoil Blades

An aerofoil blade is a streamlined body having a thick, rounded leading edge and a thin trailing edge in order to achieve a high lift-drag ratio. Its maximum thickness occurs somewhere near the midpoint of the chord. Both the stationary and Rotating blades should be designed such that it should be capable of obtaining the desired pressure drop and turning towards the tangential direction between the driving surface and trailing surface of the vane passage, so that the flow comes out of the stationary blade with a desired velocity both in magnitude and direction. The exit flow will have high velocity with a high tangential component. Thus the flow enters axially in the stationary as well as the moving blades and both the tangential force and torque exerted by the fluid jet on the following rotating blade row depends on the change in the tangential velocity of the fluid. The blade with respect to axis and blade nomenclature is shown in Figure 2 and Figure 3 respectively.

Figure 2: Blade with Respect to Turbine Axis

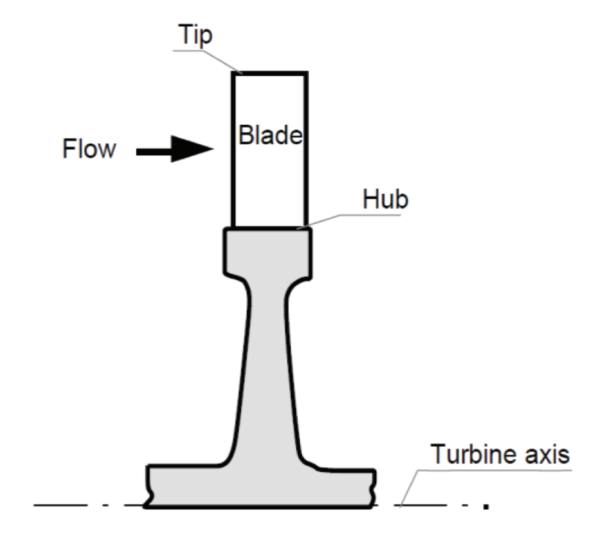
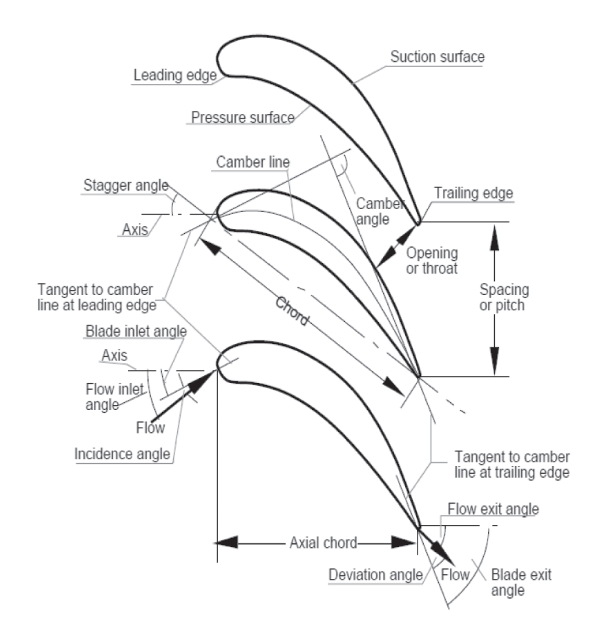


