

Transient CFD-analysis of a high head Francis turbine

Ruben Arne Christoph Moritz

Master of Science in Mechanical Engineering Submission date: June 2014 Supervisor: Michel Jose Cervantes, EPT

Norwegian University of Science and Technology Department of Energy and Process Engineering



Norwegian University of Science and Technology

Department of Energy and Process Engineering

EPT-M-2014-74

MASTER THESIS

for

Ruben Moritz

Spring 2014

Transient CFD-analysis of a high head Francis turbine

Transient CFD-analyse av en høytrykks Francis turbin

Background

Francis-99 is a set of upcoming workshops jointly organized by the Norwegian University of Science and Technology (NTNU), Norway and Luleå University of Technology (LTU), Sweden in the same spirit as the previous Turbine-99 workshops. The Francis-99 workshops aim during the coming years to determine the state of the art of high head Francis turbine simulations under steady and transient operating conditions as well as promote their development and knowledge dissemination openly.

A high head Francis turbine model, named the Tokke model, has been designed and experimentally investigated in the Waterpower Laboratory at NTNU. The complete geometry of the model and mesh are now freely available on the present site. NTNU and LTU expect this geometry to become with time a reference test case to the hydraulic community for research and development and the workshops a meeting place to discuss developments, potentials, issue on a common and open test case.

The student will carry out CFD-analysis on the existing turbine geometry, mainly on the runner and draft tube in order to investigate the transient flow characteristics in the draft tube. Another student will carry out LDV-measurements in the draft tube and they will work together on both CFD-analysis and LDV-measurements.

Objective

Carry out a non-stationary CFD-analysis of the flow in the draft tube of a high head Francis turbine and compare with measurements from model tests in the laboratory.

The following tasks are to be considered:

- 1. Literature survey
 - a. Hydraulic design of Francis turbines runners and draft tubes
- 2. Software knowledge
 - a. Get familiar with the CFD-tool for mesh generation and numerical simulation; Ansys
- 3. CFD-analysis
 - a. Stationary and transient analysis of the runner and draft tube
- 4. Model tests (this will be carried out together with another student)
 - a. Performance of the model turbine
 - b. LDV-measurements in the draft tube
 - c. Comparison of the LDV-measurements and CFD-analysis

Within 14 days of receiving the written text on the master thesis, the candidate shall submit a research plan for his project to the department.

When the thesis is evaluated, emphasis is put on processing of the results, and that they are presented in tabular and/or graphic form in a clear manner, and that they are analyzed carefully.

The thesis should be formulated as a research report with summary both in English and Norwegian, conclusion, literature references, table of contents etc. During the preparation of the text, the candidate should make an effort to produce a well-structured and easily readable report. In order to ease the evaluation of the thesis, it is important that the cross-references are correct. In the making of the report, strong emphasis should be placed on both a thorough discussion of the results and an orderly presentation.

The candidate is requested to initiate and keep close contact with his/her academic supervisor(s) throughout the working period. The candidate must follow the rules and regulations of NTNU as well as passive directions given by the Department of Energy and Process Engineering.

Risk assessment of the candidate's work shall be carried out according to the department's procedures. The risk assessment must be documented and included as part of the final report. Events related to the candidate's work adversely affecting the health, safety or security, must be documented and included as part of the final report. If the documentation on risk assessment represents a large number of pages, the full version is to be submitted electronically to the supervisor and an excerpt is included in the report.

Pursuant to "Regulations concerning the supplementary provisions to the technology study program/Master of Science" at NTNU §20, the Department reserves the permission to utilize all the results and data for teaching and research purposes as well as in future publications.

The final report is to be submitted digitally in DAIM. An executive summary of the thesis including title, student's name, supervisor's name, year, department name, and NTNU's logo and name, shall be submitted to the department as a separate pdf file. Based on an agreement with the supervisor, the final report and other material and documents may be given to the supervisor in digital format.

Work to be done in the Waterpower laboratory Field work

Department of Energy and Process Engineering, 14. January 2014

Olav Bolland Department Head

Research Advisor: Professor Ole Gunnar Dahlhaug

Michel Cervantes

Academic Supervisor

Preface

The present work has been written as the master thesis required for the Master of Science degree in Mechanical Engineering at NTNU. The work was conducted in the spring semester 2014 at the hydro power lab at NTNU. The thesis has been written as a scientific paper.

I would like to thank Michel Cervantes and Ole Gunnar Dahlhaug for their supervision and inspiration throughout the work. I also want to thank PhD candidate Bjørn Winther Solemslie for always being helpful. A special thanks goes to all employees and my fellow students at the hydro power lab at NTNU for making my last year as a student a really memorable experience.

Ruben Monte

Ruben Moritz, June 10, 2014

Abstract

The purpose of the present work has been to investigate the ability of the profile transformation method in ANSYS CFX to give realistic inlet boundary conditions to a draft tube without using excessive amounts of computational resources.

