KME

CONSORTIUM MATERIALS TECHNOLOGY for demonstration and development of thermal energy processes

Improved steam turbine design for optimum efficiency and reduced cost of ownership

Magnus Genrup and Srikanth Deshpande

KME-607

Improved steam turbine design for optimum efficiency and reduced cost of ownership

Magnus Genrup and Srikanth Deshpande

KME-607

Preface

The project has been performed within the framework the fifth stage of the material technology research programme KME.

KME, Consortium Materials technology for demonstration and development of thermal Energy processes, was established 1997 on the initiative of the Swedish Energy Agency. In the consortium, the Swedish Energy Agency, seven industrial companies and 18 energy companies participate. The programme stage has been financed with 60.2 % by participating industrial companies and with 39.8 % by Swedish Energy Agency. The consortium is managed by Elforsk.

The programme shall contribute to increasing knowledge to forward the development of thermal energy processes for various energy applications through improved expertise, refined methods and new tools. The programme shall through material technology and process technology developments contribute to making electricity production using thermal processes with renewable fuel more effective. This is achieved by

- Forward the industrial development of thermal processes through strengthen collaboration between industry, academy and institutes.
- Build new knowledge and strengthen existing knowledge base at academy and institutes
- Coordinate ongoing activities within academy, institutes and industry

KME's activities are characterised by long term industry relevant research and constitutes an important part of the effort to promote the development of new energy technology with the aim to create an economic, environmentally friendly and sustainable energy system.

Abstract
The cost of ownership of a power plant is partly governed by the efficiency of the turbine island. The efficiency of an industrial size steam turbine can be increased by an improved steam-path design. Most industrial size units have fairly simple prismatic blades in a significant part of the turbine. Development in manufacturing technology has levelled the cost for more advanced geometries and offers higher performing turbines. The work in KME-607 aims at exploring the potential by introduction of the advanced blading.

Sammanfattning

Ångturbinens prestanda påverkar produktionsekonomin under anläggningens hela livslängd. Ångcykelns prestanda bestäms både av processparametrarna och ångturbinen. För att få hög verkningsgrad väljer man i allmänhet avancerade ångdata, dvs. höga tryck och temperaturer. Primärt påverkar dessa cykeln men också ångturbinen i sig själv. Cykelparametrarna påverkar volymflödet för ett visst massflöde och detta har en stor inverkan på ångturbinen. En turbin som konstrueras för stora volymflöden har långa skovlar som utlovar hög verkningsgradspotential. Anledningen till den högre verkningsgraden är att förluster som finns t.ex. lokalt vid ändväggarna och pga. flöden genom tätningar, får relativt betraktat mindre inflytande. Turbinens verkningsgrad påverkas också av dissipation i t.ex. gränsskikt på skovelprofilerna.

Det här projektet har utförts som ett samarbete mellan Institutionen för Energivetenskaper vid Lunds universitet och Siemens Industrial Turbomachinery i Finspång.

Tidigare utveckling inom det industriella ångturbinsegmentet har varit inriktat på att få ner kostnaderna snarare än att öka verkningsgraden. De flesta industriella ångturbinerna använder prismatiska eller raka blad där skovlarna är tillräckligt korta. Innebörden i ”kort” är att stegets nav/topp-förhållande understiger en viss nivå. Vanligtvis innebär det alla steg utom de tre sista i en industriturbin. De sista stegen måste konstrueras för större variationer över höjden och bladen blir därför tre-dimensionella. Ett två-dimensionellt blad fräses i en två-axlig maskin, medan mer avancerade bladformer måste tillverkas i en fem-axlig maskin. Kostnaden för fräsning med en fem-axlig maskin har minskat avsevärt och skillnaden mot enklare två-dimensionell är idag inte för stor. Denna produktionsteknik har införts på större kraftverksturbiner med följd att verkningsgraden har ökat högst avsevärt. Man kan tyvärr inte uppnå samma verkningsgradsnivåer in en industriturbin som i en kraftverksturbin – trots att samma teknologi är tillgänglig. Skillnaden i storlek och tryckförhållande i husen gör att den mindre turbinen alltid får lägre verkningsgrad.

Målet med projektet är att hitta en framkomlig väg för att introducera mer avancerad beskovling tidigare i industriturbiner och på så sätt höja verkningsgraden.

Arbetet utförs med typiska moderna en-, två- och tredimensionella beräkningsverktyg. Det sistnämnda var inte med i den ursprungliga planen men var nödvändigt för att etablera ett referenssteg och båda närliggande. Referenssteget är ett förhållandevis långt steg med låg reaktionsgrad från en Siemens SST-900 turbin.

Arbetet visar att det finns potential för att höja verkningsgraden och på så sätt reducera livscykelkostnaden. Arbetet pågår fortfarande och de absoluta nivåerna är under fortsatt utredning.

Nyckelord: Ångturbin, Aerodynamik, Kanalutlägg, Verkningsgrad

Summary

The cost of ownership of a power plant is partly governed by the efficiency of the turbine island. The turbine stands for the production revenues when transferring the fuel into electric power and district heating. One gauge of the quality of the individual processes is the component efficiency – the current project addresses the turbine part of the power plant. The turbine efficiency is dependent on both process parameters and the blading aerodynamics. The former is typically the steam data (e.g. temperature, pressure and mass flow) that influences the volumetric flow through the turbine. A larger turbine performs better than a smaller ditto and this can be explained by the relative importance of clearance- and secondary flows. The blading aerodynamics is the local flow processes in each stage in the turbine. The efficiency of a stage (i.e. a stator and a rotor) is limited by losses due to e.g. dissipation in boundary layers, dissipating of secondary flows, leakage mixing and lost work, etc. The project has been conducted in collaboration between the department of Energy Sciences at Lund University and Siemens Industrial Turbomachinery in Finspong.

Most previous development efforts in the industrial steam turbine segment has been towards reduced first cost and not necessarily the efficiency. Most industrial steam turbines utilizes prismatic (or un-twisted) blades for shorter stages. The word “shorter” is used in a sense that all stages except for the last three are constant section blades. The last stages have to be designed for large span-wise flow variations by three-dimensional blade shapes. A constant section rotor blade typically is milled in a two-axis machine whilst more advanced shapes requires five-axis flank milling. The costs associated with the latter has today levelled with the more simple manufacturing methods. This technology step has been introduced in larger size utility type of turbines with very high attainable efficiency levels. An industrial size steam turbine cannot reach the same level of efficiency and lags several points behind as for a utility type unit, because of the lower volumetric flow and the cylinder pressure ratio.

The goal of the project can be summarized as “exploring the future flow path for higher-performing steam turbine plants” by evaluating the maximum cost effective efficiency potential of industrial steam turbines. The latter is by utilizing improved blade profiles and advance stacking technologies.

The work has been carried out in state-of-the-art turbine design tools and comprises: one-dimensional tool, two-dimensional throughflow analysis and full three-dimensional high-fidelity CFD. The latter was not part of the initial plan but was introduced because it was necessary to establish a baseline for back-to-back comparisons. The datum stage is a tall stage in an assumed Siemens SST-900 turbine. The work, however, is generic for low-reaction steam turbines.

The work shows that the stage performance can be increased. The full potential, however, is still to be evaluated as the work continuous in the project. The project work at Lund University effectively started in late 2012 and is not yet completed.

Keywords: Steam Turbine, Aerodynamics, Flow Path Design, Efficiency

Table of contents

1 Introduction

1.1 Background

The cost of ownership of a steam plant is, indeed, complicated to discuss in simple terms – or even to assess properly. The life-cycle costs are governed by both the initial cost and the costs associated with the running of the turbine. The latter can, with respect to the KME-project, be summarized as fuel costs, operation- and maintenance spending. The revenues, on the other hand, are directly related to the previous reasoning, but in addition, the produced power. Both are related to the plant efficiency and one can show that the fuel input for a certain power, or heat duty for a district heating plant, is proportional to the efficiency. The efficiency for a steam plant can be shown to be related to:

- Steam admission temperature
- Steam admission pressure(s)
- Final feed water temperature
- Number of feed heaters
- Exhaust pressure
- Turbine expansion efficiency

The first five are thermodynamic properties and all have a strong influence on the cycle efficiency. It is, at this point, convenient to introduce a Carnot-type efficiency for the steam (or Rankine) cycle in concert with the concept of average temperature of heat addition ($T_m=(h_{adm}-h_{FFW})/(s_{adm}-s_{FFW})$). The average temperature of heat addition is increased by increasing the first three (admission temperature, admission pressure and final feedwater temperature), whilst the exit temperature is set by the turbine back pressure. Most industrial type of plants have fairly advanced steam data (e.g. 140 bar and 540…565°C), but lags significantly behind utility type turbines (300 bar and 600°C). The reason for the lower steam data is partly driven by cost for the boiler and piping, rather than necessarily the direct steam turbine cost. On top of direct costs for materials, the fuel may pose limitations with respect to corrosion (typically from chlorides in the final superheater). The other part is the size of the turbine, since too low volumetric flows will lower the available turbine efficiency. On top of the volumetric flow, the amount of moisture (condensed water) inside the turbine has a detrimental effect on the efficiency.

