Scientific Journal of Impact Factor (SJIF): 4.72
e-ISSN (O): 2348-4470
p-ISSN (P): 2348-6406
International Journal of Advance Engineering and Research Development
Volume 4, Issue 12, December -2017

DESIGN OF 20-30 MW RADIAL INFLOW STEAM TURBINE FOR POWER GENERATION

T.VenkataNagendra Kumar[1],Y.Rameswara Reddy[2]

[1]Departmentof Mechanical engineering, JNTUA College of Engineering (Autonomous), Pulivendula.
[2]Departmentof Mechanical engineering, JNTUA College of Engineering (Autonomous), Pulivendula

Abstract— *Radial inflow steam turbine technology is limited to turbocharger, waste heat recovery, gas turbine, solar, thermal, nuclear and geothermal power plant application up to 2-3 MW. The radial turbines are cost effective solution for power generation as compared to axial turbines. To understand whether radial turbine design can be scaled up to in the range of 20-30 MW, effort were made to carry out the design of 20-30 MW radial turbine using in-house developed program and commercial codes. Meridional flow path design is developed for 11 number of stages with stage loading coefficient in the range of 0.982-1.12 keeping in view to minimize the number of stages. The overall flow path length is achieved is approximately 2500 mm with total to static efficiency of 81%. A uniform tip clearance of 1 mm is assumed for the estimation of efficiency. MST and CFD analysis is carried out to understand the flow and loss behavior within the various components in the turbine. The flow path is modified iteratively to improve the flow behavior and associated losses in the rotating as well as stationary passage. Further, to understand the mechanical behavior of the rotor, Finite Element Analysis (FEA) of the rotor parts is carried out.*

Keywords-*Radial inflow steam turbine; meridional flow path design; MST; CFD; FEA;*

I. INTRODUCTION

Radial inflow turbines have a long history and exist before axial turbines by many years. In the lower ranges the radial inflow turbine competes with axial turbine in terms of manufacturing, cost and efficiency. Radial inflow turbine is compact in size because rotor can be manufactured as a one piece casting whereas axial rotor often demands separate blades and disk. Radial turbine accommodate large enthalpy drops in a single stage hence, radial turbine stage can deliver a greater specific power (power per unit mass flowrate of gas) than an equivalent axil stage because the fluid undergo a significant radius change in passing through the rotor where as in an axial stage the fluid enters and leaves in a predominantly axial direction with no change in radius and this may also imply smaller and fewer stages.

1.1 Working principle of radial inflow turbine

Figure 1. shows the fundamental components of a radial inflow turbine consists of Volute (0-1), Nozzle (1-3), Interspace (3-4), Rotor (4-6) and exhaust diffuser (6-7). The stations 2&5 are the throats of nozzle and rotor respectively. The flow enters the volute in a tangential direction from the external pipe and the flow is turned in the volute towards the rotor and its gives large swirl component of velocity & uniform velocity around the circumference. Some portion of the flow re - enters and mixes with the incoming flow at the tongue of the volute. The fluid further enters the nozzle vanes and it must be turned by an annular ring of vanes in order to give it tangential velocity and accelerated the flow before entering in to the rotor. The nozzle vanes have a large influence on the mass flow rate of a turbine because its depends on nozzle throat area. Again the flow the flow enters in the radially inward direction at the rotor inlet, turned in meridional plane and finally the flow is exit in axial direction. The work extraction is higher at the inlet region and lower at the rotor outlet region when the flow continues moves inwards and the radius. The blade speed and the tangential velocity decrease from inlet to exit and this implies a high blade loading. The fluid leaves the turbine rotor with a certain kinetic energy and entering in to the return channel. In this some amount fluid pressure is increases and its gives the high pressure fluid to the next stage nozzle inlet.

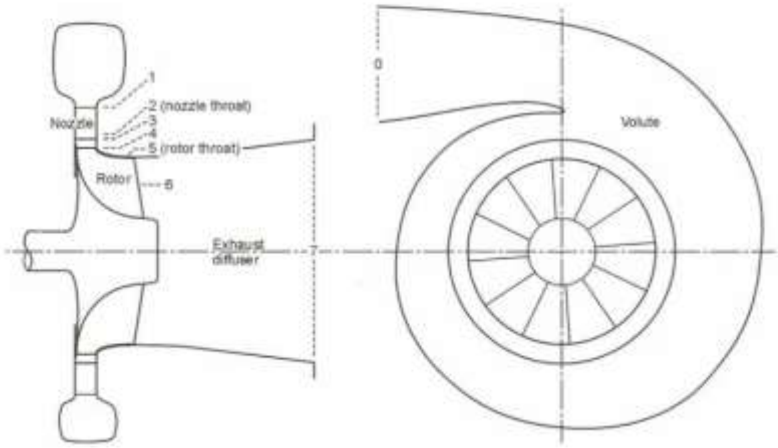


Figure 1. Components of radial inflow steam turbine

1.2 Design specification

The following design specifications has been taken for design of 20-30 radial inflow steam turbine as shown in Table 1.

Table 1. Design specifications

Inlet total pressure	Inlet total temperature	Exit pressure	Mass flowrate	Rotational speed
106.39 bar	540^0 C	1.0325 bar	33.7 Kg/s	7700 rpm

II. DESIGN THEORY

The rotor is primarily a work transfer device, and any consideration of its design will logically start with the Euler turbo machinery equation. This can be used, in combination with the geometry of the velocity triangles at inlet and exit, to derive the following expression for the work transfer per unit mass flow in terms of fluid and rotor velocities as follows.

$$W_x= (U_4^2 - U_6^2) + (C_4^2 - C_6^2) + (W_6^2 - W_4^2)] \text{ ------(1)}$$

This equation clearly shows the change in blade speed, absolute velocity and relative velocity makes contribution to the work output. The velocity triangles at the inlet and exit of the rotor as shown in fig. 2. The nozzle would be designed to accelerate the flow at inlet of the rotor in order to maximize the inlet absolute velocity C4. To minimize the absolute velocity at exit C6, the exit velocity triangle must be arranged properly. The exit radius r6 selection depends on the exit velocity U6 which is derived from W6 &C6. This is a difficult and there is no reasonable source for selecting favored exit radius.