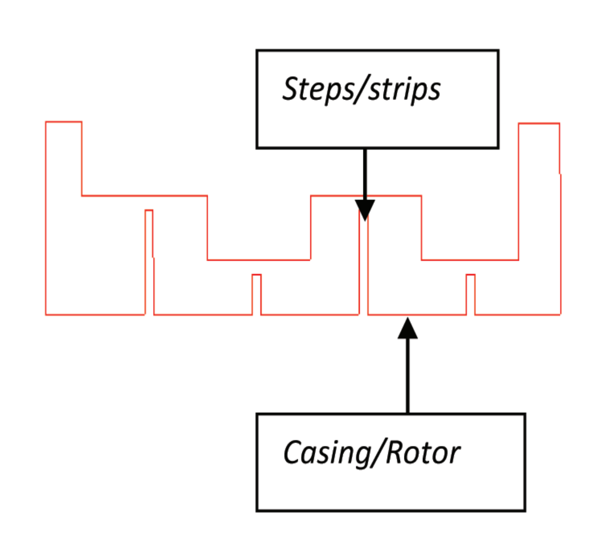
Figure 3: Blade Nomenclature



Labyrinth Seals

The provision of seals is necessary to minimize the leakage whenever there is a clearance between a moving and a stationary part with pressure difference across the clearance. In a steam turbine seals are provided at the two turbine ends where the shaft is taken out of the casing, at the clearance between the diaphragm and the rotor of an impulse stage and on the blade tips when provided with shrouds. Mostly the labyrinth and strip type of seals are used in the turbo machines. The number of strips used and their arrangement depends upon the pressure difference across the clearance and the basic construction arrangements used for sealing the diaphragm are shown in Figure 4 and these are generally known as Labyrinth seals. The flexible type of labyrinth seals used on diaphragms of the high pressure stages are as shown. Tip seals to the turbine stage in the CFD models are used for the more accurate stage performance predictions.

Figure 4: Flow Domain at Labyrinth Seal



METHODOLOGY

Geometrical Model

In order to analyze the flow and to evaluate the performance, basically three steps are required as follows:

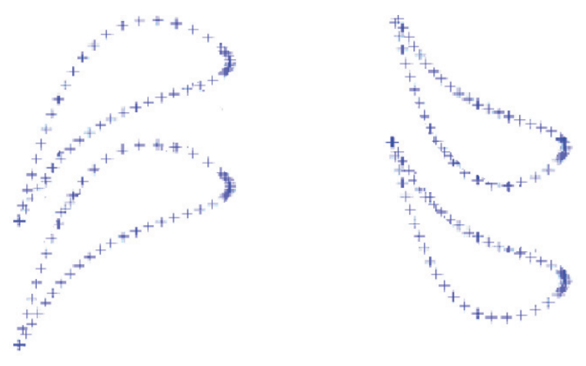
1. Modeling of components
2. Grid generation
3. Analysis

As the flow domain consists of blade and seal passages, the modeling is carried out as described below.

Geometrical Model of Blades

The blade of the IP steam turbine is of cylindrical type and blade extends between hub and shroud surfaces. The geometry of blade is extracted from blade profile co-ordinates are shown in Figure 5a and Figure 5b for 8th stage guide blade and moving blade respectively, given in the form of suction side and pressure side points, which are located along the radial positions of the blade.

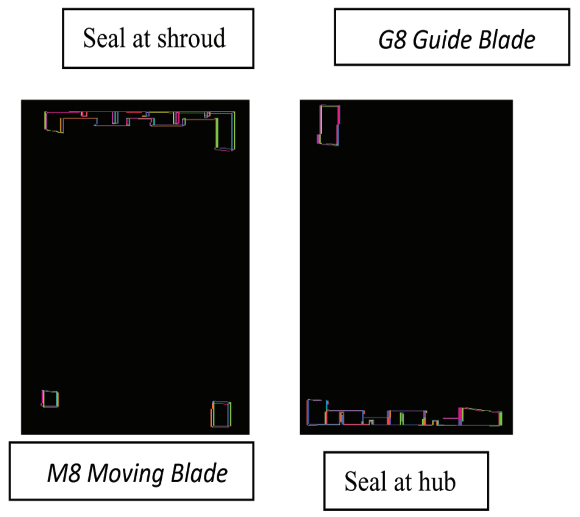
Figure 5a & 5b: 8th Stage Guide Blade and 8th Stage Moving Blade



Geometrical Model of Seals

Labyrinth seals are attached at the hub and shroud surface of the blades to reduce the leakage flow. Modeling of seals has been done in IDEAS by extracting the data from the AutoCAD drawing in Figure 6 by extruding the seals in either of the directions then the solid model of the seal with the required length is obtained.