An increased efficiency means that the produced power for a certain amount of fuel increases. Hence, regardless of the fuel, an increased production could be considered as an effective means to reduce the carbon footprint.

The last part of the previous list is the turbine efficiency per se. Since on top of the cycle driven issues, the blading technology sets a maximum achievable level. The efficiency of a turbine cylinder is a function of the volumetric flow, pressure ratio and the blading aerodynamics. The volumetric flow is primarily a gauge of size and one can argue that a large turbine performs better than a smaller. The rationale behind the reasoning is based on the relative influence from boundary layer related end-wall and tip-clearance losses. One can to a first order of approximation assume that the absolute thickness of the end-wall boundary layers scales with a length-based Reynolds number and the transverse force is the blade loading – hence a taller blade should have relative less influence from end-wall flows. The same reasoning yields for clearance flows since one can assume a certain physical clearance and sealing technology. This is the main driver behind the higher efficiency for geared industrial turbines, where the volumetric flows are small. The turbine stage count is also greatly reduced for typical geared designs. A lower stage count implies a shorter rotor with potentially lower requirements for certain clearances in order to prevent from rubbing. A typical state-of-the-art gear typically has an efficiency of 0.985 to 0.993 for normal and vacuum types, respectively. The cylinder pressure ratio sets the amount of "reheat" where the loss in stage turns into work in the next. This means that losses earlier in the expansion have a less severe impact on the total cylinder efficiency.

Most of the industrial size turbines have prismatic turbine blades for cost reasons, except for the stages N-2, N-1 and N (the last). The word "prismatic" is used in a sense that the blades have un-twisted-, un-leaned[1]- and constant section blades. All blades, except for typically the last three, in a steam turbine are made out of forgings. This is either by grinding or two-axis milling for guide vanes and rotors, respectively. This is a result of the earlier prohibitively higher costs associated with five-axis flank milling. Today, however, the costs have levelled and one can therefore manufacture more advanced vanes and blades earlier in the turbine – for higher efficiency at the same cost. The value of efficiency has changed over the years and it was quite common in the nineties to utilize technology improvements to reduce costs (by e.g. lower part counts) rather than necessarily increase the performance. This situation has changed today and performance is indeed important for competitiveness. The larger utility type of turbines paved the way in the nineties and one can today see, indeed, very high efficiency levels. Larger turbines experiences all of the previous reasoning related to volumetric flows and cylinder pressure ratios (e.g. 300/0.02 = 15000). Hence, by virtue of the power class, the efficiency potential is higher. One cannot therefore expect that the industrial size ever will approach the level of the utility turbines. Siemens [1] has retrofitted one of the turbines at the Boxberg plant in Germany where the performance test in-situ showed 94.2 and 96.1 percent for the HP- and IP-cylinder, respectively. These amazingly high levels are due to the advanced three-dimensional blading throughout the turbines. It should be mentioned, that the low-pressure (LP) turbines still have efficiency issues related to moisture. It should be noted that the efficiency can be measured and assessed with high precision for HP- and IP-turbines, whilst LP-turbines are more intricate and one has to revert to indirect methods.

The work in the present project aims to introduce certain features from the larger-size technology into the industrial range of turbines.

[1] Not necessarily the case for all turbines

1.2 Description of the research field

The research field is related to turbine aerodynamics where the aim is to minimize the losses (i.e. entropy gain) in the turbine flow path. There are several ways to approach the research question, namely; numerically or experimentally. In this project, the former approach is adopted. The reason for not running experimental research is the prohibitive costs of a test turbine campaign and no availability of such turbine at Lund University.

1.2.1 Turbine stage efficiency

The efficiency of a turbine stage a function of how one can minimize the entropy gain due to [2], [3], [4]:

- Viscous dissipation in either boundary layers or free shear layer (e.g. mixing out secondary flow vortices)
- Mixing of flows with different mass, momentum and energy
- Non-equilibrium processes (e.g. droplet condensation shocks and "normal" shocks)

One can show that the entropy production in a boundary layer scales with free-stream velocity cubed [2], [3], [4]. This is true for turbulent boundary layers, whilst laminar scales with free-stream squared. The transition is primarily driven by the Reynolds number, curvature and turbulence and is therefore not, to a first order, in the hands of the researcher.

The end-wall related loss is a combination of a highly-skewed new thin boundary layer and dissipation of a secondary-flow structure. Secondary flows are typically defined as "any flow" that is not following the main streamlines by viscous action and a pressure gradient. One classic example is the tea-leafs that, when the tea is stirred, gathers at the centre. The tea leaves actually serves as flow marker for the boundary layer and shows the migration towards the centre. The flow in a turbine is indeed, a complicated un-steady and three-dimensional flow and the research requires a competent toolbox.

Another prominent example of viscous dissipation is the boundary layers on the blade profiles. A blade row accelerates the flow by turning it away from the axial direction and one should reason this in terms of blade lift (force and reaction force) – this turning requires a suction and pressure side to provide the force from one direction into another as dictated by the Newton second law. The flow is mostly locally accelerating but one typically has to lower the pressure at the suction side below the back pressure – hence a requirement to lower the velocity locally in order to fulfil the Kutta-condition (i.e. an off-loaded trailing edge). This process should for a properly deigned blade take place in the uncovered part after the cascades' throat. This deceleration is very lossy and may be on the order of 70…80 percent of the blade total profile loss. The lift duty is also proportional to the mass flow per pitch (i.e. the number of blades in a row). The underlying reason for the increased loss is the combination of boundary layer growth (due to the pressure gradient) and viscous dissipation. As mentioned earlier, the state of the boundary layer governs the loss production to a first order. A designer should therefore strive to optimize the blade with some optimum velocity distribution / diffusion strategy, for minimum loss [5]. This part is, to some extent, heuristic since an absolute numerical (or quantitative) loss-value may be erroneous. It is therefore typical to use a pre-defined desired velocity profile when designing a blade. The profiling philosophy also indirectly controls the secondary flow(s) at the end-walls. The reason for this that the blade lift produces the intra-blade pressure gradient, which causes the boundary layer migration (and other corner issues). Un-twisted blades will have local incidence (i.e. miss-alignment with the incoming flow). This will have an effect on the boundary layer growth early on the blade due to the displacement of the stagnation point.

Clearance flow is another source of entropy generation where the aero-designer has some means of mitigation. The typical approach is to reduce the pressure difference over the seal. One could consider two possible approaches by a combination of the reaction level and the seal technology and the reaction gradient. The latter is possible to optimize by combinations of wall-contouring and lean – and span-wise swirl distributions.

1.3 Research task

The overall research task is to improve the performance of an industrial size steam turbine, by introducing advanced blading technology.

The vehicle for the work is a pre-defined stage in a (fictive) Siemens SST-900 turbine. The parts can be summarized as:

- Axi-symmetric throughflow modelling of the stages in the turbine section with the tool AxCent™ from Concepts NREC.
- The second part is establishing a high-fidelity[2] CFD model for the actual stage and both neighbouring stages for capturing the actual unit. The actual stage has a real detailed seal set-up for the stator and the rotor. This task (and the previous to some extent) is to define a datum turbine model where all changes will be assessed from.
- The third part is still ongoing and the aim is to create the modified stage philosophy, whist maintain certain important features of a low-reaction turbine.

[2] Boundary layers resolved down to a y^+ value of unity.

1.4 Goal

The goal of the project can be summarized as:

- Optimized Blade Profiles
    - Inventory
    - Optimized prismatic
- Optimized Stacking
    - Technology
    - Where in the turbine?

The latter part (Optimized Stacking) is to create a design philosophy and increase the knowledge for high-performing turbine stages, by:

- A three-dimensional turbine stage philosophy for optimum performance [3], [6]:
    - Damped forced-vortex swirling, in combination with:
    - Compound lean
    - A combination of "parabolic forced-vortex" and compound lean.
- A cloning strategy for other stages

One could argue that there should be an infinite number of configurations available for setting up the design philosophy. The suggested combinations are, however, not set arbitrarily and could be considered as a heuristic approach based on the literature study. [1]

1.5 Project organisation

The project has been performed in collaboration between the division of Thermal Power Engineering (Lund University, LU) and Siemens Industrial Turbomachinery (SIT).

Project organization:

LU – Energy Sciences:

- Project lead
- Turbine modelling

Siemens Industrial Turbomachinery:

- Turbine performance data (including supplying the software)
- Detailed turbine geometry
- Allocating the Siemens turbine design tool to LU and providing training for the PhD-student
- Providing turbine design expertise from several senior staff and specialists
- Siemens internal development project with goal to improve turbine efficiency (Project lead: Claes Waksjö).