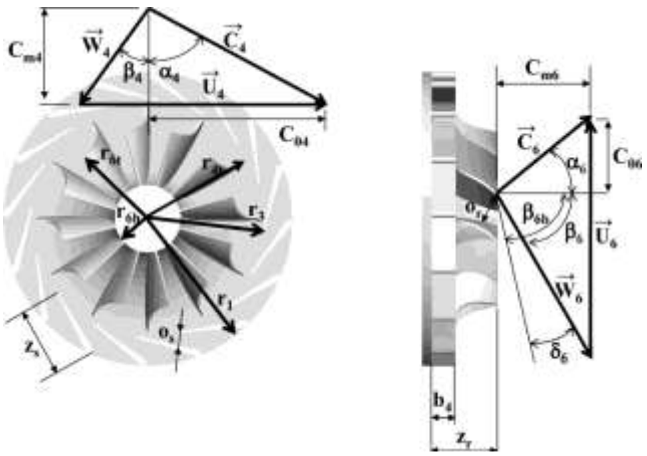


Figure 2. Nomenclature and Velocity triangles of Radial inflow turbine

The blade speed at rotor inlet U4 is selecting based on loading coefficients and it is defined as the ratio of change in enthalpy to the square of rotor inlet blade velocity.

$$\psi= (\Delta h/ (U_4^2)) \cong C_{\theta 4}/U_4 \text{ ------(2)}$$

Therefore, the tangential velocity, Cθ4 & blade velocity, U4 can be calculated after selecting the reasonable loading coefficient and Its select based on the experimental results. The exit meridional velocity Cm6 can be determined by selecting suitable flow coefficients, ϕ as follows.

$$\phi= (C_{m6}/U_4) \text{ -------(3)}$$

The values of stage loading coefficients and flow coefficients can be chosen together over a good total to static efficiency as 0.9-1.0 and 0.2-0.3 respectively. The meridional velocity ratio of the rotor is needed in order to establish the complete inlet and outlet velocity triangles. The meridional velocity ratio is defined as the ratio inlet radial velocity to the exit radial velocity of the rotor as follows.

$$\xi=C_{m4}/C_{m6} \text{------ (4)}$$

For design operating point, the value of meridional velocity ratio must be unity and adjusting this ratio while the inlet radius of the rotor was fixed. Based on the meridional velocities, the blade height and area at the inlet and exit is calculated by applying the continuity equation. The mass flowrate at inlet of the rotor as follows.

$$\dot{m}=2*\pi\ *r_4*b_4*\rho_4*C_{m4} \text{------(5)}$$

The meridional velocity Cm6 and absolute velocity C6 is equal when rotor is designed for zero swirl at the exit. The density at the exit can be calculated from the exit static pressure and enthalpy. By this density, the exit area and blade height is calculated by applying continuity equation. The rotor exit area, A6 as follows.

$$A_6=\dot{m}/(\rho_6*C_{m6}) \text{------(6)}$$

The rotor hub radius to inlet radius ratio is needed to calculate the exit blade height and the reasonable ratio is kept as 0.29-0.31 in order to avoid the overcrowding rotor blades.

$$\nu_h=r_{6h}/r_4 \text{ ------(7)}$$

Based on the hub radius, the rotor tip radius and blade height at the exit is calculated by the following formula.

$$r_{6t}=\sqrt{((A_6/\pi+r_{6h}^2))} \text{-------(8)}$$

Finally, the geometry of the turbine and velocity triangles are obtained based on the above design theory. The another approach is also used for the performance of a radial inflow turbine by specific speed and velocity ratio. The specific speed is calculated by the following formula.

$$Ns=((2*\pi*N)/60) * (\sqrt{(m/\rho)}/\Delta h^{0.75}) \text{-------(9)}$$

The velocity ratio is the ratio of inlet blade velocity to the spouting velocity and is calculated by the following formula.

$$\nu= (U_4/C_S) \text{ -------(10)}$$

The optimum specific speed and velocity ratio corresponding to total to static efficiency lies between 0.4-0.6 and 0.6-0.7 respectively. The specific speed is characterizing the turbine in terms of their type & size.

III. MEANLINE DESIGN OF 20-30 MW RADIAL INFLOW STEAM TURBINE

The meanline design of radial inflow steam turbine for the given specification is carried out to supplement or verify the claims made in open literature, regarding non-availability of such higher ratings radial turbine. The expansion lines are fixed by varying the heat drop across stage and studying the effect of specific speed, velocity ratio, loading co-efficient & flow coefficient in order to obtain feasible geometrical configuration. Total heat drop is varied from 830 to 1052 kJ/kg to realize the power in the range of 20-30 MW and this amount of heat is drop into 11 stages. An excel sheet program has been developed in-house to estimate the effect of various mean line design and performance parameters. IF97 steam routines is integrated in the excel sheet and used for all steam properties calculation. The geometry and flow parameters of the turbine is calculated by assuming the enthalpy drop, pressure ratio, loading coefficient, flow coefficient, meridional velocity ratio, hub to tip ratio, incidence, deviation angle exit swirl velocioty are 71.3 KJ/kg, 1.3, 0.9, 0.2, 1, 0.3, -20 degree, -5 degree and zero respectively in order to get the good efficiency. The calculated values of power, specific Speed and velocity Ratio of first stage are 2333 kW, 0.196 and 0.65 respectively; and The geometry and flow parameters of the first stage as shown in Table 2

Table 2. Calculated parameters of a radial inflow turbine for single stage.

Parameters	Units	Values	Parameters	Units	Values
Inlet Tip Radius, r_4	m	0.345	Inlet Meridional velocity, C_{m4}	m/s	55.7
Inlet Blade height, b_4	m	0.009	Inlet ab.Flow Velocity, C_4	m/s	257.1
Hub radius at outlet, r_{6h}	m	0.104	Inlet Relative velocity, W_4	m/s	62.36
Exit Blade height, b_6	m	0.032	Inlet Tangential Velocity, $C_{\theta 4}$	m/s	250.9
Inlet Blade Angle, β_{b4}	deg	6.56	Exit Blade Velocity, U_6	m/s	97.5
Exit Blade Angle, β_{b6}	deg	-65.2	Exit Relative Velocity, W_6	m/s	112.4
Inlet ab.Flow Angle, α_4	deg	77.4	Inlet Relative Flow Angle, β_4	deg	-26.5
Inlet Blade Velocity, U_4	m/s	278.8	Exit Relative Flow Angle, β_6	deg	-60.24

For accurate solutions, the Generic commercial codes are developed for carrying out the meanline design. These codes perform the design is fast and gives accurate solution. The codes are capable designing out single stage at time. The meanline (1D) programs are often used to optimize the most basic parameters of a machine based on design goals and constraints. The first stage is modelled with symmetric volute and 0.5% pressure drop is assumed across the throat. In the design of all the stages only profile and tip leakage loss are accounted. The secondary losses and windage losses are not included because of non-availability of such models inside the codes. The tip clearance is kept 1 mm for all the stages. The important geometrical parameters assumed for designing of all the 11 stages is shown in Table 3.

Table 3. Important geometrical parameters for 11 stage radial turbine

Ratio of Inlet radius to exit radius of nozzle	Ratio of nozzle exit radius to rotor inlet	Number of stator vanes	Ratio of exit hub radius to inlet radius of rotor	Number of rotor vanes
1.20-1.26	1.02-1.07	15	0.28-0.31	13

Multiple design iteration is carried out by changing the pressure ratio, the inlet and outlet tip radius of rotor & nozzle, blade inlet & outlet height and blade inlet & outlet angles to removing the choke and optimizing the velocity triangle.

3.2 Inputs (Geometrical parameters)

The pressure ratio is changes from stage to stage for maintaining the constant enthalpy drop and constant power at each stage. The first stage and last stage pressure ratio is 1.3 and 1.95 respectively. The losses are not incorporated in this commercial codes hence we assume 5% exit pressure is drop across each stage. The pressure ratio of all 11 stages as shown in fig 3.The rotor hub, rotor inlet tip, nozzle inlet and nozzle exit tip radius is kept constant because need to maintain uniform blade velocity. The last stage is difficult to design with uniform velocity. The last stage is demand the increased radius in order to avoiding the crowding of the inlet and exit blade heights of the rotor. Hence, the nozzle inlet & exit radius and hub radius is also increased. The Nozzle inlet & exit radius and rotor inlet & exit hub radius as shown in fig 4.