Figure 6: Fluid Model of the Seals (Hub and Shroud)



Grid Generation of Blades Using CFX-Turbo Grid

The flow inside a Steam turbine passes through the bladed and seal passages, which can be described as periodic passages. Geometrically these passages are rotationally periodic about its axis of rotation. For the CFD analysis, it is assumed that the flow is also rotationally periodic in these passages. Therefore, the flow computation can be made in one of the periodic passage while applying periodic boundary conditions at periodic interfaces. For the purpose of flow domain discretization, one blade passage is considered for 3D-grid generation. The tool used for grid generation is CFX-TURBOGRID software package for the stator and rotor blade passages. Input to this software is given by three data files namely, hub.curve, shroud.curve, and profile.curve. These files contain the hub, shroud and profile curve data files in global cartesian coordinates or cylindrical form.

Hub Data File

The hub curve runs upstream to downstream and must extend of the blade leading edge. The hub data file contains the hub curve data points in cartesian form and downstream of the blade trailing edge. The profile points are listed, line-by-line, in free format ASCII style in order from upstream to downstream. These data points are used to place the nodes on the hub surface, which is defined as the surface of revolution of a curve joined by these points.

Shroud Data File

The shroud data file contains the shroud curve data points in cartesian or cylindrical form the shroud curve runs upstream to downstream and must extend upstream of the blade leading edge and downstream of the blade trailing edge the points are listed, line by line in free format ASCII style in order from upstream to downstream. These data points are used to place the nodes on the shroud surface, which is defined as the surface of revolution of a curve joined by these points.

Considering XZ plane with 'X' as axis of rotation, the Figure 7 shows, hub curve and shroud curve. Shroud. curve

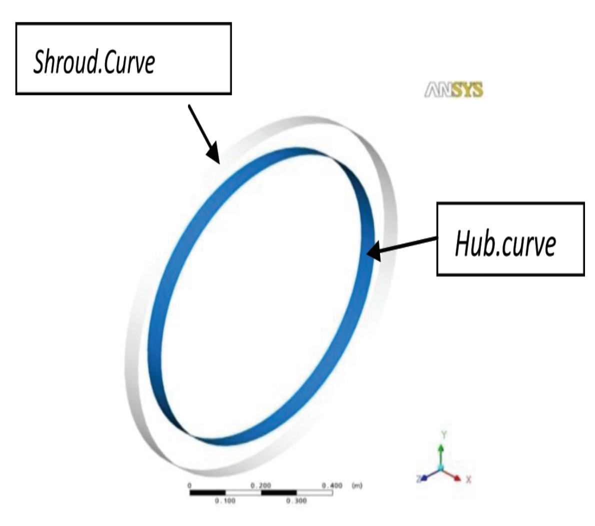
Shroud Curve

14.725	0	639.574
12	0	640.5
-12.2	0	645.5
-15.125	0	646.1158

Hub Curve

14.725	0	518.8
11.8	0	519.6
-11.8	0	522.4
-15.125	0	522.8

Figure 7: Hub Curve and Shroud Curve



Profile Data File

The "profile" data file contains the blade "profile" curves in cartesian or cylindrical form. The profile points are listed, line-by-line in free format ASCII style in a closed loop surrounding the blade. The blade profiles should lie on a surface of revolution to facilitate transformation to conformal space. A minimum of two blade profiles are required, one which lies exactly on the hub surface and one which lies exactly on the shroud surface. The profiles must be listed in the file in order from hub to shroud. Multi bladed geometries are handled by placing multiple blade profile definitions in the same profile.