Budget follow up:

In SEK	for period	in kind	financed by KME
Reported	121001-130630	2’608’100 SEK 4742 h	
Reported	130701-131231	728’750 SEK 1325 h	
Original budget		2’895’000	1’900’000

The bulk of the work at Lund University has been carried out by the PhD-student and the supervisor.

The project was rewarded to Lund University in May 2012 but the PhD-student was not in place until November 2012. The reason for the delay was the expatriation process for the research student from India.

The involved personnel are listed below:

Project leader (LU): Magnus Genrup

Senior staff: Magnus Genrup

PhD-student:	Srikanth Deshpande (from 2012-11-01)
Stakeholders SIT:	
Project leader (SIT):	Markus Jöcker
Reference group[3]:	
SIT:	Markus Jöcker
	Arne Karlsson
	Lars Hedlund
	Åke Göransson
Vattenfall:	Maria Johnsson/Nicklas Simonsson
Lund University:	Magnus Genrup
	Marcus Thern

1.6 SIT Project "Improved Efficiency"

Project "Improved Efficiency" is a long term project devoted to development of design improvements in the steam path for the SST-700/900. The aim is to increase the internal turbine efficiency, while simultaneously strengthening the knowledge and skills for the R&D staff in the area of applied aero- and thermodynamics.

Main topic is currently blade development. Methods and tools used are the same or similar in the two projects, thus enabling synergies on a practical level. Training on SIT in-house tools has also been provided, as well as continuous support.

Exhaust diffuser design has been recognized as another field of improvement, where numerical studies have shown a potential efficiency gain. These studies have received external attention and a scientific paper is accepted for presentation at the 2014 ASME conference [11].

[3] The current report is revised (ad notam) per the comments from the steering committee.

2 Methods

The methods adopted for this research are based on numerical flow analysis.

It is was decided to study the datum turbine and establish a base-line that will serve as a reference for the continued work. The importance of a baseline case cannot be stretched too much since most changes alone will only introduce minute changes to the overall stage performance. This means that the modifications will be compared on a back-to-back basis within the same tool environment such as Ansys CFX.

2.1 Aerodynamic methods

The methods used in this project is briefly described in this section. The used methods are the same as in the turbomachinery industry (e.g. Siemens) and could be considered as standard practice. The procedure (Figure 1) follows a typical one-dimensional, two-dimensional, two-dimensional blade-to-blade and three-dimensional viscous (or CFD) modelling sequence.

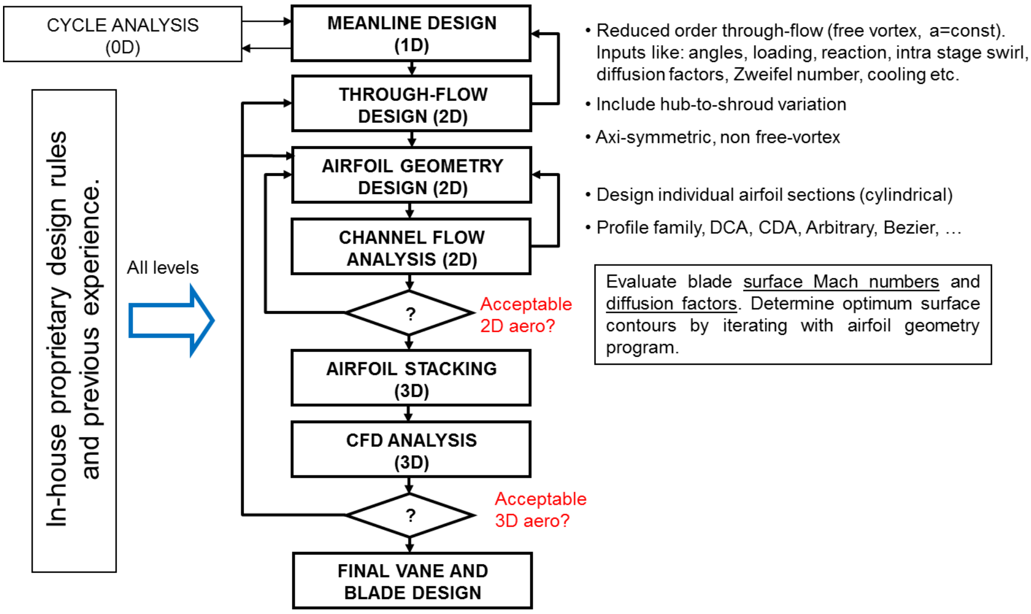


Figure 1. Typical turbine design loop [7]

One common project "boundary" is to maintain the overall turbine low-reaction design technology. The reaction sets the possible stage loading ($\Delta h_0/U^2$) and the shaft thrust force. The stage loading (i.e. how an individual stage can be loaded in terms of heat drop) is a function of the reaction and the inter-stage swirl angle and can be written as:

$$\frac{\Delta h_0}{U^2} = 2(1 - R - \varphi \cdot \tan\alpha_2) \qquad \text{Eq.1}$$

Steam turbines are typically designed without inter-stage swirl[4] and the preceding equation (Eq. 1) shows loadings of 2 and 1 for zero and 50 percent reaction, respectively. The exact level of reaction is proprietary and will not be discussed here. The reaction is in simple terms a gauge of the acceleration in the rotor relative to the stator. Hence, for a zero reaction the rotor acceleration would be zero. The other "standard" case is 50 percent (or symmetric) where the split between the rows is equal. By virtue of the turbine working principle, there exists a reaction gradient over the height. This means that the reaction at the hub (in practice) always should be less than for the mid-span. The design principle is to retain a certain rotor hub acceleration. This means that the reaction at the hub should have some minimum positive value.

Shaft thrust is either compensated for by the gear, or absorbed in the thrust bearing.

2.1.1 One-dimensional or reduced-order throughflow

This level of modelling is the backbone in turbomachinery modelling. The project has access to the Siemens in-house proprietary tool "AXIAL" or Y25018. This standard tool is used when a turbine is designed or analysed for a certain project. The program contains about 100 years of Finspong turbine collective in-house experience. This level does not include any detailed blade profile geometries and the modelling is based on developed proprietary correlations. The built-in models can be either generic with respect to losses and deviation or highly specific for the current turbine technology.

The one-dimensional level sets the mid-span datum with respect to average stage flow properties, such as: reaction, loading, flow coefficient, shaft thrust and thermodynamic properties.

2.1.2 S2-Throughflow

The two-dimensional axisymmetric throughflow level is the subsequent step in the analysis. The word throughflow is used for a class of tools where the flow is analysed in the meridional/span-wise. The purpose of using such a tool is to be able to describe spanwise variations – based on the previous mid-span values. This is also commonly referred to as the S2 approach after Wu. The throughflow equation provides the coupling between the meridional- and the tangential velocity, hence the flow angle. The method is used to find the span-wise variations of flow properties and is the last step before the profiling phase. The simplest form of the practical equation is shown below [8]:

[4] Axial reference plane

$$c_m \frac{dc_m}{dl} = \sin(\mu) c_m \frac{\partial c_m}{\partial m} + \cos(\mu) \frac{c_m^2}{r_c} - \frac{c_\theta}{r} \frac{d(rc_\theta)}{dl} + \frac{dh_0}{dl} - \\ - T\frac{ds}{dl} - \sin(\mu) F_m - \cos(\mu) F_n \qquad \text{Eq. 2}$$

Equation 2 can be solved numerically if the variation of the tangential velocity along the station (c_θ=f(l)) is given, or substituted by the flow angle variation in the spanwise direction. The equation is typically referred to as the SCM-equation after the second term on the right side. The equation may be re-casted to fit several input data types but an in-depth discussion is outside the scope of the current report. The numerical grid is course and one typically has 21 streamlines and 5 quasi-orthogonals to represent each blade. The equation should be parabolic from a strict mathematical view, but the second (curvature) term on the right side introduces ellipticity to the system. There is no need for time-marching since the meridional velocity must be super-sonic for all realistic cases. The equation is solved in a step-wise fashion starting from the mid-streamline, supplemented by an integration constant from the continuity equation.

The work has mainly been carried out in a more advanced type of throughflow tool AxCent™ by Concepts NREC. The overall principle of specifying the spanwise distributions of tangential velocity or flow angle, has been replaced by the "gauge angle" distribution over the height. The gauge angle is defined as α_{gauge}=acos(o/s) and describes the ratio of the throat in the passage to the pitch. The actual flow angle is to a great extent described by the gauge angle.

The fundamental difference between the previous SCM-method and the AxCent™ tool is the CFD-type of time-marching solver.

2.1.3 Profiling and blade-to-blade analysis

The next step is to describe the blade geometry at certain stations along the span. The span-wise data is taken from the previous throughflow step in the design loop. This part could be seen as developing an optimum blade profile in order to turn the flow from the inlet- to the exit angle. The blade geometry is also chiefly defined by the exit Mach number and, to some extent, the inlet Mach number level.

