2nd National Conference on CFD APPLICATIONS IN POWER AND INDUSTRY SECTORS
Centre of Excellence for CFD, BHEL
28th – 29th January 2009, Hyderabad, INDIA

ARANIRD001

ESTIMATION OF EFFICIENCY OF LOW PRESSURE STEAM TURBINE BLADING USIND CFD TECHNIQUE

KCH Peraiah
Research & Development, ARANI POWER SYSTEMS LTD
Hyderabad, A.P., India

Dr. Udai Singh
UKS Consultants
Trendles, Ryme Intrinseca, Sherborne DT9 6JX
England

Ramu Maddi
Research & Development, ARANI POWER SYSTEMS LTD
Hyderabad, A.P., India

Sumalatha H
Research & Development, ARANI POWER SYSTEMS LTD
Hyderabad, A.P., India

ABSTRACT

The flow in a Low Pressure Steam Turbine, carrying longer blade passages, is complex and involves understanding of energy conversion in three dimensional geometries. To develop better performing blades, it is essential to identify the losses generating mechanism and study their influence and effects on performance. This paper outlines design considerations and the estimation of efficiency of Low Pressure Steam Turbine using CFD, thus aiding in optimizing the design and helps in integrating CFD into the design process itself. The CFD results are in concurrence with the two dimensional mean line code & stream line curvature method.

INTRODUCTION

During the course of expansion of steam in turbines, the state path crosses the saturation line and hence subsequent turbine stages (mostly last stages i.e. Low Pressure stages) operate with wet steam. Since the pressure is low, the volumetric flow is more, to accommndate increased volumetric flow[1], it is necessary to have larger area leading to long blades and these stages have lower thermodynamic efficiencies than those operating in the superheated region. Modern turbomachinery designs aim to increase blade loading and pressure ratio while maintaining the same high efficiency level. This results in a higher power density and lower part count and therefore lower cost. In this perspective, secondary flows and the interactions between leakage, radial flows,mainstream flow seperation and flow reversals[2] contribute considerably to the overall turbine losses. The recent developments of the CFD code performance in terms of accuracy, sensitivity and efficiency, enables to reduce the design cycle by coupling CFD codes with optimization tools.

Need for 3D design

Most of the phenomena involoved in turbo machinery flow can be understood and predicted on a two-dimensional or quasi three-dimensional basis, but some aspects of the flow(refer Need for CFD indesign) must be considered as fully three-dimensional (3D). Mostly 2D and quasi 3D methods are very useful in the preliminary design process. When it comes to the Low Pressure blading, the blade heights are high the flow is complex and 3D in nature. This necessiates 3D design for the Low pressure steam turbine.

Need for Twist

In general, if the Blade Height to Hub diameter is greater than 0.14, we may have to go for the twist of the Blade. Across radial height of the blade the change in peripheral velocity changes the velocity triangle along the span of the blade. Thus to compensate this and to maintain optimum incidence and deviation angles at all radii, it may be necessary to go for Twist of the blade.

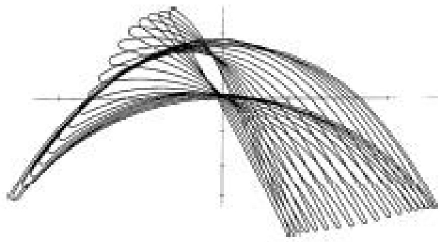

Fig. 01 Schematic diagram of twisted profile

In deciding the twist additionally both the Radial Equilibrium and Free-Vortex theories are considered. The Figure 01 shows the schematic diagram of twisted profile from top view.

Basic Losses in Turbomachinery

It is assumed that the overall ioss in a machine blade row is the sum of a number of basic loss components. It is usually assumed that these can be evaluated independently and then combined linearly to obtain the total loss. The basic loss components are usually listed as follows:

Profile loss: loss due to the boundary layer growth on the blade pressure and suction surface in a uniform two-dimensional flow plus the trailing-edge loss.

Secondary loss: The secondary losses arise from complex three-dimensional flows set up as a result of the rolling up of the endwall boundary layer, and associated streamwise vorticity and separation of the flow. This formation of losses is shown in Fig 02. This interaction between the flow through the blade passage and the end wall boundary layer results in a total pressure loss which is often a significant fraction of the total loss for low aspect ratio turbines.