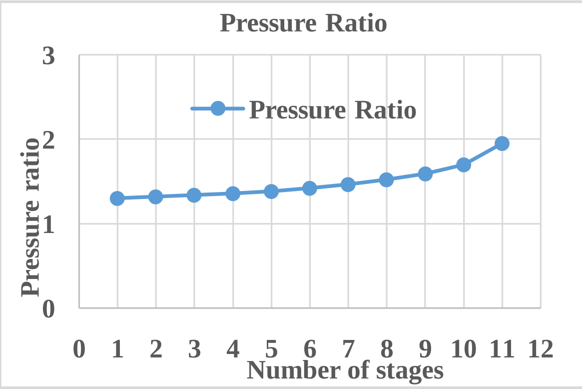


Figure 3. Pressure ratio of all 11 stages

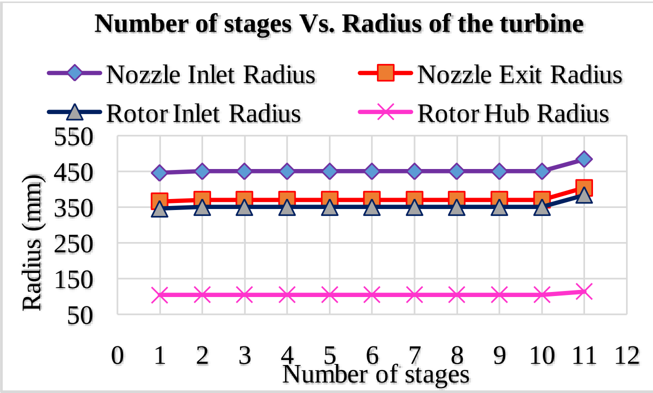


Figure 4. Radius of the nozzle and rotor for all 11 stages

For any multistage turbine is demand the increased tip radius when the fluid is flowing from stage to stage. Pressure and temperature is drop from stage to stage so that we need to increase the passage area. Based on design requirements maintains the constant radius at each stage and increases the inlet and exit blade heights at each stage. The inlet and exit blade angle is change until we get the required optimum incidence, deviation angle andoptimum velocity triangle. The inlet and exit blade heights and angles of all 11 stages as shown in fig 5.

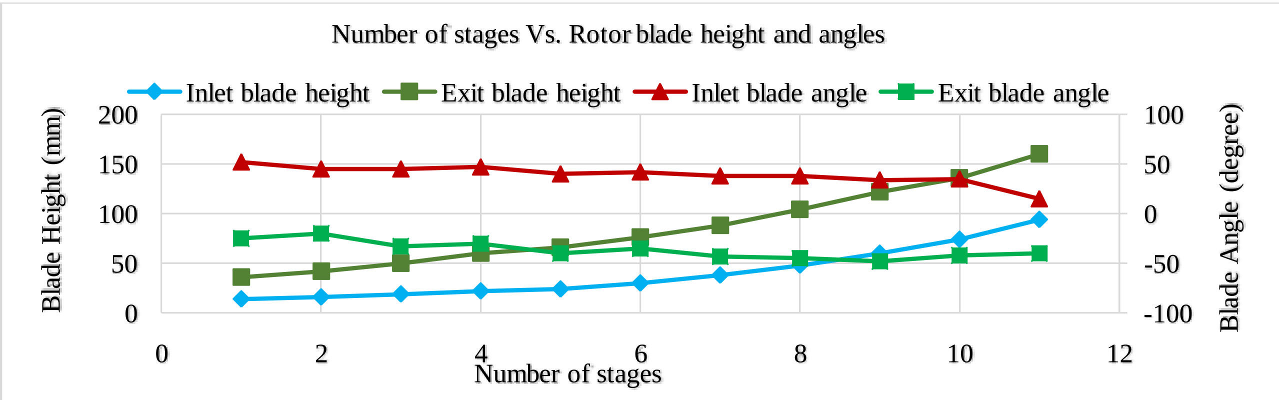


Figure 5. Rotor blade heights and angles for all 11 stages

3.2 Outputs (flow parameters)

The inlet and exit blade velocity is depending on the inlet and exit tip radius of the rotor. The inlet blade velocity is constant for all the stages excepting the last stage. The blade velocity at exit is increase from stage to stage because the exit blade height is increase from stage to stage. Meridional velocity ware kept on increasing at the rate of 1.2- 1.5% for passing the flow smoothly across the turbine. The blade and meridional velocity of the rotor as shown in figure 6. The average power, velocity ratio, reaction and total to static efficiency at each stage is 2.63 MW, 0.67, 0.45 and 86 % respectively.

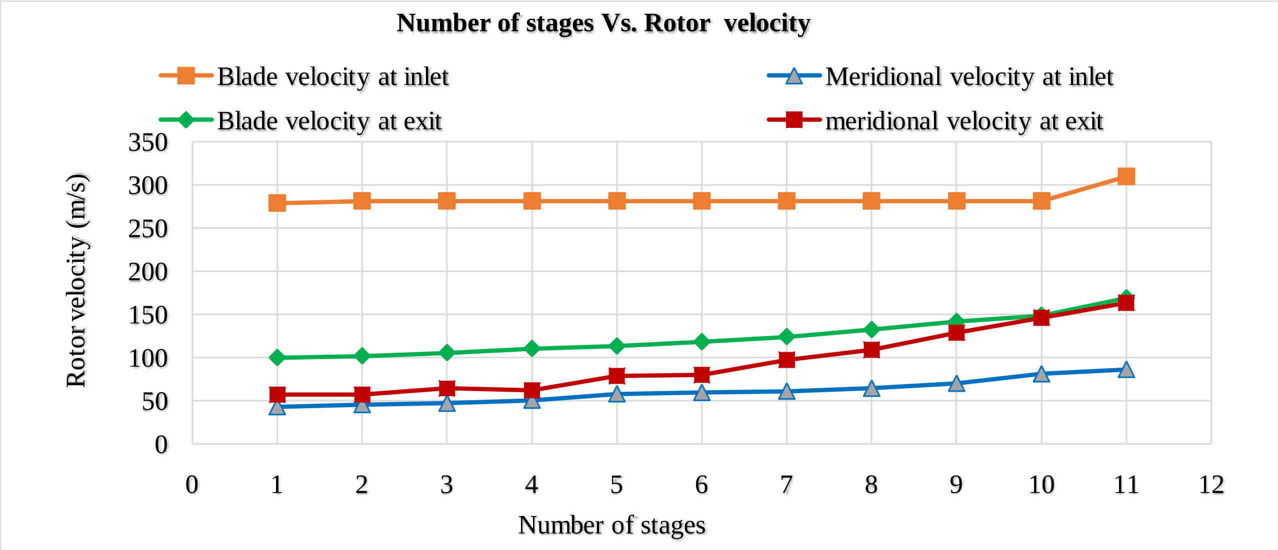


Figure 6. Blade velocity and meridional velocity for all 11 stages

IV. MULTISTAGE FLOW PATH DESIGN

The flow path commercial codes are developed for carring out multistage flow path design. Once the meanline design has been completed, we can proceed to a complete flow path design. The meanline programs allow to transfer meanline design information directly into flow path design codes. The flow path codes capable to assembling all the stages and also generates an initial 3D design that uses all of the geometric parameters from the meanline. The flow path is further modified due to uniform flow across the passage. The flow parameters are obtained by using MST and CFD analysis.