The blade geometry at each section is defined by higher-order Bezier-Bornstein sections, together with 12 basic profile parameters. The blade profile is created in a heuristic manner and a certain desired velocity distribution, rather than a certain minimum, loss value is the target. The reason for this approach is that the loss value is strongly governed by the transition point and it is therefore preferred to seek a certain velocity profile. The desired velocity profile is created by the higher-order Beziers to create suitable blade section curvature distributions.

Two tools are being used for this purpose, namely AxCent™ and the Siemens in-house tool CATO. Both tools works is a similar way and have their own set of pros and cons.

2.1.4 Viscous three-dimensional modelling

The high-fidelity three-dimensional modelling is carried out in the commercially available package Ansys-CFX. The mentioned tool is standard in the turbomachinery business, as well as in academia.

The AxCent package also includes an “early” CFD-solver for rapid flow field assessment.

2.2 Approach

Firstly, it was decided to study the datum turbine and establish a base-line that will serve as a reference for the continued work. The importance of a baseline case cannot be stretched too much since most changes alone will only introduce minute changes to the overall stage performance. This means that the modifications will be compared on a back-to-back basis within the same tool environment such as Ansys CFX.

The reference group meetings were used to align the investigations done at Lund University with the ongoing development work at Siemens with the goal to maximise synergies of the projects.

3 Results

The presented material in this section is based on the Siemens SST-900 unit. The turbine should be seen as a vehicle for the research but the results are general in nature for all low-reaction units.

All the results presented here correspond to datum geometry.

3.1 Literature study and proposed swirl philosophies

An elaborate literature review was done on the ways to improve on efficiency and an official report was issued [9]. Based on the literature review and discussions, it was decided to start with a damped forced vortex for the nozzle. The word "damped" is used in a sense that the full extent of the flow angel variation is limited to a certain value. The swirl philosophy is indicated in Figure 3.1. The second option that will be investigated is referred to as the "Brand X" by Rolls Royce [6]. The mentioned stacking technique is also referred to as "parabolic forced vortex". The vane is opened towards both end-walls and the principal underlying idea is:

i. Reduce (i.e. less swirl) the flow angles at end-walls and increase (i.e. more swirl) toward the mid-span.

ii. Exit mass flow moves toward the end-walls

iii. More lift toward the end-walls (i.e. a balance between larger mass flow and lower turning) and reduced lift towards the mid-span

iv. The improved mass flow distribution increases the stage efficiency and less over- and under turning

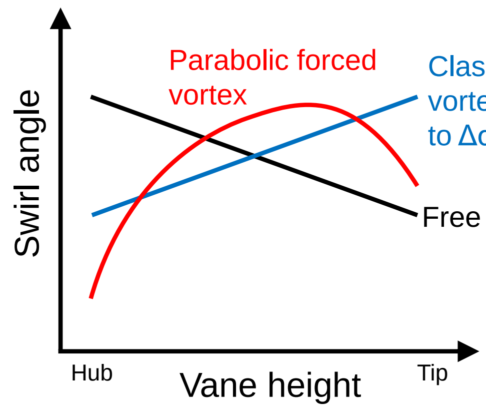


Figure 3.1 Proposed Swirl Philosophies

The design approach has been successfully used for high performance gas turbine designs. So, the first swirl philosophy that will be tried is the damped forced vortex philosophy.

On the other hand, rotor has lesser geometrical freedom since it has to be stacked on centre of gravity. Rotor philosophies which needed to be tried as indicated in Figure 3.2.

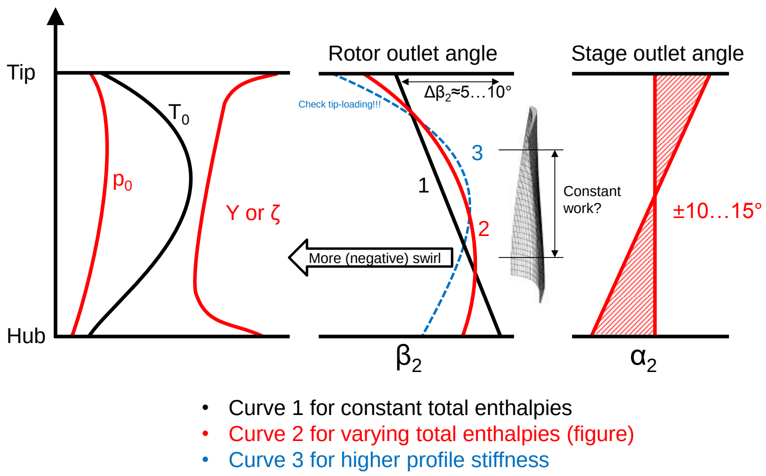


Figure 3.2 Proposed Rotor exit profiles

Baseline geometry to work was provided by SIT. Decision was to work on Intermediate pressure (IP) turbine of fictive SST900 steam turbine which has 15 stages. Since it is difficult to model the entire IP section, stage 21 was made the point of focus. All design modifications were to be tried on this stage. Initial plan was to give the through flow analysis. Later the scope was broadened and CFD analysis was also included.

3.2 Description of datum geometry

Stage 21 has the following geometrical features. Gauge angle distributions of stator and rotor blade are also shown in Figure 3.3.

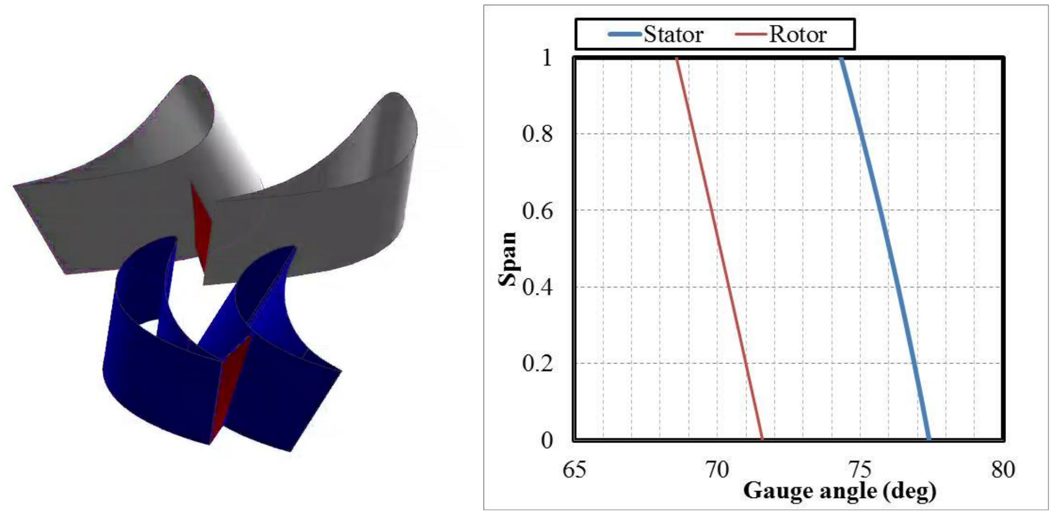


Figure 3.3 Datum Geometry detail of Stage 21

To capture the effects of boundary layers coming into the stage and also to avoid back pressure effects if any, a stage upstream and a stage downstream were added for the analysis. So, stages 20, 21 and 22 were modelled and analysed. The datum geometry has constant, or prismatic, blade sections. By virtue of that design philosophy, the rows shows the typical increased gauge angle towards the hub. This means that a constant angle stage resembles more a free-vortex type than a constant angle design [3]. This is all driven by the geometrical features (i.e. pitch-to-chord-ratio driven deviation) of the blade rows.

3.3 Through flow analysis in Axcent

With one dimensional boundary conditions obtained from SIT, 3 stage model was setup in Axcent. Fluid used was steam from NIST Refprop™. Meridional view of the model setup is as shown in figure. Losses were included in the form of built in correlations [10].

