Research Article *ISSN: 2319-507X*
Mokhadkar PA, IJPRET, 2013; Volume 1(8): 187-198 *IJPRET*

INTERNATIONAL JOURNAL OF PURE AND APPLIED RESEARCH IN ENGINEERING AND TECHNOLOGY

A PATH FOR HORIZING YOUR INNOVATIVE WORK

NUMERICAL STUDY OF DYNAMIC BEHAVIOR OF STEAM TURBINE

PROF. P. A. MOKHADKAR, PROF. A. A. SHAHADE

Department of Mechanical Engineering, IBSS College of Engineering Amravati, Maharashtra India.

Accepted Date:

27/02/2013

Publish Date:

01/04/2013

Keywords

Steam Turbine

CFD Analysis

Ultra Wide Band

Planar Antenna

Corresponding Author

Prof. P. A. Mokhadkar

Abstract

In this project work computational fluid dynamics (CFD) analysis tends to utilize for the study of steam flow path in the last stage of rotor-stator disc cavities in steam turbine. With the aid of CFD analysis the effect of twisting of blade on the velocity and the pressure drop for particular turbine can be analyzed. In this thesis the main functional area is to study the flow over the LP steam turbine blade. This report will introduce a CFD analysis methodology for steam flow studies, which makes use of an interface separating the stator rotor blade for the analysis. Series of numerical computations of full Navier-Stokes equations coupled with an appropriate turbulence model will be solved for rotor stator models to identify factors that affect flow path over the blade. Thus obtain the correlations and record observations relevant to the performed computational analysis. Both 3D models featured detailed CAD geometry of a periodic turbine blade passage sector and its associated upwind rotor-stator disc cavity. The analysis will yield results that provide the comparative studies for changing the flow parameters, but, in the absence of conclusive empirical results and experimental verification. The computational solutions will allow the results providing the effect of change in inlet flow angle on the pressure drop at last stage of LP turbine. With this study the main emphasis is to study the flow effect on the blade geometry of the steam turbine. The analysis provides the results like velocity distribution at the last stage of LP steam turbine, the pressure drop and how the phase change occurs at the last stage of steam turbine.

INTRODUCTION

Steam turbine is one of the most versatile and oldest prime mover technologies still in general production. Power generation using steam turbines has been in use for about 100 years, when they replaced reciprocating steam engines due to their higher efficiencies and lower costs.

Conventional steam turbine power plants generate most of the electricity produced in the United States. The capacity of steam turbines can range from 50 kW to thousands MWs for large utility power plants. Steam turbines are widely used for combined heat and power (CHP) applications.

Unlike gas turbine and reciprocating engine CHP systems where heat is a byproduct of power generation, steam turbines normally generate electricity as a byproduct of heat (steam) generation. A steam turbine is captive to a separate heat source and does not directly convert fuel to electric energy. The energy is transferred from the boiler to the turbine through high pressure steam that in turn powers the turbine and generator. This separation of functions enables steam turbines to operate with an enormous variety of fuels, from natural gas to solid waste, including all types of coal, wood, wood waste, and agricultural byproducts (sugar cane waste, fruit pits, and rice hulls).

In CHP applications, steam at lower pressure is extracted from the steam turbine and used directly or is converted to other forms of thermal energy. Steam turbines offer a wide array of designs and complexity to match the desired application and performance specifications. Steam turbines for utility service may have several pressure casings and elaborate design features, all designed to maximize the efficiency of the power plant.

For industrial applications, steam turbines are generally of simpler single casing design and less complicated for reliability and cost reasons. CHP can be adapted to both utility and industrial steam turbine designs.