4.1 Assembling all the 11 stages

Import the stages whatever the stages is designed in the meanline codes and assemble all the stages. The return channel is added between the downstream of previous stage and upstream of next stage. This channels designed based on the rotor exit passge area. An expansion ratio is maintained closer to unity, the return channel passage area is kept same as the rotor exit passge area for smooth flow. the passage is aligned to 7-12 degs towards inlet direction to further reduce the flow path length. The inter stage gap is kept more than 28 mm in order to provide sufficient diaphragm thickness.

4.2 Making geometric adjustments

The geometric adjustments of the design can be made by adjusting plot points (control points & junction points) and This plot points are adjusted by the following two methods.

By dragging control points: Using this method the geometric adjustments made quickly without specifying exact point values and we can immediately see the effects of edits in the graph window.

By specifying Junction points: Using this method the geometric adjustments made by specifying the Z & R values of the junction points in the plot table and also easily made the Z or R values of two points equal, and reuse junction point values by copying and pasting in the plot table. The flow path of multistage radial inflow turbine as shown in figure 7.

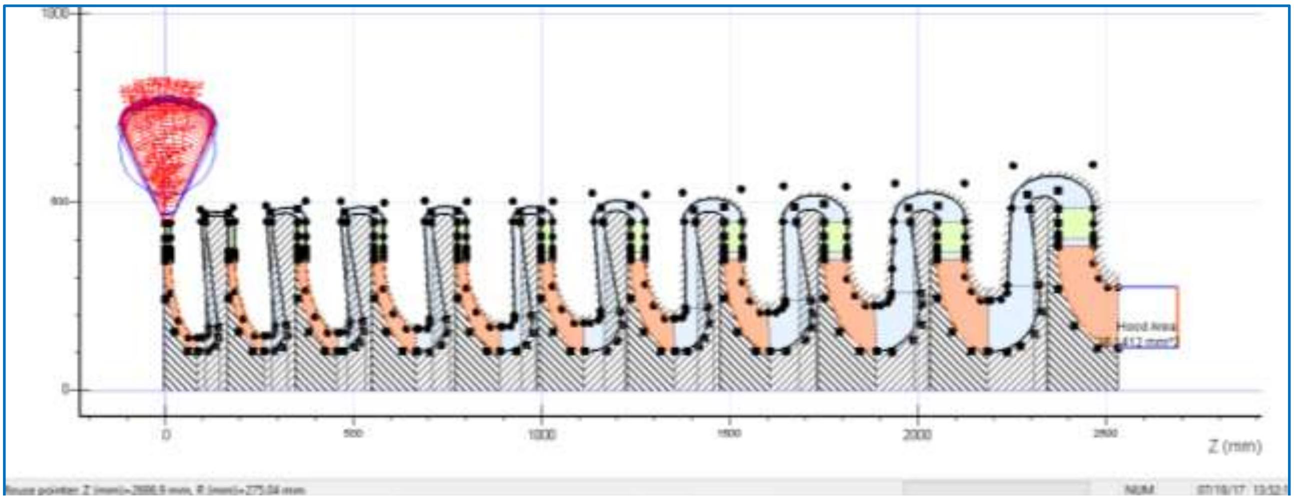


Figure 7. Flow path of 20 - 30 MW multistage radial inflow turbine

4.3 MST (Multi Stream Tube) Analysis

MST analysis is used to determine the fluid dynamic loading on the blade surface and velocity distribution from hub to shroud along the meridional flow path. MST analysis is a quasi-three dimensional inviscid flow analysis that uses streamline curvature method. This method use assumptions of linear velocity variation from blade to blade in order to calculate the flow field on the blades. MST permit an overall loss to be executed on the solution as an alternative to a two zone model and this analysis breaks up the passage into 7-11 number of streamlines from hub to shroud. The following parameters required to perform MST analysis.

MST parameters: In this number of streamnlines is taken as 11 and number of quasi orthogonals in side segment is selected as automatic and the number of iteration is taken as 45. Some other parameters like mass flow accuracy , damping factor, streamline curvature damping is taken deafult.

Blade loading: The mass flowrate, rotational speed, number of blades and reference velocity of nozzle and rotor for all stages is specified. Blade to blade calculation method is selected as normal method. The calculation is done normal to the passage area.

Fluid properties: The steam is selected as a fluid and also specified the inlet total temperature and total pressure of steam for all stages.

Flow modeling: In this, the primary zone exit deviation angle and loss coefficient is specified and also selected two zone model for all the stages. Also specified the fluid blockage at leading and trailing edge is zero.

MST stream lines: The MST streamlines for all stages as shown in figure 8.

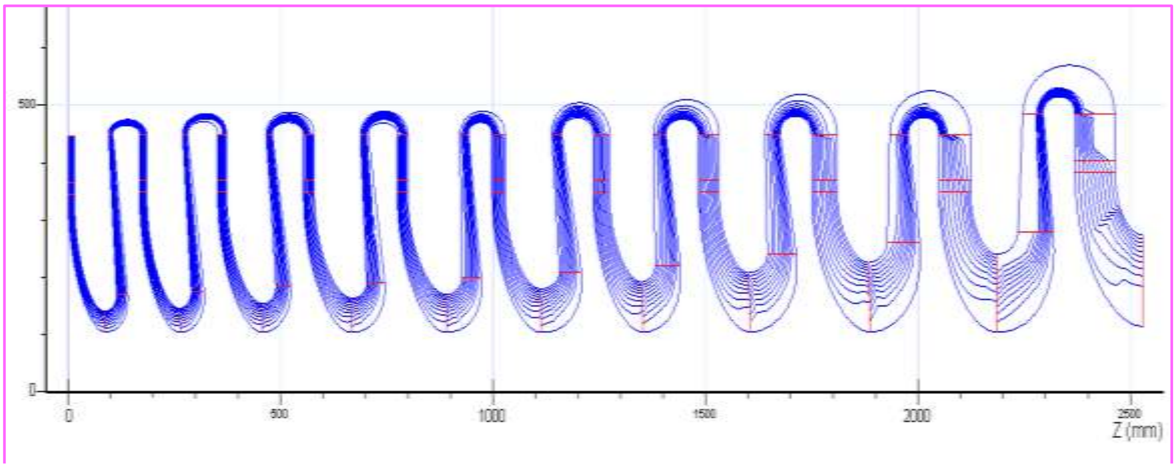


Figure 8. MST stream lines

Streamline calculation has been carried out for all the stages simultaneously. Almost all stages were modified to achieve the smooth streamline across the turbine. Width of the inlet nozzle is altered to remove the choke in various stages. The last two stage were given tremendous problem in convergence. These two stage width and blade angle modifies by approximately 30% form the single stage design obtained from the meanline program.