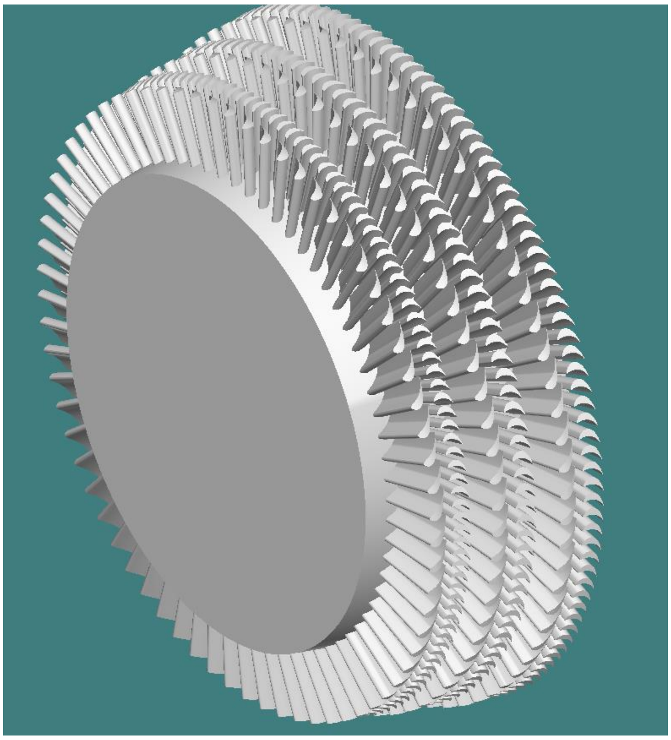


Figure 3.4 Datum Turbine setup in Axcent

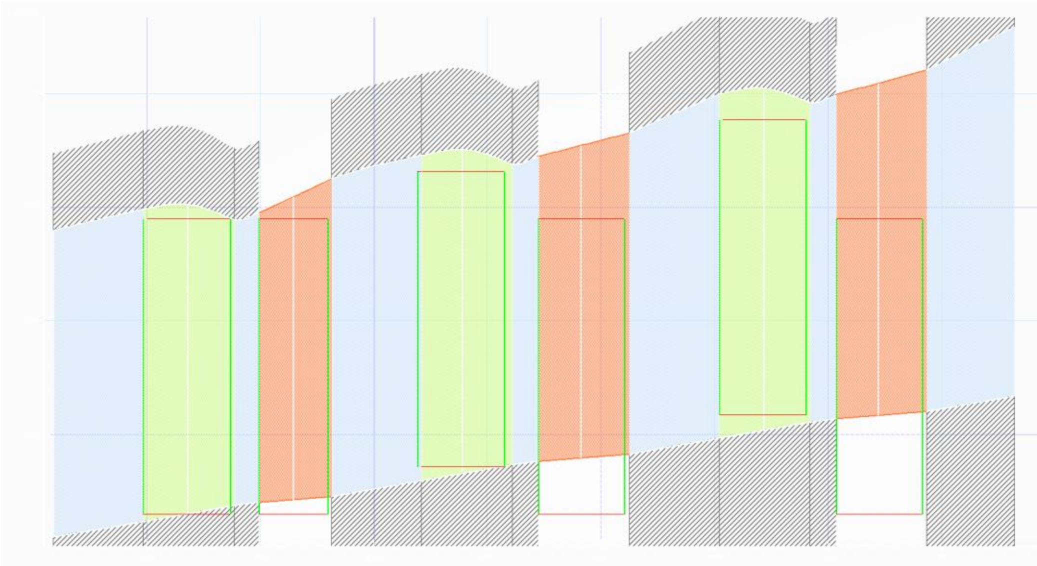


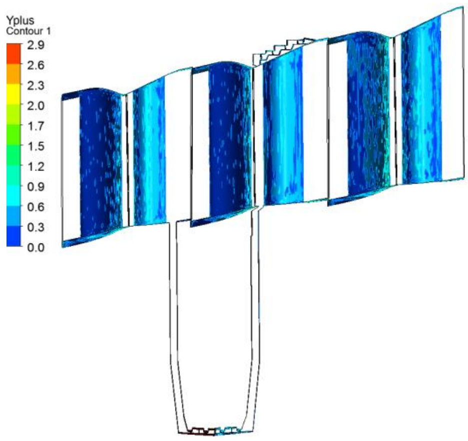
Figure 3.5 Through flow setup in Axcent

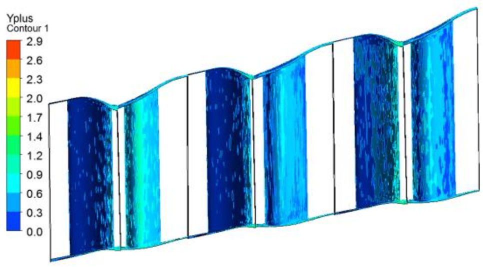
Unlike common through flow solvers, time marching solution to a fixed grid is used in AxCent™ through flow solver.

3.4 Three-dimensional CFD in CFX

Extending the scope of the project and in the interest of the project, it was decided to go for three dimensional CFD analysis. This was agreed in reference group meeting. CFX 14.0 was used for the purpose. Structured grid with determinant value around 0.45 were maintained. Y^+ was targeted to be around 1…2. Figure 3.6 shows the y^+ contours in the CFD models analysed. With this objective, the mesh size to be analysed went up to 20 million for 3 stages. To further add to details, root cavity and tip cavities were modelled with a two-fold purpose. One, to arrive at the amount of cavity leakage that happens and two, to study how the cavity flow mixing with the main flow happens and how much loss does it account for. Cavity geometries were obtained from actual turbine and few simplifications (like fillets not being considered) were done for ease of meshing.

Extensive meshing effort was carried out using ICEM-CFD.





el - with and without cavity

For the analysis, IAPWS-97 steam table was used and to arrive at the boundary conditions at inlet of stage 20 in analysis, CFD was run with higher y^+ on stages upstream. For all the CFD analyses, the k-ω SST turbulence model was used. Inlet boundary conditions for the upstream stages was obtained from old inlet volute analysis done at SIT. Figure 3.7

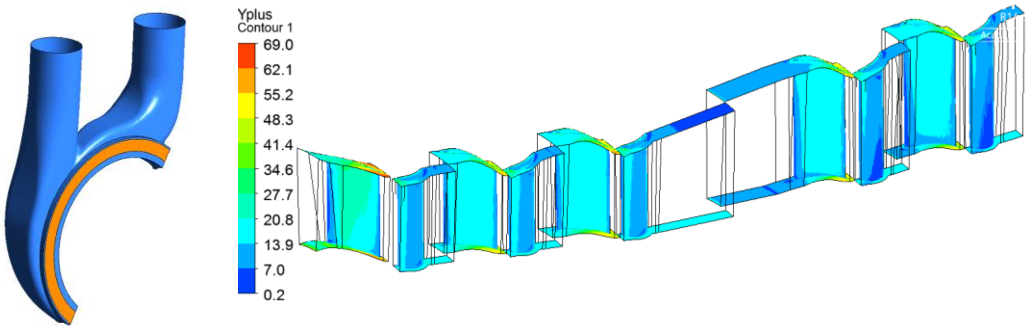


Figure 3.7 Five stage CFD to obtain inlet boundary condition

At this point, it is appropriate to compare the efficiency values obtained from AXIAL (1D code from SIT), Axcent™ TF and CFD analysis with and without cavities. Figure 3.8 shows the comparison

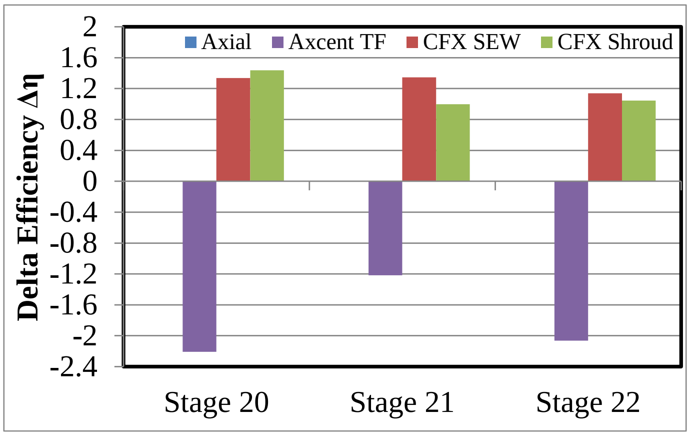


Figure 3.8 Efficiency Comparison

AxCent™ through flow under-predicts the efficiency when compared to AXIAL by quite a considerable value and CFX over-predicts the efficiency. One important thing to be noted here is that, the efficiency difference between three-dimensional CFD runs i.e without cavity(SEW) case and with cavity (CFX Shroud) case seems to be around 0.3% which is quite low. This point will be discussed further in chapter 4. Another purpose of this study was to set a target for efficiency improvement.

4 Analysis of the results

In this chapter, three studies carried out on the datum turbine are discussed. Each study helps to set realistic targets for every modification and what numbers to be aimed at. Also, blade loading plots in the datum turbine are discussed in the last section.

4.1 Effect of Tip leakage flow – Comparison of SEW and cavity case

Effect of tip leakage flow and losses incurred due to cavity flow mixing with main flow can be studied using comparison between smooth end wall (SEW) analysis and analysis of model with cavities. Two models are as shown before in Figure 3.6

Radial profiles at different axial stations provide good indication to study effect of cavity flows. Since the stage to focus is just stage 21, radial profiles at stage 21 inlet, stator exit and stage exit are reported here. Flow variables in the graphs are total pressure, static pressure, swirl angle, flow angle, Mach number and axial velocity. Swirl direction follows the right hand thumb rule convention where thumb points the direction of main flow and curled fingers indicate positive swirl. Also, meridional flow angle phi is defined with the help of ratio of radial velocity to axial velocity. Flow angle or phi angle which is defined as Phi = $\tan^{-1}(C_r/C_m)$ is another angle which is shown.