Tip clearance loss: The tip clearance loss arises due to the required gap between the end of a rotor blade and the adjacent wall. For the two types of blades generally encountered, i.e. unshrouded and shrouded blades, the loss generating mechanisms are quite different. In the case of shrouded rotor blades, the clearance flow is separated from the flow through the blading. Losses arise then from energy dissipation across the shroud and from mixing of the clearance flow with the main flow downstream of the blade row. For unshrouded rotor blades, a clear separation between secondary and clearance losses is not possible. In this case the secondary loss obtained at zero clearance is generally assumed to remain unaltered as the clearance gap is increased. The tip clearance loss is then taken as the difference between total losses with and without clearance. However, it is obvious that secondary loss will be affected somewhat by the clearance. Other losses include disc friction losses, windage losses and wetness losses etc.

All the above correlations express losses as a function of key geometric and flow parameters.

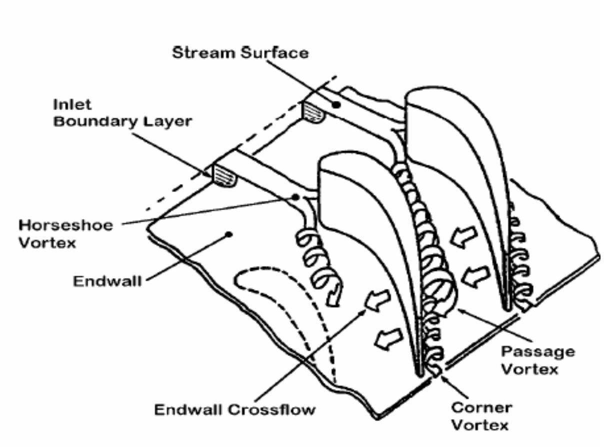

Fig. 02 Formation of Secondary losses

NEED FOR CFD IN DESIGN

The goal of the aerodynamic design process, for turbomachinery components, is to minimize aerodynamic losses and maximize performance, within the geometric, physical, and economic constraints placed on the component. This goal is accomplished through a process that consists of two primary phases: preliminary design and detailed design. The preliminary design phase establishes the overall characteristics of the component, such that it will satisfy the requirements and constraints of the overall turbine design. Basic flow-path configurations, blade counts, blade-row spacings, and initial blade shapes are among the characteristics determined during preliminary design wherein structural requirement are also considered. The process is highly iterative, due to the large number of component and flow-path features that must be optimized, through the analysis of many configurations.

In contrast, the detailed design process focuses on one or a small number of design configurations that offer the optimum combination of features, and the best match with aerodynamic performance objectives, based on the analyses of the preliminary design. The objective of the detailed design process is to predict, as realistically as possible, those characteristics of the flow that are critical to the aerodynamic performance of the turbomachinery component being analyzed. Such characteristics would include **tip clearance flows, shock–boundary-layer interactions, blade–end-wall interactions, flow separations, wakes**, and any other regions of high loss. The level of detail and capability required of a particular flow model will be determined by the phase of the design process to which it is applied. During the preliminary design process, simplifying assumptions are typically made that allow the flow to be modeled in less detail. Once the overall characteristics of a particular design have been established using these 2D and quasi 3D tools, then the detailed behavior of the flow for that configuration must be determined, using the full capabilities of the available CFD analysis tools.

During the detailed design process, design features that are specifically three-dimensional in nature are being evaluated. Characteristics that can significantly impact component performance, such as **blade twist, blade metal angles, blade throat, Leading and trailing edge radius, blade position, and tip clearance height,** must be optimized, based on their influence on a viscous flow field. To evaluate such issues in a test rig would be time-consuming. Therefore, the analytical tools of the detailed design process must model the flow physics with sufficient accuracy to assist in the evaluation process. To achieve the necessary level of accuracy, a **3D viscous transonic flow model** must be employed in the analysis tools. The application of such a model will result in solution times several orders of magnitude longer than those for preliminary design tools. However, the expense in terms of computation time can be offset by reductions in both design/development cycle time and design risk. By employing 3D viscous flow analysis tools during the detailed design, the necessity of performing one or more redesign/ retest iterations can be greatly reduced. In addition, for those components that cannot typically be redesigned, due to schedule constraints caused by long lead times for hardware acquisition, the likelihood of a successful design can be greatly enhanced by using accurate flow modeling tools.