V. CFD ANALYSIS

Computational fluid dynamics (CFD) perhaps plays a better part in the design of turbine and the design of a modern turbine has been impossible without the help of CFD and this need has increased where the flow becomes agreeable to numerical estimate. CFD analyses are performed at the final stage of the design process, after run the design through a multi streamtube (MST) analysis. Computational Fluid Dynamics is an essential part of the blade design process. Hence, the CFD codes were developed for performing CFD analysis with quickly and easily. This codes provides three-dimensional analysis and full Navier-Stokes CFD tools with rapid run times that allows to create a large number of design iterations during a short period of time. Default values are supplied for the grid topology, boundary conditions, and solver settings. The codes have integrated Multi-block solver which is used to generate the grids.

5.1 Grid generation

For grid generation, the Multi-block solver can be used to generate OH-grids for nozzle blade row and OH-C grids for rotor blade row. The solver is divided the flow into the four number of blocks such as H1, H2, H3 and H4 around the circumference of the blade rows. The grid blocks and any number of cells per block is controlled easily and grid generation as shown in fig. 9. The element goodness is calculated based on the maximum skewness angle and aspect ratio. In our analysis, maximum skewness angle is 75 degrees and aspect ratio is 55 and these are optimum values.

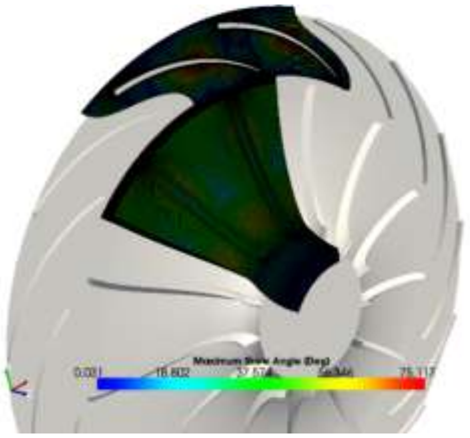


Figure 9. Grid generation

Table 4. Grid parameters

	Nozzle	Rotor
Topology	OH type	OH-C type
Solution	Full 3D	Full 3D
Passage	Single	Single
Grid type	Coarse	Coarse
Circumferential nodes	239	239
Blade to blade O block	5	5
Blade to blade H2 block	15	15
Blade to blade H4 block	15	15
Steam wise inlet cells	-	12
Steam wise exit cells	-	32

Single passage is considered for CFD analysis in order to reduce the time. Total 551 meridional nodes, 19 hub to shroud nodes, 4 clearance cells and 3 grid sections are taken from inlet of nozzle to rotor exit. Total number of nodes is 55464 and each grid nodes of nozzle and rotor is adjusted as shown in Table 4.

5.2 Boundary Conditions

Boundary conditions define the inlet and exit flow conditions that determine the flow field along with the passage geometry. The inlet boundary conditions and exit boundary conditions are applied on the inlet plane and exit plane of the CFD domain. The inlet total properties of the flow and the exit static pressure is specified. For designs with compressible flows, the exit static pressure is changes and this lead to different flow rates. The exit static pressure is used only to construct the radial pressure gradient using the simplified radial equilibrium equation.

Total pressure – 10634 kPa; Total temperature – 540^0C; Exit static pressure – 8440 kPa;

5.3 Solver settings

Before CFD run the solver settings such as type of solver, turbulence model, wall treatment options and implicit schemes is needed to in order to calculate the flow field.

An upwind scheme (AUSM) third order of accuracy is selected in the multiblockNavier stokes solver. The Smoother and more accurate solutions can be achieved by using the third order upwind schemes, particularly for low speed and low head machines. This preconditioning technology and deals with all flow regimes.

Log law hub / case shear stress is selected as wall treatment. The wall boundary condition is turbulent and the wall log law is used for the calculation of shear stress on the wall.
The SpalartAllmaras – 1 equation is selected as a turbulence models. This is best turbulence model and calculated the boundary layer growth near to the blade surface.

Gauss Seidel Implicit is selected as scheme. This scheme can produce a significantly faster convergence rate and also easily calculated where the flow problems are encountered. The main flow problem is faced at blade trailing edge where the average pressure is acting. Remaining all parameters are kept in default and finally in the CFD run the number of iterations are specified as 3000 to converge to within +- 2% of the design mass flowrate.

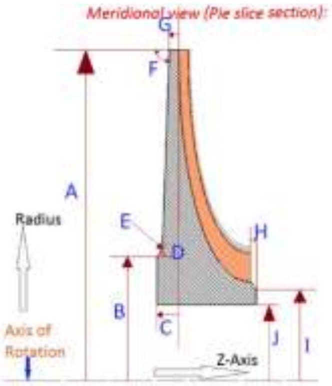
5.4 CFD plots: The mass flowrate - 39 kg/s, Bulk efficiency - 75 % and Pressure ratio - 1.225 is obtained after modified the blade geometry based on strength point of view. The flow passage is demand the increasing at the rate of 1.16 % after fine tuning the blade heights, blade angles and blade thickness.

VI. FEA ANALYSIS

Finite Element Analysis (FEA) is a numerical technique for computing the strength and behavior turbine components. It can be used to determine displacement and stresses. The computer is essential because of the more number of calculations needed to analyze a component. FEA Codes have been developed for stress analysis and flexible parametric modeling of radial inflow turbine. In design process, stress analysis is typically employed after defined the blade sections and stacking. This analysis the rotor is unshrouded, no balance ring, no balance notches and no diffuser. The following procedure provides an overview of the basic steps for performing stress analysis with FEA codes on radial turbine rotor.

6.1 Geometry generation

In geometry generation to specify dimensions for each element of a rotor such as backface, bore and blade fillet dimensions of the rotor. leading edge & trailing edge tip locations, blade width, blade angles, blade thickness, number of blades and flow path of the rotor are already designed in meanline and flow path commercial codes. The dimesions of each part of the rotor is changed until we get the required level of stress level, displacement and moment of inertia. The final dimensions of the rotor is shown in fig. 10. The circular type of fillet is used in between wheel and blade surfaces. The fillet radius of pressue & suction surfaces is taken at the hub and shroud is 5mm and 3.5 mm respectively.



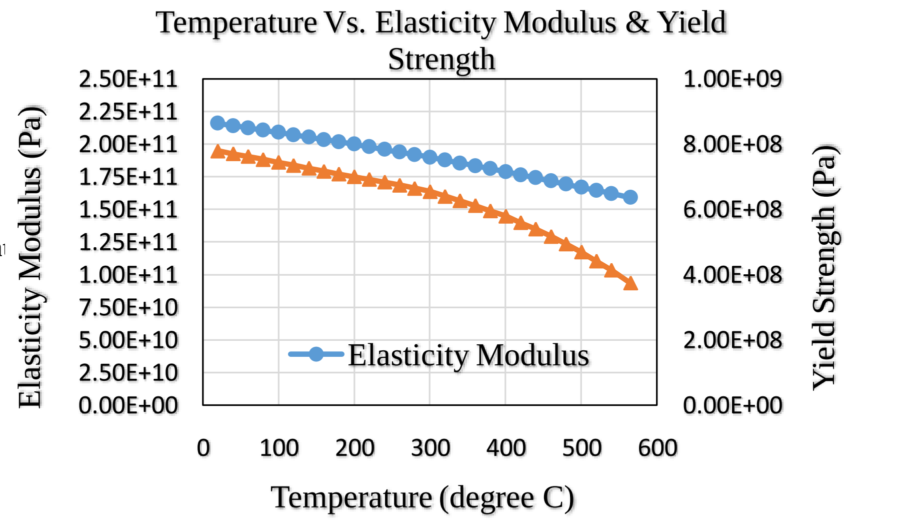
A-Back face zero thickness radius = 346 mm
B-Shoulder radius = 130 mm
C-Shoulder position = 23 mm
D-Shoulder angle = 45^0
E-Arc radius = 95 mm
F-Angle of back face = 86^0
G-Web thickness = 10 mm
H-Nose length = 6 mm
I-Nose radius = 95 mm
J-Bore radius = 80 mm