1.1 OBJECTIVE

To use CFD Software computer code, for following:

- The design and analysis of steam turbine flow path for individual stages,

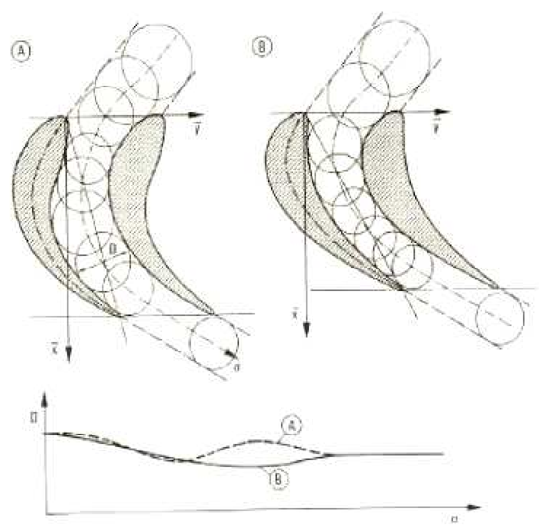
- Evaluation of proposed unit upgrades and load range change.
- The analysis of units for repowering to a combined cycle consideration.

Steam turbine technology achievements have been made in the areas of steam path, frame architecture, component design and material development. Pursuing bespoke solutions by a holistic approach involving mechanical, aerodynamic and manufacturing aspects has allowed offering industry-leading steam turbine designs.

Development and introduction of three-dimensional (3D) blading allowed many of the traditional stage efficiency limitations to be overcome. With 3Dblading, the blade profile (cross-section at a given height) and blade shape ('stacking' of profiles between hub and tip) are optimized with respect to reducing steam flow causing secondary losses.

4.4. TURBINE BLADE GEOMETRY

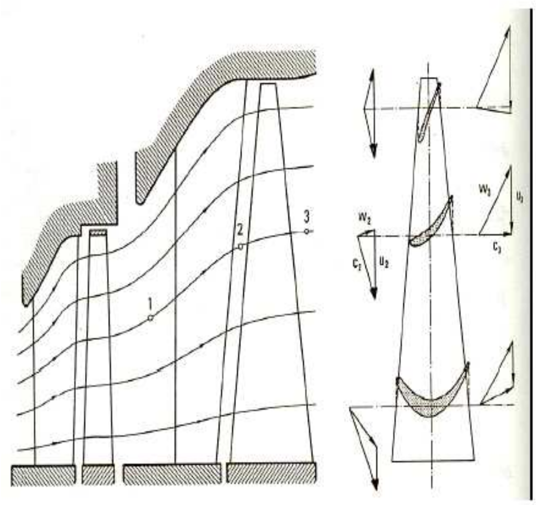
The latest generation of blades has twisted profiles that follow the incident steam direction, since orientation of the steam flow to the blades varies between hub and tip. This ensures the optimum effect of loading along the entire length of the blade. Turbine efficiency is determined primarily by the blading, because this is the component that actually transforms the available energy of the steam into useful work. Research on blade design improvement was started on turbines for fossil steam power application many years ago. The gained results have been taken over to the nuclear steam turbine design to reach maximum thermo dynamical benefit.



- After determining the velocity triangles the shape of the blade profiles can be determined

• The blade geometry yields the flow passage (e.g. to be determined by tangent circles) and by this the flow velocity distribution around the blade

• Although in- and outflow cross sections



are identical the change in cross section throughout the passage might different task of the designer to find optimum Blade design, nowadays assessed by means of CFD Advanced aft-loaded profiles have streamlines over the blade with varying cross sectionbeen designed that make the passage loading more uniform and produce a smaller pressure difference at the leading edge between the concave side and the convex side of the blades. This result in reduced secondary flow losses at blade hub and tip.

As far as project is concerned the aim is to study the flow pattern on the blade. The results can be used to study the various parameters to optimize the flow field, degree of reaction, flow angle and mass flow distribution on the blade height. In order to achieve the mentioned aim two case studies had been carried out:

- Flow path computation for twisted blade.
- Flow path computation for untwisted blade, in order to study the effect of velocity, pressure distribution on the blade.