CRITICAL CAPABILITIES FOR TURBOMACHINERY FLOW ANALYSIS TOOLS

Governing equations that model 3D viscous transonic flow are essential to satisfy this requirement. The specification of boundary Conditions [3] for the governing equations should be general enough to accommodate all types of boundaries encountered in turbomachinery flow paths. In addition, the turbulence model should adequately account for the characteristics of typical turbomachinery flow fields, such as flow-path curvature, rotating flow, high-pressure gradients, separated, and recirculating flows. To analyse bladerow interations unsteadyflow analysis are to be considered..The selection of numerical solution techniques[4] is also a critical issue. Discretization of the governing equations and choice of solution techniques for the resulting finite-difference or finite-volume approximations to those equations are at least partly dictated by simplifying assumptions made regarding the character of the flow. Thus, these selections must be compatible with the flow behavior anticipated in the components being analyzed. Representation of the geometric configuration of the component, via the solution grid, must also reflect physical reality. Solution grids must be able to accurately depict blade and flow-path shapes and model complex details such as tip clearance regions. In addition, these grids must be able to resolve the details of the flow structure. Various grid types exist that will satisfy these requirements with varying degrees of success, depending on the application. The proper selection of a solution grid is essential, to ensure the accuracy of both the modeled component geometry and the flow analysis. Program performance is also of considerable importance for detailed design flow analysis software. Because of the complex nature of the equations being solved, such software typically can require several days of computation time to obtain a solution. Therefore, optimization of the software is essential. Use of **parallelization**, where possible, are also valuable techniques for improving performance of the analysis tools. Pre- and postprocessing also warrant consideration, due to the significant amount of time that must be invested in both preparing the input for an analysis and interpreting the results of a completed analysis. “User-friendly” tools are essential to assist in these processes. Most importantly, the use of an interactive graphical user interface can dramatically improve the efficiency of both of these stages of the analysis process.

GEOMETRY DESCRIPTION

Before a computational grid can be imposed on a particular flow-path solution domain, the geometry of the flow-path boundaries must be modeled. This numerical definition of the flow-path shape provides the framework upon which to build the computational grid. In our present work modeling was done as per the layout of LP blading considering the tip clearances. The degree to which a CFD analysis program functions successfully as a detailed design tool depends on how closely it meets certain requirements. Obviously, the physical model must be sufficiently accurate to represent those aspects of the flow that will impact aerodynamic performance. The geometric modle is shown in Fig 03.

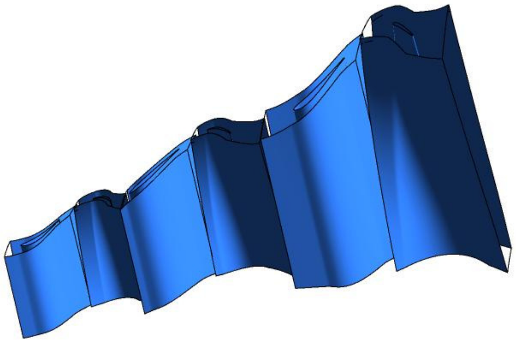

Fig 03. Geometric model of LP Blading

The approach taken to describe the flow-path geometry will be determined, in part, by the complexity of the flow-path boundary shapes. Since the bladed disks are axi-symmetric in nature, modeling of single blade row with necessary boundary conditions will be sufficient. The technique selected for the geometric description and modeling of components should offer sufficient flexibility to accommodate design changes. Because component and flow-path shapes constantly evolve during the design process, modification of the geometric model of the flow-path boundaries must also be easily accomplished, and the updated model must be made rapidly available. These requirements necessiates to utilize the geometric modeling capabilities of a computer-aided design (CAD) system. We used relatively automated scheme for the description and modification of component geometry. Parametric representations of the model and tabular representation of the flow path coordinates at a selected streamwise calculations are used. This faciliates the flexibility for any flow path changes in future. Another issue that must be considered in the description of flow-path geometry is that of the transfer of consistent information to other functional groups during the design process. Particularly in a **concurrent engineering** [5] environment, it is essential that all functional organizations have access to and work with the same "master" geometric model of an object or flow path.