Profile Curve:

Profile 1

16.3173	-3.7185	427.1885
16.2156	-4.2598	427.1924
15.8607	-5.0716	427.2057

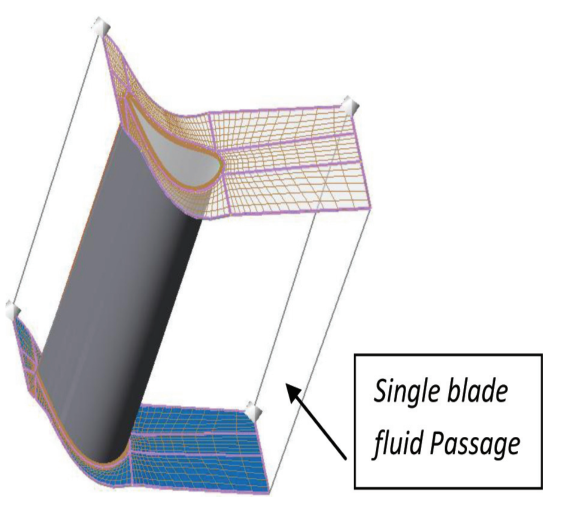
Profile 2

16.3173	-3.7185	427.1885
16.2156	-4.2598	427.1924
15.8607	-5.0716	427.2057

Profile Curve

The blade profile must lie on the surface of revolution of hub and shroud as shown in Figure 8.

Figure 8: Single Bladed Passage After Using



The mesh is generated for the stator and rotor blades with the total number of nodes, maximum and aspect ratio obtained from TURBOGRID are given in Table 1.

Table 1: Mesh Data for Components of 8th Stage Blades

S.No.	Component	Number of Nodes	Number of Hexa Elements
1	Stator Blade (G8)	1207444	112176
2	Rotor Blade (M8)	100380	93024

Seals Meshing Using ICEM-CFD

The total number of nodes, number of hexa element and aspect ratio obtained from ICEM-CFD In a steam turbine labyrinth or strip type of seals are invariably used. The flow domain of the seals is modeled in IDEAS from the 2D drawings and exported into ICEM CFD to generate the mesh. Before generating the hexahedral-mesh the geometry should be repaired in order to get no negative volumes and to get the better quality of the mesh are given in Table 2.

Table 2: Mesh Data for Data for 8th Stage Seals (Hub and Shroud)

S.No.	Component	Number of Nodes	Number of Hexa Elements
1	G8 Seals	68,096	58,860
2	M8 Seals	71,864	60,925

Ansys CFX

CFD analysis is carried out to understand the flow through the turbine, predict the pressure distribution and velocity profiles on the blades and predict the various losses. Ansys CFX-11 software tool is used for analysis purpose. The Analysis is carried out using CFX-Pre, CFX-Solver and CFX-Post modules.

RESULTS AND DISCUSSION

The analysis is carried out in two stages. First, individual stage analysis is done and later combined analysis for all the 5 stages has been carried out. The stage analysis has been carried out for the turbine stages with the constant mass flow and it consists of stator, rotor, and seals. The various performance parameters like pressure, temperature distribution and velocity profiles on the blades, isentropic efficiencies, power have been computed using the CFX Macros and with the help of mollier chart. As the eight stages consisting of guide blade, moving blade with a stage interface between the blades is simulated, and the solution is obtained with high resolution convergence up to 1e-5.The analysis is carried out with and without seals for 8th stage shown in Figures 9 and 10 respectively. The results obtained are discussed below.

Figure 9: Eighth Stage Blade with Boundary Conditions

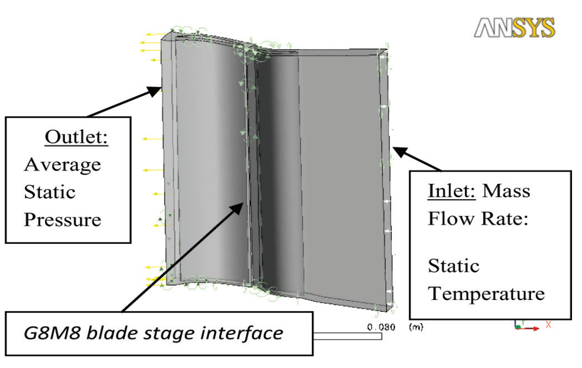
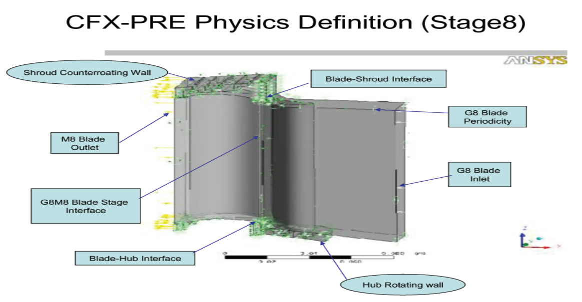


Figure 10: Boundary Conditions for 8th Stage with Seals



In the pre processing the following fluid domains and boundary conditions are specified for the eighth stage analysis.