Figure 10. Dimensions of the pie slice section

6.2 Material properties

Material properties are determined by choosing material type and temperature. A database is used to store temperature-dependent properties for each of materials on the list. The material X19CrMoVNbN11-1 is high strength, corrosion resistance and high heat resistance. Hence, this material is used to manufacturing the rotor. The material properties such as density, poisson’s ratio, linear expansions coefficient, elasticity modulus and yield strength at different temperature as shown Table 5.

TABLE 5: Material properties

Material	**X19CrMoV NbN11-1**	**Chemical composition (%)** C-0.17, Si-0.35, Mn-0.70, P-0.035, S-0.035, Cr-10.6, Mo-0.60, Ni-0.40, V-0.25, Nb-0.30, N-0.07				
Mechanical Properties	Elasticity modulus at 540 ^{0}C	Yield Strength at 540 0C	Density	Poisson’s ratio	Linear Expansion Coefficient	
	Pa	Pa	(kg/m^3)	-	(1/K)	
	1.622E+11	414E+6	7700	0.29	0.0000125	
Elasticity modulus and Yield strength at different temperature						

6.3 Loads and constrain

Aero loads include Pressure and thermal loads are applied to the outer surfaces of the FEA model. Data is read from a pre-run Rapid load, MST or CFD analysis, or from flow field data file to define the pressure, temperature, velocity, and steam properties along the blade as well as hub & shroud contours. Due to the rear and front leakage of the steam, the user specified the values for pressure and temperature loads on the backface, shoulder and nose of the rotor. Inlet pressure & temperature is 10.634 MPa and 540 ^{0}C respectively. The outlet pressure & temperature is 8.725 MPa and 509 ^{0}C respectively as shown in fig. 11 The aero loads are defined as a function of radius i.e. the loads are specified by linear interpolating between inlet & outlet of the rotor.

Displacement constraints placed on the cut faces of the hub and shroud sectors. The displacement constraints is shown in fig. 12. Rotor shoulder is provided by spacer rings that are shrink fitted on both sides and there will restrain deformation of impeller in axial and tangential direction but allows to deform radial direction. Hence, rotor shoulder is constrained in both axial & tangential directions. The rotor bore is constrained in radial, axial and tangential directions because the rotor is fitted to shaft and also bore most closely simulate the actual impeller attachment. Hene, the deformation in all direction is zero. Boundary conditions of rotor nose allows the deformation in every direction.

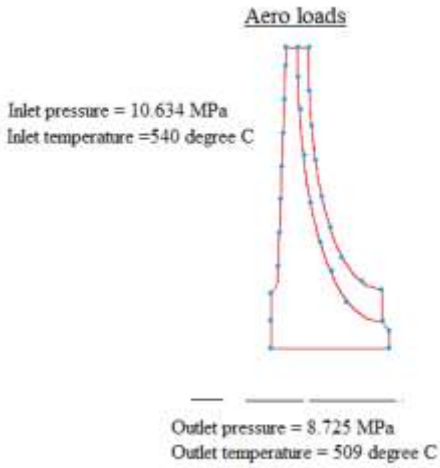


Figure 11. Aero loads

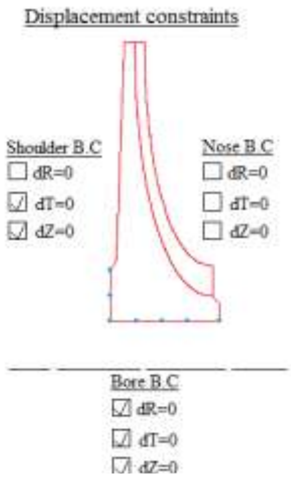


Figure 12. Displacements constraints

6.4 Meshing

In meshing, solid model of rotor classified into number of smaller elements. When increasing number of elements to get the accurate results. The type of mesh and mesh sizing of different parts of rotor such as blade leading edge, trailing edge, pressure side, suction side and hub fillet is important. Codes specify the different mesh size on different parts of the rotor. In our analysis the tetrahedral (four faces) type element is selected and meshing of the rotor component as shown in fig. 13. The meshing parameters as shown in table 6.The element goodness is calculated by aspect ratio and element angle & its 0.4 and 53.060 respectively. The aspect ratio (0-30) & element angle (30-60 degree) are optimum values.

Figure 13. Mesh generation

Table 6. Meshing parameters

Web nodes	7
Nose nodes	7
Pie layers	9
Nose extension nodes	5
Blade thickness	5 mm
Hub fillet nodes	9
Streamlines	35
Distribution from hub to shroud	32
Meridian nodes	30
Total Node Count	81609
Blade node count	19889
Hub node count	6310
Hub Fillet node count	5771
Bore node count	703
Backface node count	5913
Total Elements	51697

VII. RESULTS AND DISCUSSION

The overall performance of the 11 stages radial inflow steam turbine as shown in Table 7. and these results are obtained after developing the meanline design and flowpath design. In this the main contribution parameter is the power and 29.13 MW of power is achieved at reasonable efficiency and given design specification.

TABLE 7: Overall performance of the 11 stage radial inflow steam turbine

Sl. No	Parameter	Units	Values
1	Specific speed	-	0.18-0.6
2	Speed ratio	-	0.68664
3	Loading coefficients	-	1.06
4	Inlet total Temperature	^0C	540
5	Inlet total Pressure	kPa	10639.13
6	Exit static pressure	kPa	93.64
7	Meridional velocity at rotor exit	m/s	113
8	Rotational speed	rpm	7700
9	Mass flow rate	Kg/s	33.6
10	Av. Reaction	-	0.4449
11	Length of the turbine	m	2.53
12	Total to static efficiency	-	0.86873
13	Power	kW	29138.3

7.1 CFD Results

The CFD analysis is done on the first stage radial inflow turbine and these results are plotted along the flow passage. The velocity magnitude is the main parameter in order to identify the flow separation and secondary flows in the blade passages. The absolute velocity distribution is plotted in three dimensional flow with streamlines as shown in figure 14. The flow is reversed and separated in the nozzle middle portion at nearer to the blade surface due to the higher losses present in the nozzle portion which is shown in blue color. The absolute velocity is 250 m/s and 140 m/s at rotor inlet and outlet respectively. The flow separation is existing little bit at the rotor trailing edge due to blade is turned opposite to the rotation direction. The meridional velocity magnitude distribution is plotted in stream wise direction with vectors along the flow passage as shown in figure 15.