At stage 21 inlet, effect of root cavity is not evident in radial profiles. Apart from some minute difference in swirl angle at root, there is nothing much to take away between the radial profiles of two cases.

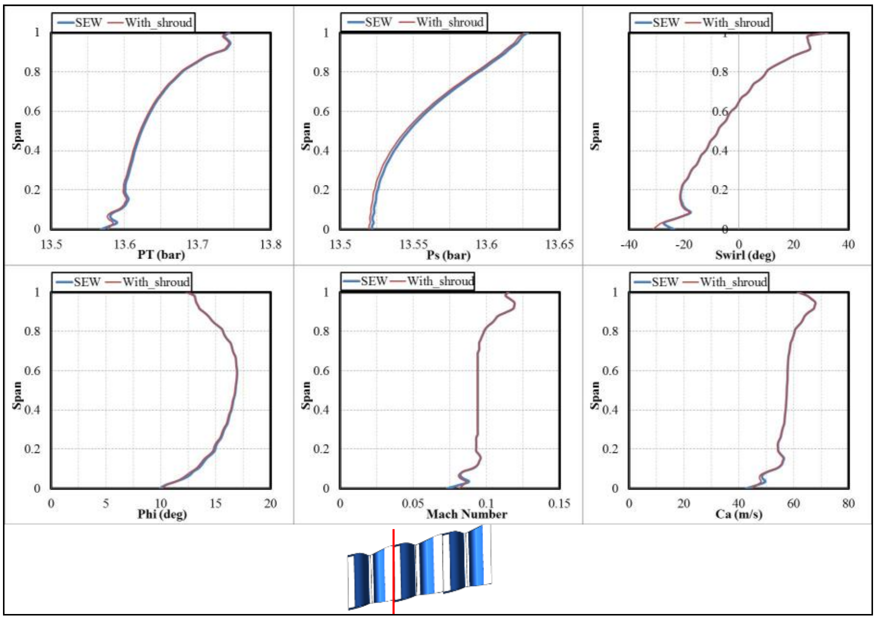


Figure 4.1 Radial Profiles at Stage 21 Inlet

At stage 21 stator exit, effect of root cavity can be seen significantly. Considerable changes in swirl and total pressure values between SEW and with cavity case. Root gap of 0.8 mm at radius of 275 mm as in actual machine was used. And the mass flow rate through root cavity is 0.5 kg/s of 118.3 kg/s.

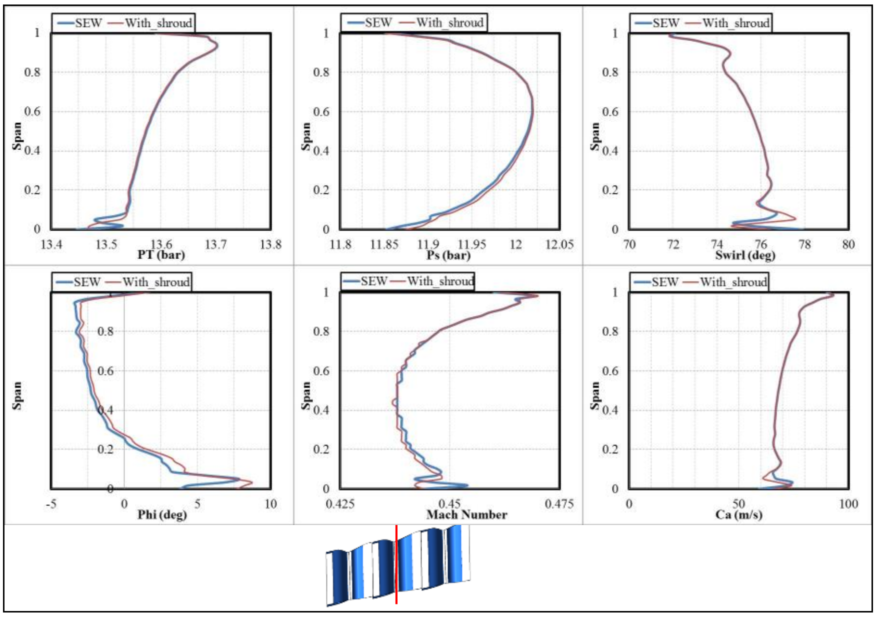


Figure 4.2 Radial Profiles at stage 21 stator exit

At the exit of the stage, it was expected that the radial profiles would be different between the two cases at tip. But the radial profiles does not show any change between the two cases at the tip. This can be attributed to two reasons. First is the kind of entry of cavity flow into the main flow that is occurring at the tip. It is more meridional entry into the main flow rather than a radial entry that makes it less effective. Second reason being the cavity mass flow is just 0.4 kg/s of 118.3 kg/s, which corresponds to approximately 0.3 percent.

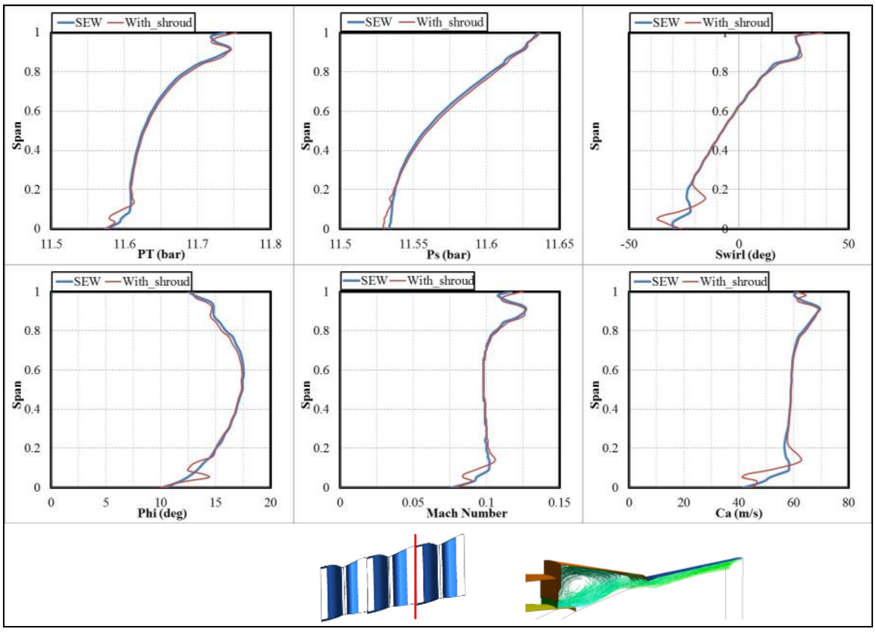


Figure 4.3 Radial profiles at stage 21 exit

Also, as seen before, the efficiency comparison between the two cases is quite small i.e. around 0.3 percent and hence can be concluded that for this particular geometry and for this particular cavity configuration, not much of loss is happening due to cavity or leakage loss.

4.2 Effect of inlet swirl – normal to BC case and 3 stage SEW case

One of the features of the current design, was that in the baseline design, gradient in swirl of flow coming into stage 21 from root to tip was too high. As can be seen from Figure 4.4 swirl variation is around -60° from root to tip. To study the penalty of such an incidence, stage 21 alone was analysed with zero inlet swirl boundary condition and the results were compared. It was found that stage with no inlet swirl has better efficiency by 0.37 percent when compared to high swirl gradient at Inlet. This is the potential number that can be achieved by improving the inlet swirl gradient.

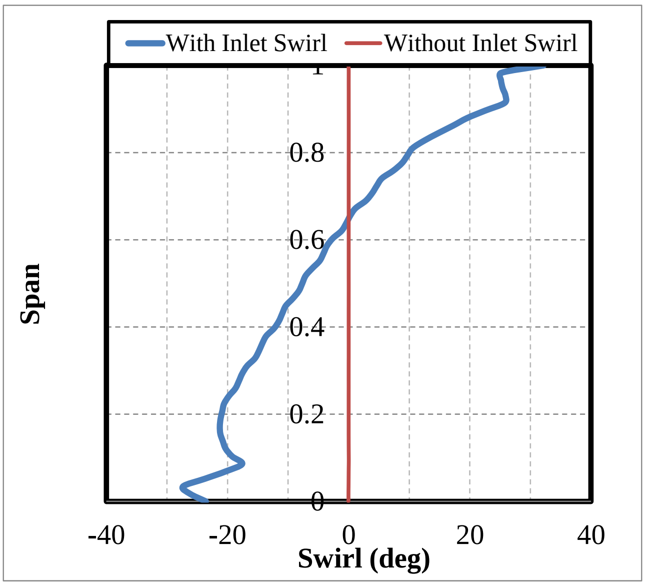


Figure 4.4 Inlet Swirl to Stage 21 Stator

4.3 Contribution of secondary loss – Free slip wall case and No-slip wall case

To study the effect of secondary loss and to reduce it, firstly the secondary losses itself needed to be quantified. All the swirl philosophies proposed in chapter three are aimed towards reducing secondary losses and hence increase efficiency. So, it is important to know how much can the maximum gain be. One of the ways to arrive at the secondary loss number is to do an analysis where hub and casing walls are defined as free slip walls and then compare the results with smooth end wall case where in the walls are no slip walls. When this study was done, results showed that in stage 21, increase in efficiency when secondary flows are completely eliminated is 1.2 %. This number helps us to decide on realistic target with the vortexing and stacking methodologies.