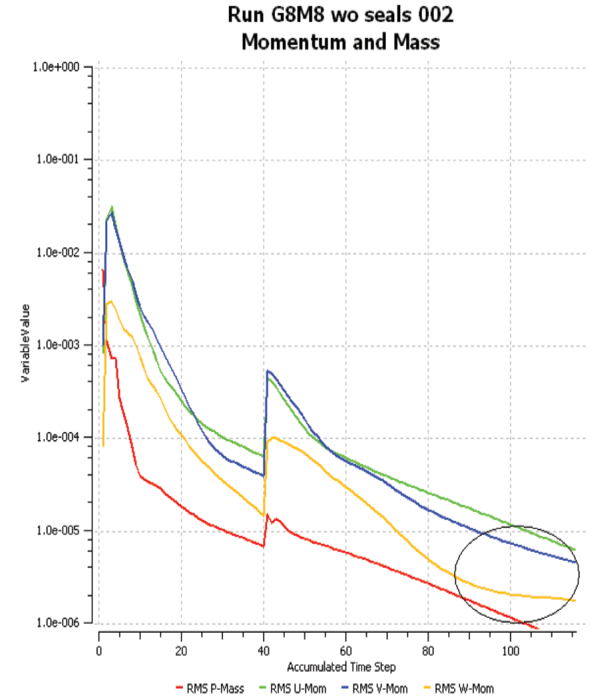
1. Simulation: Steady state
2. Domains: Fluid type
   - G8 blade: 8th Guide blade
   - M8 blade: 8th Moving blade
3. Boundary conditions
   - Inlet: Guide blade inlet
   - Outlet: Moving blade outlet
   - Inlet mass flow: 1.199795 kg/sec
   - Inlet static temperature : 688.8K
   - Wall: Smooth
   - Outlet static pressure :15.5 bar
   - Rotational speed :-3000 rpm
   - Reference pressure: 0 bar
4. Fluid Properties
   - Working fluid: Steam5 (Drysteam)
   - Dynamic viscosity: 25.0746e-6 Pa s
   - Thermal conductivity 0.0581942 W/ m. °c
5. Rotation axis : X - Axis
6. Turbulence model
   - Turbulence model: Standard k Epsilon model
   - Heat transfer model: Total energy
   - Interface between guide and moving blade
   - Type : Fluid -Fluid
   - Frame change option : Stage interface(G8M8 Blade stage interface)
8. Pitch change
   - Option: Specified pitch angle
   - Pitch angle side 1: 2.465753
   - Pitch angle side 2: 3.130435

For the present simulation solver parameters are specified as follows

- Advection scheme: Upwind (1st order) and High resolution (2nd order)
- Time scale control: Auto time scale
- Maximum iterations: 150
- Residual convergence criteria: RMS
- Residual convergence target: 1E-5
- Computational time : 6 hours and 20 min.

Run the Solver Monitor

The solver is allowed to run till the required convergence is obtained.



POST PROCESSING

Results which are obtained from the CFX macro for the eighth stage without seal.