The modeling details of blade are as follows:

For turbine blade modelling, PRO/E WILDFIRE Version 05 has been used. In order to generate the blade model following are the co-ordinate points:

Table 4.1Co-ordinate point for tip and root section of blade

Coordinates	Tip		Root	
	X	Y	X	Y
1	-11.43	-49.632	-53.34	-46.533
2	-8.966	-28.88	-42.805	-23.622
3	-6.528	-13.056	-32.258	-24.384
4	-4.064	0.584	-21.717	8.153
5	-1.6	10.77	-11.176	15.037
6	+0.864	18.085	-0.635	18.136
7	+ 3.302	23.419	9.906	17.755

8	+5.766	28.651	7.74	13.919
9	8.23	33.223	30.988	7.01
10	10.668	37.338	41.529	-2.692
11	12.065	39.573	47.117	-8.89
12	13.1318	40.945	52.07	14.91
13	-11.43	-53.462	-53.34	-48.743
14	-8.966	-45.06	-42.799	-30.919
15	-6.528	-36.627	-32.258	-17.17
16	-4.064	-28.219	-21.717	-8.28
17	-1.6	-19.812	-11.176	-3.073
18	0.864	-11.379	-0.635	-0.813
19	3.302	-2.972	9.906	-0.813
20	5.766	5.461	20.447	-2.819
21	8.23	13.868	30.988	-6.655
22	10.668	22.276	39.319	-12.192
23	12.09	28.169	47.117	-15.723
24	13.132	34.087	52.07	-19.253
25	0	50.038	0	-46.99

4.7 STEPS FOR BLADE MODEL GENERATION

1. Root Cross section for Moving blade

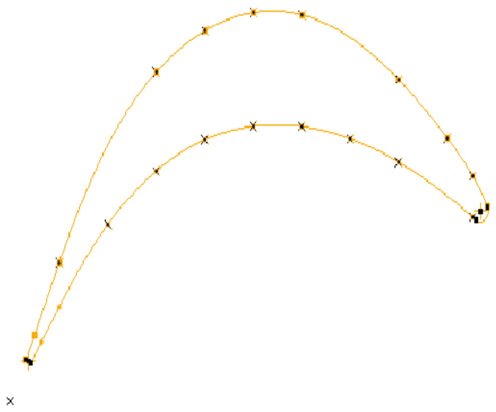


Fig 4.6 Root Cross- Section

2. Tip Cross Section for Moving Blade

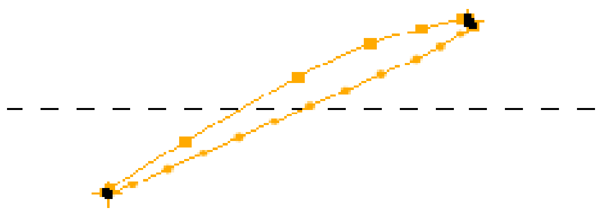


Fig Tip Cross-Section

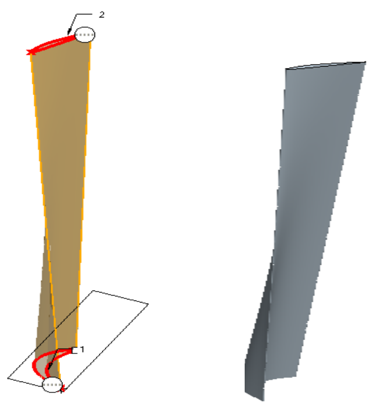
3. by Boundary Blend Option

PATH: Main Menu Manager-Insert – Boundary Blend

Fig 4.8 Moving blade model

4. By Copy option second moving blade is generated

Modeling of stationery blade is done in similar way as that of moving.



Here the length of Blade = 0.676 m

And the rotor Dia =2.132 m

5.By Boundary Blend Option

PATH : Main Menu Manager-Insert – Boundary Blend..