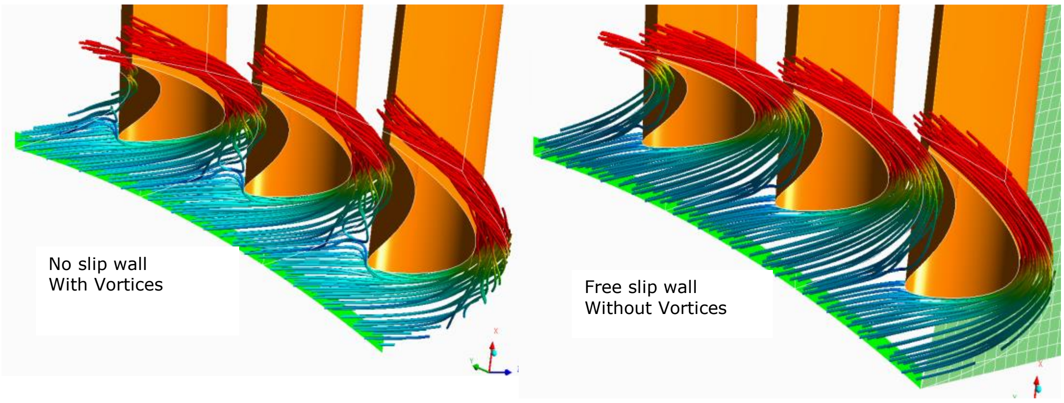


Figure 4.5 Secondary Flow

4.4 Blade Loading

From the three-dimensional CFD smooth end wall runs, blade loading plots were obtained. For Stator of stage 21, Figure 4.6 Blade Loading - Stator stage 21 gives the blade loading plots. Also, seen on the left is the curvature distribution for the same. With the curvature distribution being good, Mach number distribution also looks acceptable and hence it was decided to retain the same airfoil for stator for initial trials in the project.

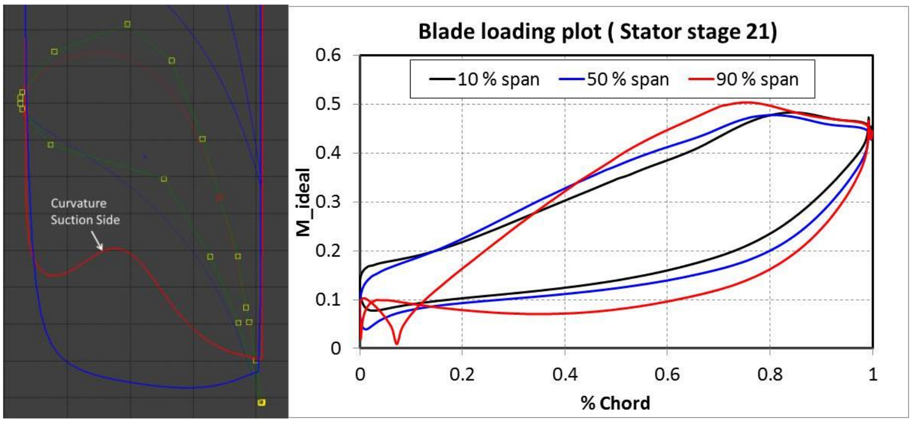


Figure 4.6 Blade Loading - Stator stage 21

For rotor blade of stage 21, blade loading plots showed few areas which needs to be focussed on. Figure 4.7 shows blade loading plots for rotor. It can be observed that diffusion factor on the suction surface being close to around 0.26 is quite acceptable for the exit Mach number of around 0.4. But the point of concern is the diffusion rate on the suction surface. Diffusion happens in last 5 % chord length of the airfoil which is high diffusion rate and is a cause for loss.

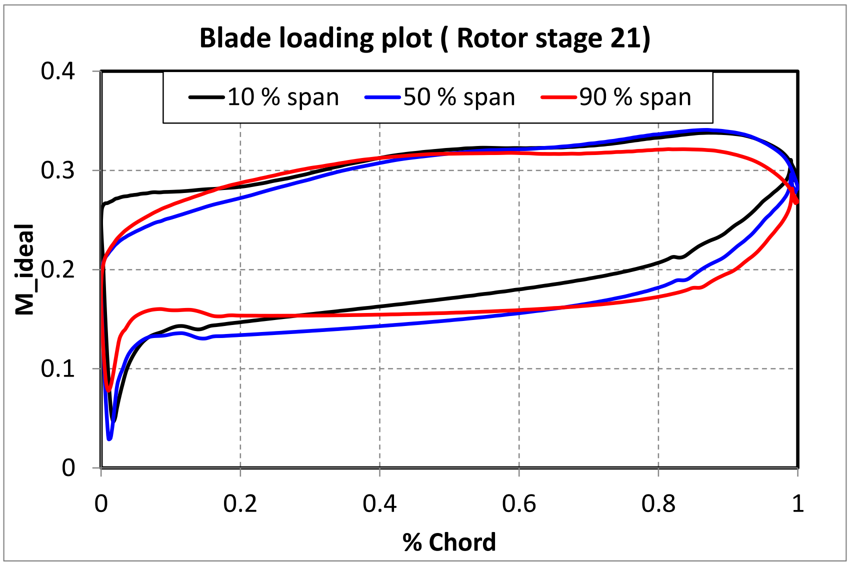


Figure 4.7 Blade Loading - Rotor stage 21

To confirm this trend of the Mach number distribution, Three-dimensional CFD blade loading at mid span was compared to Mach number distribution obtained from MISES run in CATO (Airfoil design tool used in SIT). Figure 4.8 shows the comparison of suction side Mach number distribution. Difference in Mach number till 0.5 % of chord length can be attributed to the fact that in Three-dimensional CFD, 50 % span gives meridional 50 % span where as in MISES, the airfoil was designed at constant radius. An important point to be taken from the figure is that the diffusion rate matches in both numerical calculations.

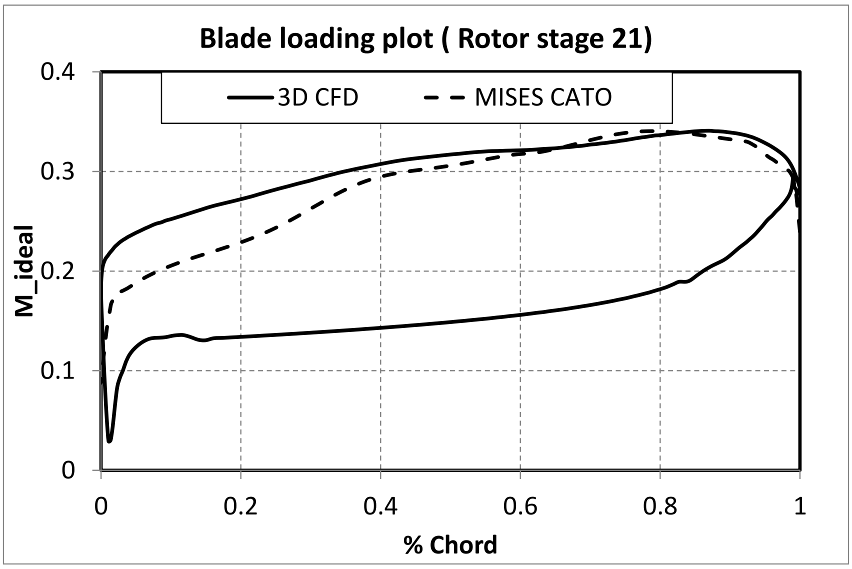


Figure 4.8 Blade Loading comparison- 3D CFD and MISES

In order to improve the diffusion rate and for better Mach number distribution, trials are being done on the rotor airfoil in MISES. Presently in two dimensional form, these profiles need to be tested in Three-dimensional environment. Figure 4.9 Blade loading - Modified Rotor Airfoil gives the improved blade loading plot. The modified Mach number distribution is the preliminary modified one as on date. Still the modification is ongoing and the final Mach number distribution will be further improved.