User Input

Inlet Region	Inlet Region
Outlet Region	M8 blade outlet
Blade Row Region	M8 blade Default
Reference Radius	0.575325 [m]
Number of Blade Rows	115

Machine Axis	X
Rotation Speed	-3000 [rev min-1]
Gamma	1.3
Reference Pressure	0 [Pa]

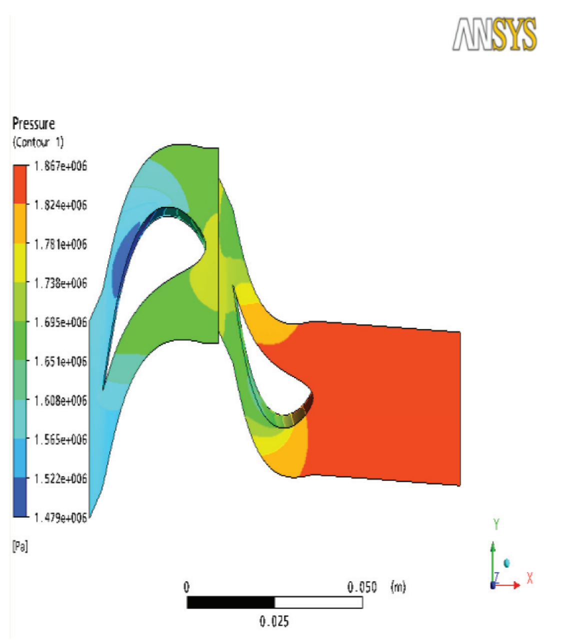
Mass Averages

Quantity	**Inlet**	**Outlet**	**Ratio (Out/In)**
Tempe rature	688.802 K	663.586 K	0.9633917 79100202
Total Tempe rature	689.863 K	665.91 K	0.9652781 59323341
Pressure	1.844e-+6 kg m^- 1s^-2	1.55+6 kg m^-1 s^-2	0.8407026 48752200
Total Pressure	1.85683e+ 06 kg m^- 1 s^-2	1.5734e+0 6 kg m^-1 s^-2	0.8473572 33011529
Enthalpy	3.28458e+ 06 m^2 s^-2	3.23404e+ 06 m^2 s^-2	0.9846126 64444908
Total Enthalpy	3.28672e+ 06 m^2 s^-2	3.23873e+ 06 m^2 s^-2	0.9853960 98760331

RESULTS

Torque (one blade row)	-233.017 kg m^2 s^-2
Torque (all blades)	-26769.6 kg m^2 s^-2
Power (all blades)	-8.03908 +06 kg m^2 s^-3
Total-to-total isen. efficiency	0.912117
Total-to-static isen. efficiency	0.851417

Figure 11: 8th Stage Pressure Contour Plot



The variation of pressure across the stage is seen in Figure 11, the pressure contour plot which is a series of lines linking points with equal values of a given variable pressure. The variable values can quickly be associated with the colored regions of the plot. It is shown in the Figure 11 that the pressure goes on decreasing from entry to exit of the stage. At the entrance the maximum of 18.56 bar is observed and a minimum of 15.5 bar is obtained at the exit is obtained.

In the pre processing the following fluid domains and boundary conditions are specified for the eighth stage analysis.

1. Simulation: Steady state
2. Domains: Fluid type
   - G8 blade: 8th Guide blade with hub and shroud seals
   - M8 blade: 8th Moving blade with hub and shroud seals
3. Boundary conditions
   - Inlet: Guide blade inlet
   - Outlet: Moving blade outlet
   - Inlet mass flow: 1.199795 kg/sec
   - Inlet static temperature : 688.8K
   - Wall: Smooth
   - Outlet static pressure :15.5 bar
   - Rotational speed :-3000 rpm
   - Reference pressure: 0 bar
4. Fluid Properties
   - Working fluid: Steam5 (Drysteam)
   - Dynamic viscosity: 25.0746e-6 Pa s
   - Thermal conductivity 0.0581942 W/m. °c
5. Rotation axis : X - Axis
6. Turbulence model
   - Turbulence model: Standard k Epsilon model
   - Heat transfer model: Total energy
   - Interface between guide and moving blade
   - Type : Fluid -Fluid
   - Frame change option : Stage interface(G8M8 Blade stage interface)