After the modeling work is over the geometry get imported into the ANSYS WORKBENCH for mesh generation.

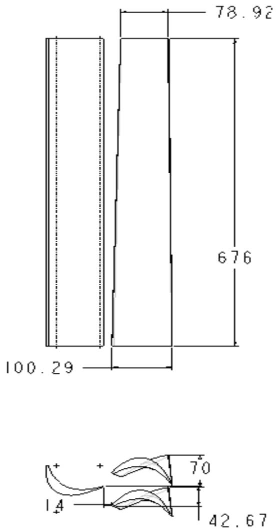


Fig Blade Geometry

Physical Timestep

This option allows a fixed timestep size to be used for the selected equations over the entire flow domain. For advection dominated flows the physical timestep size should be some fraction of a length scale divided by a velocity scale. A good approximation is the Dynamical Time for the flow. This is the time taken for a point in the flow to make its way through the fluid domain. For many simulations a reasonable estimate is easy to make based on the length of the fluid domain and the mean velocity.

Table

Advection Scheme	High Resolution
Physical Timescale	0.002 Sec
Convergence Criteria	RMS(1e-4)

The various plots generated during the solver steps are:

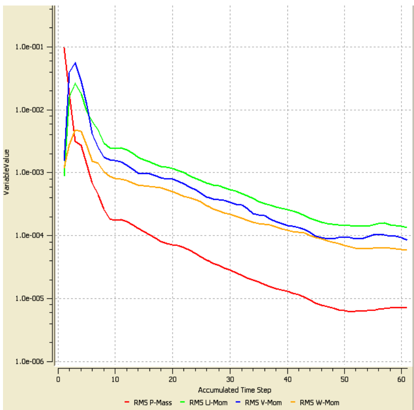


Fig MASS AND MOMENTUM GRAPH

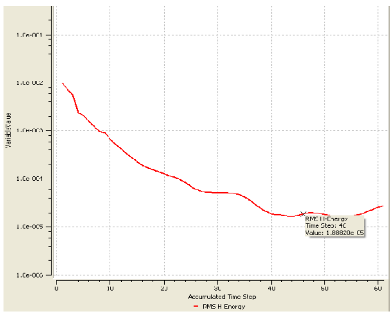


Fig 7.7 Heat Transfer plot

ISSN: 2319-507X
IJPRET

7.7 CFD RESULTS

ANSYS CFX-Post is a flexible state-of-the-art post-processor for ANSYS CFX and other CFX products. It is designed to allow easy visualization and quantitative post-processing of the results of CFD simulations. The ANSYS CFX-Post processor supports a variety of graphical and geometric objects which are used to create post-processing plots, and define locations for quantitative calculation.

The results visualized in post processor are:

The velocity and pressure distribution can be analyzed with the different colour. We cansay that velocity increases and pressure decreases after passing through the rotor domain of the blade.