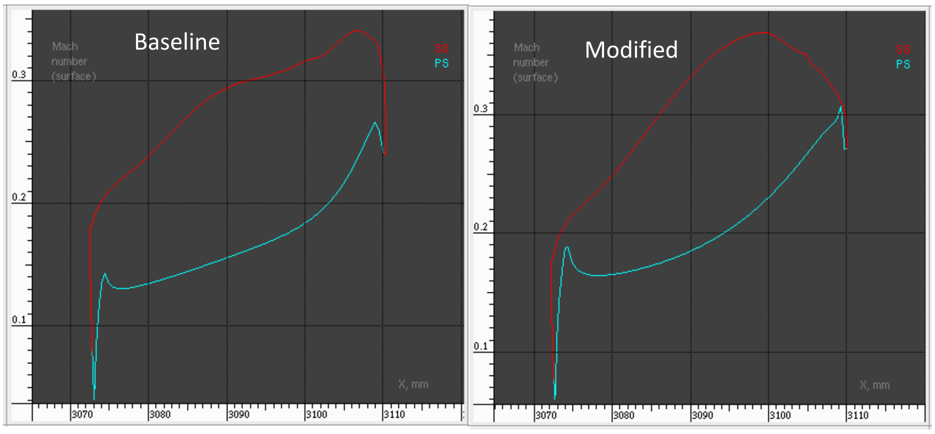


Figure 4.9 Blade loading - Modified Rotor Airfoil

5 Conclusions

The project is still has substantial on-going work and most of the conclusions are based on the datum turbine geometry.

The reference stage is a fairly tall stage with a high efficiency already at the present design. The presented numbers are valid for that configuration and will be higher for stages with a lower volumetric flow (aspect ratio).

For the stages considered, cavity flow losses are low. The difference between the one with cavity and without cavity case is just 0.3 percent.

Improving the exit swirl from the rotor has a benefit on the downstream stage and the benefit is around 0.37 percent.

If all the secondary flows in the main flow path is eliminated, the gain is efficiency is 1.2 percent for stage 21. Thus, it is the maximum gain that can be obtained for a tall, or high aspect ratio, stage.

Based on this, realistic targets for swirl philosophies to reduce secondary losses can be set. The results should be treated with a nuanced caution since it is not possible to draw firm conclusions on a back-to-back basis at this stage. As already mentioned, the results are based on stage 21 and an earlier stage will have an even higher efficiency improvement potential.

5.1 End-user perspective and benefits

Most of the report has been focused on various technical aspects and the aim of this section is to describe and clarify the end-user perspective and benefits.

The end-user perspective is typically the plant performance and not necessarily details related to the flowpath inside the turbine. The wording “turbine performance” includes/means: efficiency, swallowing capacity (or power output), etc. Hence, indeed important parameters for the profitability of a plant. The technology level within the turbine translates into the level of performance – but also comes with a cost. In the past, the turbine technology development was partly driven by cost and most development efforts were towards lower first cost. Today, modern manufacturing technology has levelled the costs between two- and five-axis machining. This means, today, that a more advanced turbine does not necessarily comes at a higher cost.

This means that, from an end-user perspective, the more advanced aerodynamics from the current research project results in a higher-performing turbine and thereby lower life-cycle cost.

6 Goal fulfilment

Detailed analysis of the baseline case and contributions of tip leakage loss, secondary loss have been established.

Blade profiling is completed and the swirl philosophies have to be modelled and analysed.

The goal of the project can be summarized as "exploring the future flow path for higher-performing steam turbine plants" by evaluating the maximum cost effective efficiency potential of industrial steam turbines. The latter is by utilizing improved blade profiles and advanced stacking technologies. The work shows that there is a potential for improved efficiency by introducing more advance blading.

The scope of the project can be summarized as:

- Optimized Blade Profiles
  - Inventory
  - Optimized prismatic
- Optimized Stacking
  - Technology
  - Where in the turbine?

The initial scope has been extended by high-fidelity CFD since it was necessary to establish a base-line for the research work. The flow field in a turbine is, indeed, complicated and "additional" flow path features such as seals will have a significant influence. It is important to recognize that e.g. changes to the swirl philosophy may influence the efficiency through changed seal flows, rather than pure blading aerodynamics.

The first part is completed whilst the second part of the work is still ongoing. The second part will be completed according to the plan before 1/6-2014. The work during this period will be reported within the next project phase. The delay is due to the requirement- and expatriation process of the PhD-candidate. The candidate has a strong background in CFD-modelling of steam turbines. This is the prime reason for adopting more advanced calculations such as high-fidelity CFD-modelling, rather than just focusing on the lower-order tools as in the initial plan. The delay is not due to the additional calculation level and is only due to the requirement process. Once the rotor work is complete, the project goals should be considered as completed.

7 Suggestions for future research work

The work is still ongoing and the plan is to develop the mentioned blading philosophies. The turbine work has so far been limited to a fairly tall and high-performing stage. Future work should include a cloning strategy so that advanced blades can be introduced even earlier in the turbine. The work has identified variants of “damped” controlled vortex in concert with lean as the most promising candidates. A natural continuation is to explore the finding in the context of a low-reaction turbine in order to maximize the benefits.

Most of the work will be based on back-to-back comparisons with a datum turbine. It should be recognised that most powerful CFD-programs of today have issues related to accurate quantitative assessment of the flow field features and modifications. It may therefore be, as a final assessment, be necessary to run a test turbine in order to quantify the actual gain, in firm numbers.

8 Literature references

[1] A. Schaarschmidt et al., ”Performance Increase Through World Class Technology and Implementation”

[2] Denton J. D., “Loss Mechanisms in Turbomachines – 1993 IGTI scholar lecture”, ASME Journal of Turbomachinery, October 1993, Vol. 115/**621**

[3] Denton J. D. et al, ”Cambridge Turbomachinery Course”, Vol. 1 & 2, 2012

[4] Greitzer E. M., Tan C. S., Graf M.B., ”Internal Flow – Concepts and Applications”, ISBN 0-521-34393-3, Cambridge University Press, 2004

[5] Moustapha H., Zelesky M., Baines N., Japikse D., “Axial and Radial Turbines”, ISBN 0-933283-12-0,Concepts NREC, 2003

[6] Haselbach F., Taylor M., “Axial Flow High Pressure Turbine Design”, Von Karman Institute (VKI) lecture notes, 2013

[7] Genrup M., Lecture Notes (steam- and gas turbine technology, MVK051), Lund University, 2013

[8] Hearsey R. M., “A revised Computer Program for Axial Compressor Design”, Volume I, ARL 75-0001 Vol. 1, 1974

[9] Deshpande. S., “Literature Study for KME 607”, 2013

[10] AxCent™ Documentation

[11] Lei Xu, Lars Hedlund, Åsa Nilsson; 2014 ”CFD Investigation of the Influence of Inflow Conditions on an Aggressive Axial Exhaust Diffuser for Steam Turbines”; to be presented at ASME Turbo Expo 2014

Internal reports at Siemens Industrial Turbomachinery AB

RT RS 141/12; Improved efficiency - Findings in flow capacity and degree of reaction; 121121

RT RS 12/13; Improved efficiency: Through flow CFD-analysis of Meridional Profiled Short Stages; 130205

RT RS 13/13; Improved Efficiency: Through flow CFD-Analysis of Twisted Long Stages; 130205

RT RS 14/13; Improved Efficiency: CFD Study of Correlations between Turbine Blades with Meridional Profiled Endwall and Straight Endwall; 130205

RT RS 79/13; Improved Efficiency: CFD-Analysis of Axial Exhaust Diffuser C761 Geometry Modifications; 130625

RT RS 80/13;Improved Efficiency: CFD-Analysis of Axial Exhaust Diffuser C583 and Modifications; 130625

RT RS 8/14; ZIA236 Improved Efficiency: CFD evaluation of potential of improved blading; 140122

RT RS 15/14; ZIA236 Improved Efficiency: SST-700/900 Twisted Blade Pre-Study

MoM of reference group meetings

2012-05-29: Kick off meeting, PMRS 11/12

2012-09-19: Reference group meeting, PMRS 26/12

2013-01-16: Introduction meeting for Srikanth Deshpande, PMRS 1/13

2013-03-27: Reference group meeting, PMRS 13/13

2013-10-09: Reference group meeting, PMRS 23/13

Status reports to KME

121122 Lägesrapport 1

130325 Executive summary

130703 In-kind report 1 from SIT AB

131014 Lägesrapport 2

9 Publications

Lei Xu, Lars Hedlund, Åsa Nilsson; 2014 ”CFD Investigation of the Influence of Inflow Conditions on an Aggressive Axial Exhaust Diffuser for Steam Turbines”; to be presented at ASME Turbo Expo 2014.

Sammak, M., Deshpande, S., Genrup, M., 2014,“Oxy-fuel turbine through flow design”, ASME Power, Baltimore, Maryland, USA.

ELFORSK

SVENSKA ELFÖRETAGENS FORSKNINGS- OCH UTVECKLINGS - ELFORSK - AB

Elforsk AB, 101 53 Stockholm. Besöksadress: Olof Palmes Gata 31
Telefon: 08-677 25 30, Telefax: 08-677 25 35
www.elforsk.se