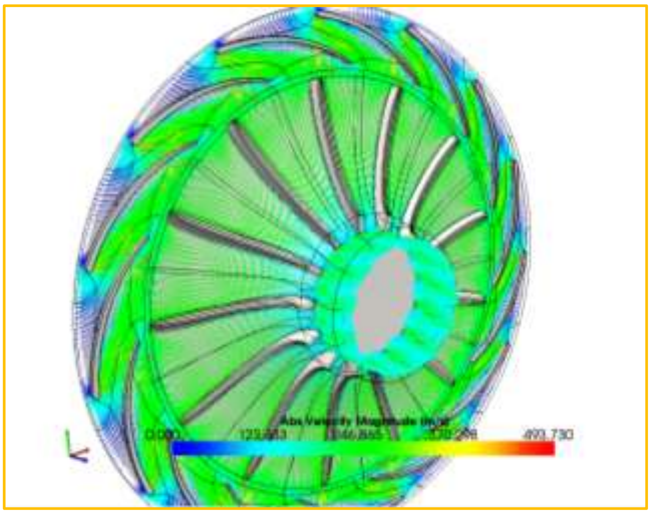


Figure 14. Absolute velocity

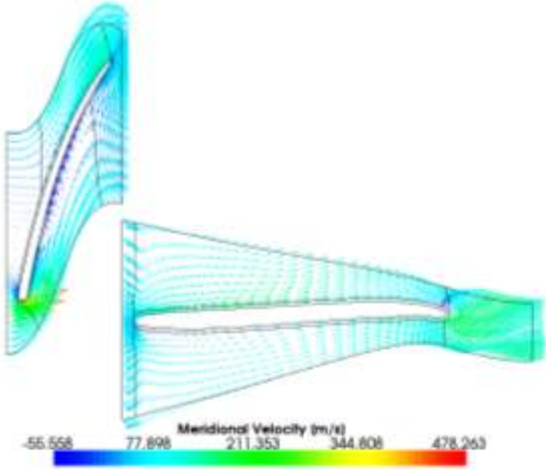


Figure 15. Meridional Velocity

The meridional velocity is 60 m/s and130 m/s & the relative velocity is 80 m/sand 140 m/s at rotor inlet and exit respectively. These velocities are recirculated at the nozzle middle portion at the suction side of the blade surface. Absolute total pressure, temperature and enthalpy distribution in blade to blade direction as shown in figure 16, 17 and 18 respectively. The absolute pressure is 10465 kPa and 8700 kPa, temperature is 538 ^{0}C and 510 ^{0}C & enthalpy is 3468000 J/kg and 3418000 J/kg at rotor inlet and exit respectively. The relative mach number is little bit higher at nozzle leading edge due to high pressure of steam is contact that portion as shown in figure 19. The Mach number is 0.19 and 0.3 at rotor inlet and exit respectively. In the nozzle and rotor, the mach number distribution is below the unity and this represents the working fluid is flowing below the sound velocity.

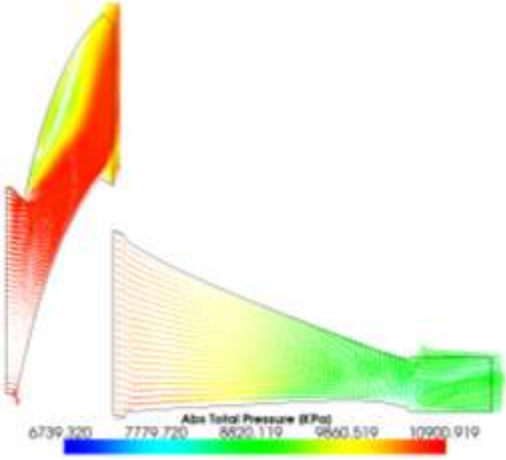


Figure 16. Absolute Total Pressure

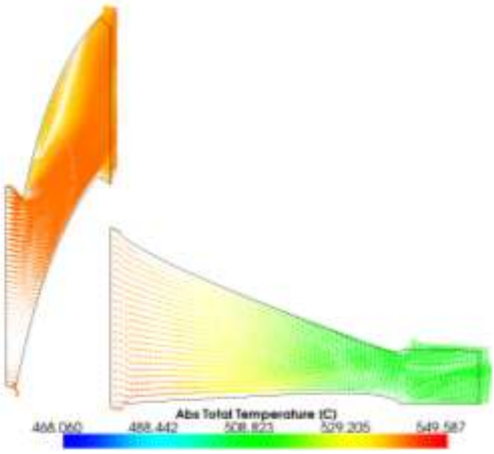


Figure 17. Absolute Total Temperature

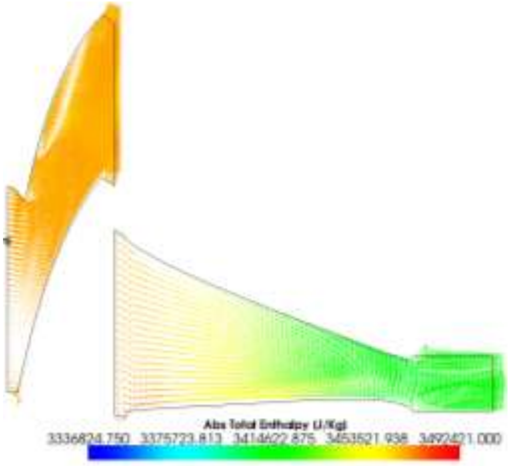


Figure 18. Absolute Total Enthalpy

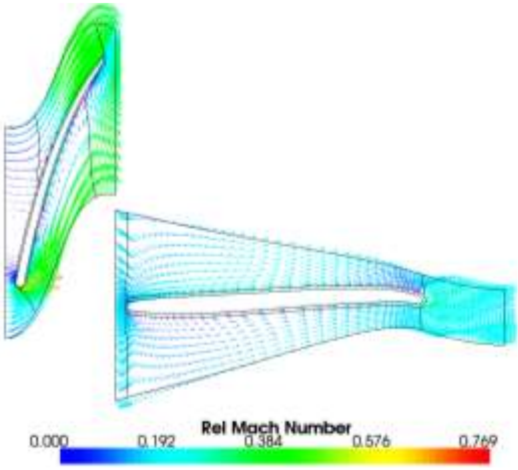


Figure19. Relative Mach Number

7.2 FEA Results

The following results such as von mises stress and deformation are obtained after performing the FEA analysis.The induced stress developed due to centrifugal force and aero loads in the rotor is lower than yield strength of material at 540 ^{0}C. Hence, rotor is safe under the loads.

- The induced stress is 322 MPa < yield strength of material is 414 MPa

The von mises stress at the inlet of rotor front face is high and decreasing upto the rotor exit. In blade portion, the stress at the inlet is low up to 30% portion and increasing upto trailing edge as shown in fig. 20. The maximum induced stress is developed at shoulder angle and shoulder radius. The stress is low at the inlet portion and increasing linearly to the shoulder angle but the shoulder is maintaining the constant stress..The deformation is high at the inlet portion of the rotor and decreasing towards exit. Deformation along radial direction is 0.01 mm, deformation in the z direction is 1.20 mm as shown in figure 21.