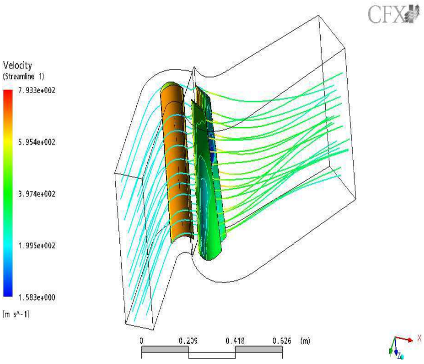


Fig 7.8 Velocity Streamlines along the flow domain

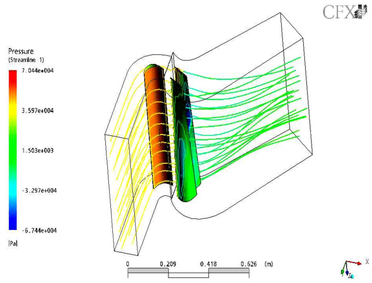


Fig 7.9 Pressure Streamlines along the flow domain

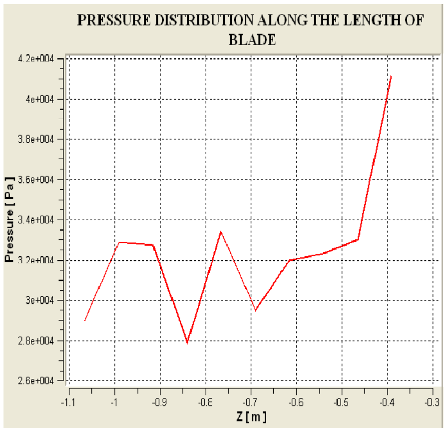


Fig . 7.10 Graph showing the pressure and velocity distribution

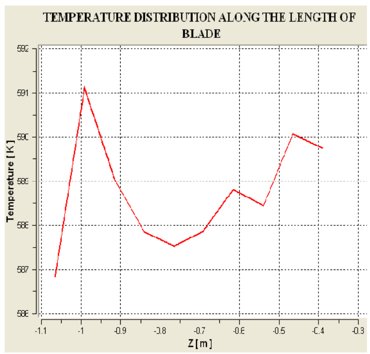


Fig . 7.11Graph showing the temperature distribution

CASE 2 Untwisted Blades

By considering the same boundary conditions and having the same mesh geometry the simulation is solved for untwisted blade and thus we get the following results

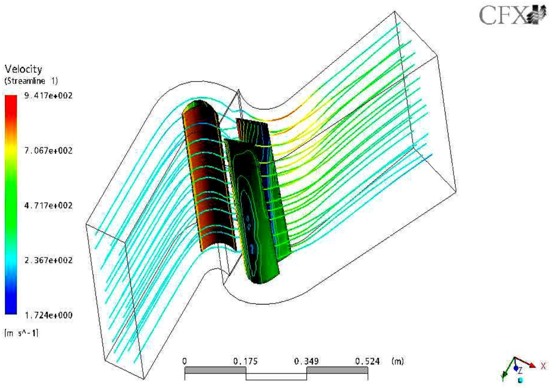


Fig 7.12 Velocity distribution for Untwisted Blade

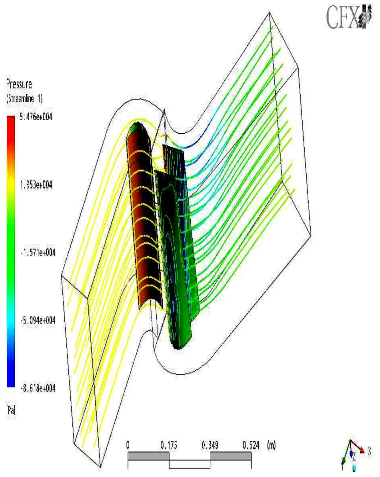
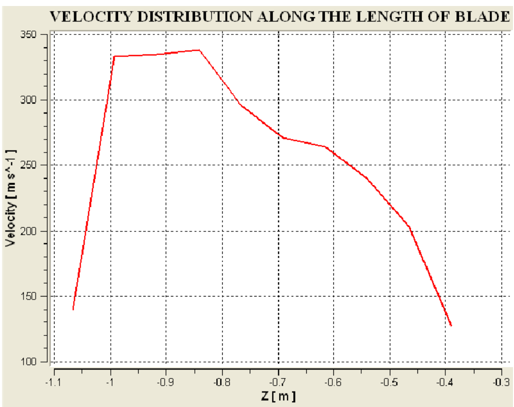
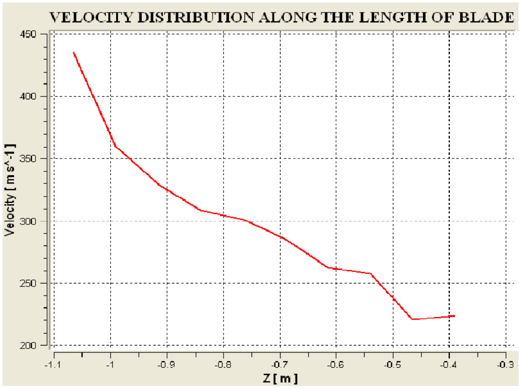