GRID GENERATION

After the geometric model of the turbomachinery component has been established, the next step in the process of communicating this configuration to the CFD analysis program is to define the computational grid within the physical domain. The boundaries of this region are typically defined by the flow-path surfaces (end walls, blades, etc.) and by the periodic boundaries between blade passages, where appropriate.

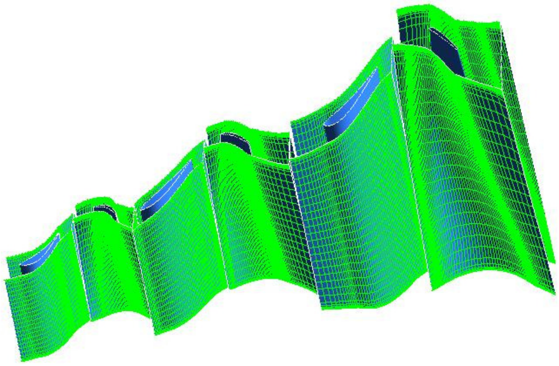

Fig. 04 Grid Generation of model

Inlet and exit boundaries are established at points upstream and downstream, where the necessary flow conditions are assumed to be known.

Within this region, a three-dimensional computational grid is applied, such that the governing equations will be solved at every point on the grid, or within every cell formed by the grid. The grid imposed on the physical domain must conform to the boundaries of that domain and must provide adequate resolution in all areas of the flow field to permit accurate prediction of the flow behavior. The mesh was generated as shown in Fig. 04. And the grid independnce check was done and the optimum number of elements found for stator or rotor are aound 2,40,000, and the y+ value [6] also considred. In our model we generated structured mesh. The typical mesh pattern aound blade profile is shown in Fig.05.

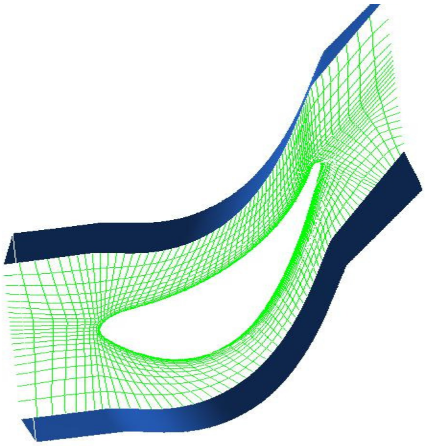

Fig. 05 Mesh pattern aound the profile

Sufficient number of nodes are taken around the blade profile and wall boundaries to caputre wall boundary layer effect.

PRE and POSTPROCESSING

A significant percentage of the total time spent on a CFD analysis is involved in pre- and postprocessing activities. Both the setup for an analysis and the evaluation of results require considerable effort on the part of the component designer. Therefore, the use of software tools to automate or facilitate these activities has the potential to substantially reduce the time required for the analysis and improve the overall efficiency of the process. Preprocessing involves the definition of the boundaries of mesh elements and interfaces between rotor and stator etc. The basic boundary conditions used for an element was shown in Fig 06. And for inflow boundary total temperature, total pressure and fiow angle are given.

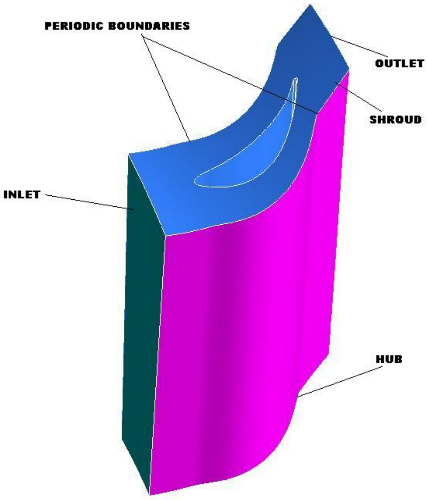

Fig. 06 Boundaries on mesh volume

For outlet flow boundary Static pressure outlet model was slected. The basic boundaries and inputs given in CFD model are given below.