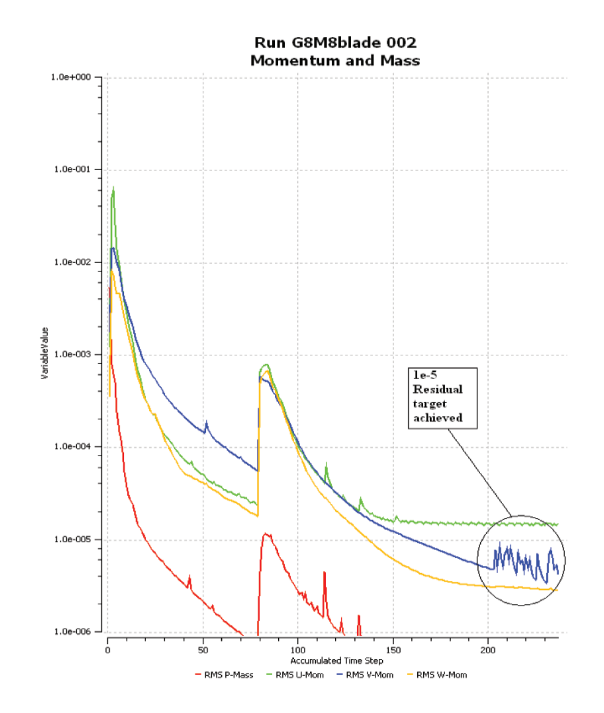
8. Pitch change
   - Option: Specified pitch angle
   - Pitch angle side 1: 2.465753
   - Pitch angle side 2: 3.130435

For the present simulation solver parameters are specified as follows

- Advection scheme: Upwind (1st order) and High resolution (2nd order)
- Time scale control: Auto time scale
- Maximum iterations: 300
- Residual convergence criteria: RMS
- Residual convergence target: 1E-5
- Computational time : 8 hours and 22 min.

Run the Solver Monitor

The solver is allowed to run till the required convergence is obtained.



POST PROCESSING

Results which are obtained from the CFX macro for the eighth stage with seal.

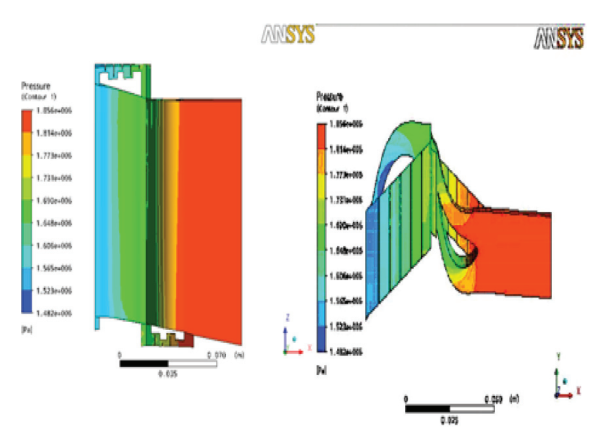
Inlet Region	G8 blade inlet
Outlet Region	M8 blade outlet
Blade Row Region	M8 blade default
Reference Radius	0.575325 [m]
Number of Blade Rows	115
Machine Axis	X
Rotation Speed	-3000 [rev min^-1]
Gamma	1.3
Reference Pressure	0 [Pa]

Mass Averages

RESULTS

Torque (one blade row)	-344.527 kg m^2 s^-2
Torque (all blades)	-28584 kg m^2 s^-2
Power (all blades)	-8.10974e +006 kg m^2 s^-3
Total-to-total isen. efficiency	0.925617
Total-to-static isen. efficiency	0.905874

Figure 12: Stage Pressure Contour Plots



The variation of pressure across the stage is seen in Figure 12, it is a pressure contour plot which is a series of lines linking points with equal values of a given variable pressure. It is shown in the Figure that the pressure goes on decreasing from entry to exit of the stage. At the entrance the maximum of 18.42 bar is observed and a minimum of 15.84 bar is obtained at the exit is obtained.