Fig 7.13 Pressure distribution for

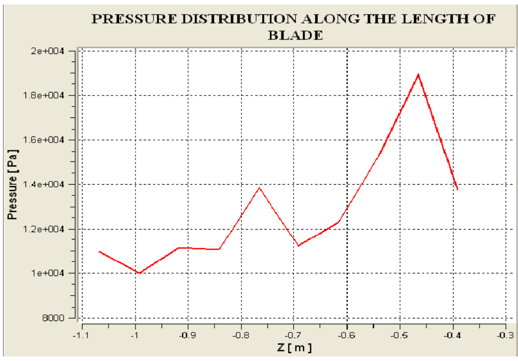


Untwisted Blade

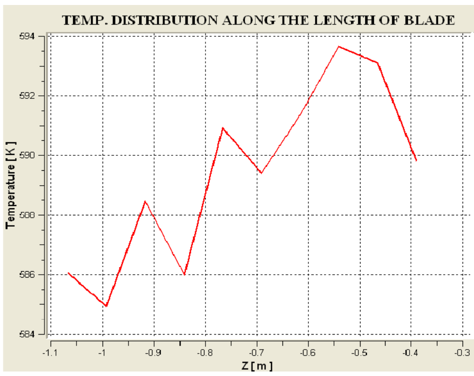
Graph showing the velocity and pressure distribution.



Graph showing the pressure distribution.



Graph showing the temperature distribution.



It shows that due to aerodynamic shape of twisted blade sudden pressure drop occur and getting more velocity as compare to untwisted blade geometry hence for getting better result or more output now a day's twisted blade geometry is recommended

CONCLUSION

Numerical simulation of the steam turbine last stage flow is considerably simplified compared to reality. The calculation is performed for water ideal gas, whereas real water steam with content of liquid phase and complex droplet structure flows in the last stage. At the inlet of the calculation domain mass flow rate is assumed without taking into account the differences in steam wetness and the three-dimensional structure of non-stationary flow downstream of the last but one stage. The geometry of the stage is modified by a changing the twisting of the rotor blade. Imperfection of mathematical model and numerical method affects also the results. Under the given situation we can consider the main characteristics of the flow as substantial results.

It describes properties of the flow in the last stage of the condensing turbine. The numerical simulation denotes sinking of stage thermodynamic efficiency in connection with the steam flow through the tip gap above the rotor blades that deteriorates the flow in the region near the outer wall of the flow channel.

The distribution of the stator blade loss coefficient shows on enhancement of losses in the outer part of stator blade cascade. From the distribution of rotor blade loss coefficient follows enhancement losses in the inner part of rotor blade cascade.

The distribution of axial velocity component downstream of the last stage based on calculation and measurement shows that it would be possible to improve the stage efficiency by reducing the steam flow near the outer and also inner walls of the flow channel.

Variation of the velocity into the rotor blade cascade during its rotation is can be Approximately presented by the graph generated along the length of the blade.

The flow structure downstream of the stage depends on the effects of rotor and also stator blades. At the exit of the stage originates a complex flow field with vortex structures in outer and also inner parts of the flow channel. The number of vortices in the exit annular cross section does not need to correspond to the number of rotor blades, but it can be even lower and can be connected with the number of stator blades.

This project has attempted to study a simple but systematic approach towards mesh generation with the applicability of 3D CFD (Navier-Stokes simulations) can be extended towards turbine blade geometries.

In this project work computational fluid dynamics (CFD) analysis intends to utilize the study of steam flow path in the last stage(LP Turbine) of rotor-stator disc cavities of steam turbine.