SIMULATION TYPE : STEADY

HEAT TRANSFER MODEL: TOTAL ENERGY

TURBULENCE MODEL: k epsilon

INLET: TOTAL PRESSURE
TOTAL TEMPERATURE
TURBULENCE LEVEL
WETNESS FRACTION

OUTLET: STATIC PRESSURE

MATERIAL: STEAM

SPEED in rpm

And the above problem was run for the convergence level of 10^{-4} in 450 iterations.

Post processing function for a CFD analysis provides the necessary information of variables such as pressure, temperature, velocity and enthalpy etc. in the flow regime. And the important parameters are calculated and tabulated in the Table 1. From postprocesing we caluculated efficiences. The postprocessing function for a CFD analysis provides a numerical flow visualization capability. This facility is essential for understanding and interpreting analysis results that consist of number of dependent variables at thousands of discrete locations within the flow field. For these CFDvisualization tools to be useful to the component designer, they must be highly interactive, and also user-friendly, requiring only minimal training. To obtain useful information in a 3D environment, the visualization tool must have the capability to display color contours of scalar properties on flow-path and component surfaces, and also on user- specified, arbitrarily oriented slices through the flow path. Vector properties at discrete grid nodes or cells are best represented by arrows, oriented in the direction of the vector, and scaled to its magnitude.

Additional scalar property information may be communicated in a vector display, by superimposing a color scale on the vectors themselves. Particle traces may be generated to visualize flow behavior, by positioning ‘‘rakes’’ at user- selected locations in the flow path. Particles are then released from these rakes, and their paths are determined by integrating through time, given the local velocity vector distribution. The Figures 07 and 08 shown the flow properties in the domain.