Quantity	Inlet	Outlet	Ratio (Out/In)
Tempe-rature	688.801 K	664.746 K	0.96118
Total Tempe-rature	691.248 K	675.846 K	0.93458
Pressure	1.84207e+ 006 kg m^-1s^-2	1.58472e+ 006 kg m^-1 s^-2	0.85235
Total Pressure	1.96454e+ 006 kg m^-1 s^-2	1.61431 e+ 006 kg m^-1 s^-2	0.89595

Comparison of CFD Values and 2D Values

The CFD analysis results are compared with 2D program output. The program output is verified experimentally. The comparison chart of 2D values and CFD values for 8th stage are shown in the table. The values obtained show that the CFD values are closer to 2D program and are within the acceptable limits.

STAGE 8 WITH SEALS

Description	Unit	2D value	CFD Value
Temp inlet	K	688.8	688.801
Temp outlet	K	667.1	664.746
Pressure inlet	Bar	18.04	18.4258
Pressure outlet	Bar	15.51	15.84
Output power	MW	8.0803	8.1097

CONCLUSION

CFD study was carried out for evaluating the performance of a utility Steam Turbine IP Module. The flow in a turbine blade passage is complex and involves understanding of energy conversion in three dimensional geometries.

- The performance of turbine depends on efficient energy conversion and analyzing the flow path behavior in the various components IP Steam Turbine.
- The CFD analysis of the turbine flow path helps in analyzing the flow and performance parameters and their effects on performance parameters like temperature, pressure and Power output.

The Intermediate Pressure turbine consisting cylindrical profiles used for stationary and moving blades. The blades are also designed with sealing strips between stationary parts and rotating parts to reduce leakage losses. The flow path of the turbine with blades and seals is modeled and meshed using different software's like IDEAS, ANSYS-ICEMCFD, ANSYS-TURBO-GRID, etc. The mesh for the blade region is generated separately with ANSYS-TURBO-GRID and mesh for the seals are generated from ANSYS-ICEM-CFD and attached by General Grid Interface. The analysis is carried out for a single stage initially and subsequently for all the combined 5 stages. The combined analysis consists of large number of element nodes with many General Grid Interfaces and stage interfaces between multiple frames of reference. IBM Cluster computing server with P615 processor is used to obtain the solution using 4 processors with 2GB RAM each. The solution is converged with 1e-5 with high resolution.

The results are analyzed for mass flow rates, temperature and pressure distributions on

REFERENCES

1. Ali Chaibakhsh *et al*., (2008), “Steam Turbine Model”.
2. Brendon *et al*., (1996), “Advance Steam Turbine Technology for Improved Operating Efficiency”.
3. K A Thole (2005), “Adiabatic Effectiveness Measurements of Endwall Film-cooling for a First Stage Vane”, *Journal of Turbomachinery by ASME*, Vol. 127, No. 297.
4. Knot *et al*., (2005), “Adiabatic Effectiveness Measurements of Endwall Film-cooling for a First Stage Vane”, *Journal of Turbomachinery by ASME*, Vol. 127, No. 297.
5. Lethander *et al*., (2004), “Optimizing the Vane-endwall Junction to Reduce Adiabatic Wall Temperatures in a Turbine Vane Passage”, *by ASME*.
6. Michael W *et al*., (2005), “Steam Turbine Modernization Solutions Provide a Wide Spectrum of Options to Improve Performance”, *Siemens Power Generation (pg), Germany, Siemens*.
7. Rainer Quinkertz *et al*., (2010), “High Efficiency Stem Turbine Packages for Concentrated Solar Power Plants”.
8. S P Harasgama *et al*., (1991), “Measurements and Computations of External Heat Transfer and Film Cooling in Turbines”, *Technical Memorandum P 1223, (ISABE)*, pp. 1-6.
9. Stanley Schurdak (2012), “Steam Turbine Blade Design”, *Conference Session b6, 2214, Twelfth Annual Freshman Conference*.
10. Sundranayagam *et al*., (2001), “The Effect of the Variation of Axial Velocity Incompressible Flow Through a Cascade of Aerofoils”, Vol. 130, No. 02, pp. 1019.
11. Vivek Asthana (2008), “Performance of Power Plants with High Temperature Conditions at Sub-critical Pressures”, *5th European Thermal-Sciences Conference, The Netherlands*.