With the aid of CFD analysis the effect of twisting of blade on the steam flow parameters for LP- turbine can be analyzed.

Computational Fluid Dynamics (CFD) is a computer-based tool for simulating the behavior of systems involving fluid flow, heat transfer, and other related physical processes. It works by solving the equations of fluid flow over a specified region, with specified (known) conditions on the boundary of that region. This field is known as Computational Fluid Dynamics (CFD), which principally focuses on solving the continuous Navier-Stokes equations in a discrete form numerically using a grid of discrete points distributed throughout a computational domain.

The method is applied to typical steam turbine design cases, namely the calculation of the fully three-dimensional flow through steam turbine stages. It is shown, that CFD supports the design process by providing an in-depth physical insight into the complex flow features and by revealing objectives for further research and optimization.

Concluding features….

- To simulate Steam Turbine designs which are very expensive for prototype testing, CAD/CFD Technology is comparatively less expensive.
- Create more reliable, better-quality designs.
- CAD-CFD Technology to reduce the amount of prototype testing 98
- Computer simulation allows multiple “what-if” scenarios to be tested quickly and effectively.
- It provides adequate insight for better designs.

Following software's are used in this project work:

- For CAD – Pro/Engineer wildfire-04.
- For Finite Volume Meshing (Discretization) – ANSYS WORKBENCH.
- For CFD (computational fluid dynamics) – ANSYS CFX.
- For Structural Analysis of Blade – ANSYS WORKBENCH.

FUTURE SCOPE

The future of steam turbine-generator technology lies in an increase in capacity, which will stem from larger and more efficient turbines. Nuclear fission will continue to compete with conventional fuel combustion in the production of steam for the turbine, but the heart of the technology will still lie in the technology of the steam turbine itself.

REFERENCES

1. M. Jansen, W. Ulm, "Modern Blade Design for Improving Steam Turbine Efficiency", VDI-Berichte Nr. 1185, 1995.

2.H. Oeynhausen, A. Drosdziok, M. Deckers, "Advanced Steam Turbines for Modern Power Plants", IMechE Paper No. C522/032, 1997.

3. J.I. Cofer, "Advances in Steam Path Technology", Power-Gen Europe, Amsterdam RAI, NL, 1995.

4. R.B. Scarlin, "Advanced Steam Turbine Technology for Improved Operating Efficiency", Power-Gen Europe, Amsterdam RAI, NL, 1995.

5. V.C. Patel, W. Rodi, G. Scheuerer, "Turbulence Models for Near-Wall and Low Reynolds Number Flows: A Review", AIAA Journal, 23, 1308-1319,1985.

6. F.M. White, "Viscous fluid flow", McGraw Hill, 1991.

7. Voelker L., Casey M., Neef M., Stueer H. (2005), The Flow Field and Performance of a ModelLow Pressure Steam Turbine. Proceedings of 6th European Conference on Turbomachinery -Fluid Dynamics and Thermodynamics, pp. 232-235, Lille, France.

8. J. M. Owen and R. H. Rogers, Flow and Heat Transfer in Rotating-Disc Systems, Volume 1 – Rotor-Stator Systems, John Wiley & Sons Inc., New York, 1989.

9. CFX Ltd., CFX-5 Solver Theory, Version 5.7, ANSYS Co. / CFX Ltd., UK, 2003

10. N. J. Hills, J. W. Chew and A. B. Turner, Computational and Mathematical Modeling of Turbine Rim Seal Ingestion, Vol. 124, American Society of Mechanical Engineers, 2002.

BOOKS:

1. Computational Methods for Fluid Dynamics by Joel H. Ferziger

2. Computational Fluid Dynamics: The Basics with Applications by John David Anderson.

3. An Introduction to Multi-grid Methods by Pieter Wesseling.

4. Steam turbine for power generation by M.V.Pande – NPTI Nagpur.

5. Mr. Sivasubramaniyan (www.ansys.com)