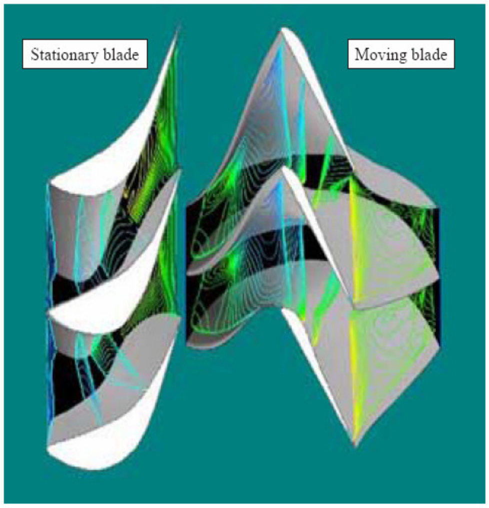

Fig.07 Mach number distribution in stage

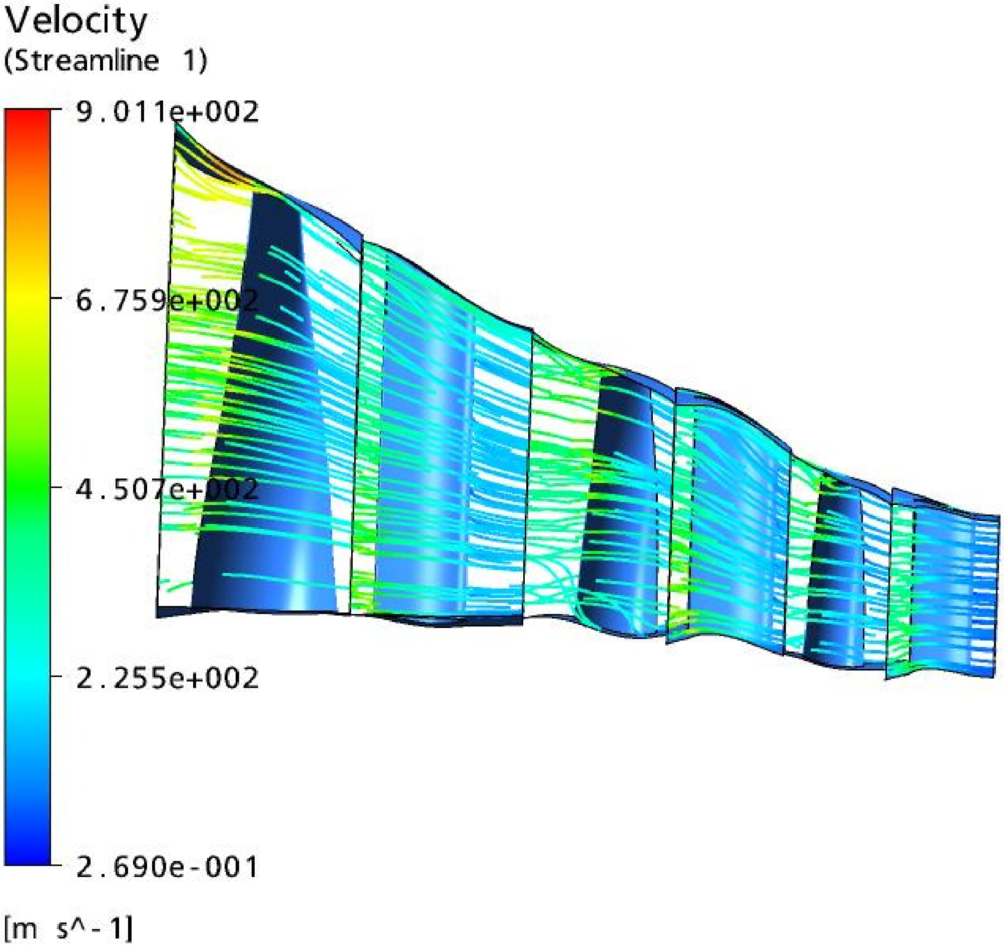

Fig.08 Streamlines shown in flow domain

Quantity	LP BLADING		Diff %
	2-D CODE	CFD	
Inelt Temperature, (deg C)	108.75931	109.294	0.4916
Inlet Pressure, bar abs	1.3739412	1.404	2.1878
MassFlow Rate, kg/s	24.938417	24.85	-0.3545
Rel. Exit Mach Number	1.2000378	1.167	-2.7531
Absolute Exit Pressure (bar)	0.098132595	0.0974593	-0.6861
Exit Temperature (deg C)	45.46455	44.752	-1.5673
Dryness Fraction	86.74959	88.2	1.6720
Stage Efficiency, %	84.5	83.68	-0.9836
Actual Total Enthalpy Drop, kJ/kg	218.87	218.46	-0.1873
Blade Row Work, KW	5456.6	5428.9	-0.5076
Exit Relative Velocity m/sec	453.23184	464.2	2.4200
Exit Axial Velocity, m/sec	232.75327	229.65	-1.3333
Exit Absolute tangential Velocity, m/sec	-44.22278	-43.9	-0.7299
Exit Sonic Velocity, m/sec	377.6813	378.23	0.1453
Exit Density, kg/m^3	0.077171996	0.07694	-0.3006

Table 01 : Comparison of CFD results with 2D code

RESULTS AND CONCLUSION

Subsequently the aerodynamic losses for CFD are corelated with 2D code and quasi 3D streamline curvature method. It gives the validation of CFD model. The proceedure, boundary conditions and the CFD model can be utilized for the Design Optimization process.

REFERENCES

[1] Earl Logan, Jr.Ramendra P. Roy, Handbook of Turbomachinery, Second Edition.

[2] J.S.Rao, K. Ch Periah et. al., Estimation of Dynamic Stresses in Last Stage Steam Turbine Blades under Reverse Flow Conditions, International Conference on Vibration Engineering & Technology of Machinery, VETOMAC-IV, December 17-19, 2007.

[3] F. J. Malzacher et.al., Aerodesign and Testing of an Aeromechanically Highly Loaded LP Turbine, Journal of Turbomachinery, OCTOBER 2006 Vol 128 / 643, Transcations of ASME.

[4] Paul Traub et. al., NUMERICAL INVESTIGATIONS FOR OPTIMIZING THE AERO-ACOUSTICAL DESIGN OF MODERN LP-TURBINES, ICSV13, July 2-6, 2006, Vienna, Austria.

[5] Richard A. Layton et. al., Conceptual Basis for a New Approach to Bladed-Disk Design, Journal of Engineering for Gas Turbines and Power, 324/ Vol.122, APRIL 2000, Transcations of ASME.

[6] Ste´phane Burguburu et. al., Numerical Optimization of Turbomachinery Bladings Journal of Turbomachinery JANUARY 2004, Vol. 126 Õ 91, Transcations of ASME.