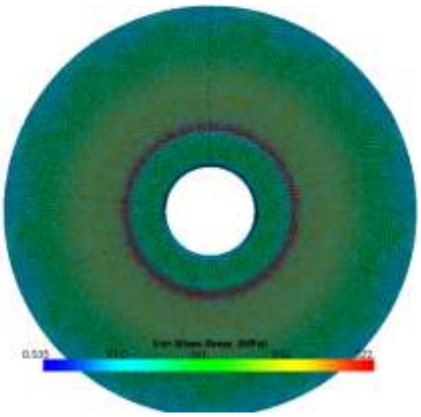


Figure 20.Vonmises Stress

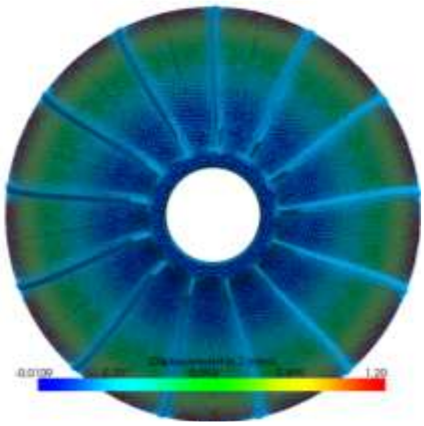


Figure21.Deformation in Z direction

VIII. Conclusions and future scope

The following conclusions have been drawn from the design of 20-30 MW radial inflow steam turbine for power generation application:

Extensive literature survey has been carried out to understand the state of art technology for radial steam turbine. The application of radial steam turbine is limited to waste heat recovery, geothermal power plant including binary cycle application up to 2-3 MW. In the lower ranges the radial turbine technology competes with axial technology in terms of manufacturing, cost and efficiency.

As on date, there is no radial steam turbine in the range of 20-30 MW seems to be available nor operating, nor any information is available on the globe. Maximum capacity of 7.4 MW radial inflow gas turbine for gas processing plant (yr.-1972) has been reported. The existing status for the same is unknown. For other radial gas turbine applications such turbocharging, gas expanding, cruise ships, tankers, food processing, textiles and biofuel etc. maximum rating is found to be 2.1 MW.

Meanline design of 20-30 MW radial inflow steam turbine has been carried out with in-house developed excel sheet program and commercial code for 11 number of individual radial stages. These stages have been designed with higher loading coefficient in the range of 0.982-1.12 keeping in view of minimum number of stages to reduce the size and cost. For, this case the feasible axial length for each stage from aerodynamic point of view is approximately 85-160 mm. The total to static efficiency is found to be 86% with tip clearance of 1 mm. This efficiency will decrease further by approx. 1.5-2% due to various losses like windage, ducting & wetness losses which is not accounted in the design calculation at this stage due to non-availability of such models in the codes.

The flow path design is carried out for assembling all the stages with return channels. The rotor length, return channel geometry and blade shape is fine-tuned by control points in order to maintaining smooth flow and reduce the total size of the turbine. The length of the turbine is obtained as 2.53 m and finally generated the three dimensional flow path.

The CFD analysis is carried out for the first stage of turbine and estimated the flow passage, velocity parameters and thermodynamic parameters. The flow separation and secondary flows are presented in the nozzle middle portion nearer to the blade suction surface due to non-availability of nozzle loss correlation in the open literature.

Finally, the FEA analysis is carried out for the first stage rotor and estimated the vonmises stress and deformation of the rotor. The maximum stress is acting at the shoulder radius and the axial deformation is occurred at the rotor tip. These are below the yield strength of the material.

Future Scope

The meanline design and flow path design is carried out for all the 11 stages of radial inflow steam turbine. The CFD analysis is done on the first stage of the turbine. However, its require for all the stages in order to complete the entire design. Hence an investigation would be conducted to evaluate the flow field in all the stages individually and as well as to observe the effect of the complete radial inflow steam turbine. Based on this results the meanline, flow path and blade design will be modified for all the stages.

The FEA analysis is carried out for the rotor of the first stage in order to estimate the stresses and deformation in the blade because the high pressure and temperature of the working fluid is moving with high speeds in the rotor. If a robust structure of the turbine is required, then the FEA analysis will be carried out for all the stages and also complete the entire design.

The loss correlations such as incidence, tip clearance and exit energy losses are only incorporated in the design due to the limited understanding of losses. The remaining loss correlation will be studied and these losses are taken into account in order to improve the accuracy of the complete design. The tip clearance losses are important in the design among all the losses. There is no loss correlation available in the open literature for higher mass flowrate. Hence a separated correlation will be established by varying tip clearance with respect to efficiency by using CFD analysis seriously in the future work.

REFERENCES

[1] N.C.Baines, "Radial Turbomachines", *Longman Scientific & Technical, John Wiley & Sons, 1990*

[2] T.Schobeiri, “Optimum Design Of A Low-Pressure, Double Inflow, Radial Steam Turbine for Open-Cycle Ocean Thermal Energy Conversion” *34th International Gas Turbine And Aeroengine Congress And Exhibition, 1990,Vol-112, Pp 71-77.*

[3] Hans D. Linhardt, " The Large Power-Output Radial Inflow Turbine", *Proceedings Of The Third Turbomachinery Symposium, 1994, Pp. 8-16.*

[4] Y. Ribaud & C. Mischel, “Study And Experiments of a Small Radial Turbine for Auxiliary Power Units”, *Presented At The International Gas Turbine Conference And Exhibit, 1986.*

[5] Harold E. Rohlik And Milton G. Kofskey, “Recent Radial Turbine Research At The Nasa Lewis Research Center”, *Gas Turbine Power And Fluids Engineering Divisions Conference Sponsored By The American Society Of Mechanical Engineers, 1972.*

[6] A. Whitfield, “The Preliminary Design of Radial Inflow Turbines”, *Presented at the Gas Turbine and Aeroengine Congress And Exposition, 1989.*

[7] H. D. Linhardt, “Process Application and Design of Large Power Output Turboexpanders”, *Presented at the Petroleum Mechanical Engineering Conference with Pressure Vessels And Piping Conference, 1972.*

[8] Jan F. Suhrmann, Dieter Peitsch, Marc Gugau, Tom Heuer & Uwe Tomm, “Validation and Development of Loss Models for Small Size Radial Turbines”, *Proceedings of ASME Turbo Expo 2010.*

[9] A. Hamed, E. Baskharone & W. Tabakoff, "A Flow Study in Radial Inflow Turbine Scroll-Nozzle Assembly", *Transactions Of The ASME, 1978, Vol.-100, Pp 30-36.*

[10] F. S. Bhinder, "A Contribution To Designing A Radial Flow Gas Turbine Stage", *Presented at the Gas Turbine and Aeroengine Congress And Exposition,1990.*

[11] Kerry L. Mclallin & Jeffrey E. Haas, "Experimental Performance And Analysis of 15.04-Centimeter-Tip-Diameter,Radial-Inflow Turbine With Work Factor Of 1.126 And Thick Blading", *National Aeronautics and Space Administration,1980.*

[12] J.R. Serrano, H. Climent, P. Fajardo, M.A. Reyes-Belmonte & Fabrice Vidal, "Contribution to the Understanding of Cold Pulsating Flow Influence in the Efficiency Of Small Radial Turbines For Turbochargers" , *Proceedings of ASME Turbo Expo 2012.*