A mesh supplied for the Francis-99 workshops was cut and modified. Transient simulations at the best efficiency point were conducted on the modified mesh containing one guide vane, one runner blade, one splitter and the draft tube. The results were compared to results from a simple draft tube simulation, a transient rotor-stator simulation, laser Doppler velocimetry measurements and pressure measurements. The profile transformation method simulation was done for one and seven runner rotations to evaluate the convergence. It was found that the profile transformation simulation used far less computational resourced compared to the transient rotor-stator simulation but the flow in the draft tube was not as well captured.

The transformation method approach has potential but several sources of error must be further investigated to conclude on the methods suitability for application to high head Francis turbines.

Sammendrag

Hensikten med det presenterte arbeidet har vært å undersøke evnen til profile transfromation method i ANSYS CFX til å gi realistiske innløpsbetingelser til et sugerør uten å bruke store mengder beregningsressurser.

Et mesh laget for Francis-99 workshopene ble kuttet og endret. Tidsavhengige simuleringer på beste driftspunkt ble gjennomført på de modifiserte meshene som består av en ledeskovel, en løpehjulsskovel, en halvskovel og sugerøret. Resultatene ble sammenlignet med resultatene fra en enkel sugerørsimulering, en transient rotor-stator-simulering, laser dopplervelosimetri målinger og trykkmålinger. Profile transformation method simuleringen ble gjort for ett og syv løpehjulsrotasjoner for å evaluere konvergens. Det viste seg at profile transformation method bruker langt mindre beregningsressurser i forhold til den transiente rotor-statorsimuleringen, men strømningen i sugerøret ble ikke beregnet så godt.

Transformation method tilnærmingen har potensial, men flere feilkilder må undersøkes nærmere for å kunne konkludere for metodens egnethet for bruk med Francisturbiner med høy fallhøyde.

Transient CFD-analysis of a high head Francis turbine

Ruben Moritz and Michel J. Cervantes

Abstract—The purpose of the present work has been to investigate the ability of the profile transformation method in ANSYS CFX to give realistic inlet boundary conditions to a draft tube without using excessive amounts of computational resources. A mesh supplied for the Francis-99 workshops was cut and modified. Transient simulations at the best efficiency point were conducted on the modified mesh containing one guide vane, one runner blade, one splitter and the draft tube. The results were compared to results from a simple draft tube simulation, a transient rotor-stator simulation, laser Doppler velocimetry measurements and pressure measurements. The profile transformation method simulation was done for one and seven runner rotations to evaluate the convergence. It was found that the profile transformation simulation used far less computational resourced compared to the transient rotor-stator simulation but the flow in the draft tube was not as well captured. The transformation method approach has potential but several sources of error must be further investigated to conclude on the methods suitability for application to high head Francis turbines.

I. NOMENCLATURE

BEP	Best efficiency point
CFD	Computational fluid dynamics
CPU	Central processing unit (in a computer)
DT	Draft tube
FFT	Fast fourier transform
GGI	General grid interface
GV	Guide vane
LDV	Laser Doppler velocimetry
\mathbf{PR}	Pressure recovery
\mathbf{PT}	Profile transformation
RAM	Random access memory (in a computer)
RMS	Root mean square
SST	Shear stress transport
TRS	Transient rotor-stator
$\triangle t$	Time step [s]

		-	L J		
$\triangle x$	Length	of	mesh	cell	[m]

- f.l.t First layer thickness of mesh at wall [mm]
- ν Kinematic viscosity [m²/s]
- P Pressure [Pa]
- ρ Water density $[kg/m^3]$
- RPS Rotations per second [/s]
- t time [s]
- au Wall shear stress [Pa]
- v Velocity [m/s]

$$y+ \sqrt{\frac{\tau}{\rho}} \frac{f.l.t.}{v}$$

Subscripts

1 Draft tube inlet

- 2 Draft tube outlet
- all In axial, tangential and radial direction

II. INTRODUCTION

The draft tube (DT) is an important part of a hydro power plant as it increases the effective head over the runner and affects the dynamic behaviour of the whole system. The flow in a DT is complex and time dependent. Computational fluid dynamics (CFD) is widely used in the development and the design of hydro power plant components including the DT. The inlet boundary conditions are crucial in such simulations, especially for transient simulations. It has become very important to study the transient effects in hydro power components as hydro power plants increasingly are used outside their best efficiency point (BEP) where transient effects have a large impact on the machine reliability and performance.

A transient rotor stator (TRS) simulation from the spiral casing inlet to the DT outlet is necessary to capture most of the transient effects. This requires a large mesh and a transient simulation consuming a lot of computer resources and time.

Such a simulation has been conducted by Trivedi [1] and the results were compared with experimental measurements. It was found that the results were evenly matched, specially at the BEP. The simulations were done on a cluster with 84 processors and took a full week to complete.

It is common practice to simulate a DT with no upstream components or with a limited part of the upstream components to make the simulation feasible. Simulations done on the geometry of the Turbine-99 [2] workshops by for example Breivik [3] and Moritz [4] were similar to many Turbine-99 simulations; steady state simulations done with given inlet boundary conditions consisting of a experimental velocity profile at the inlet. Simulations done with both the Turbine-99 runner and DT have been carried out by Jedvik [5] and he concludes that there is a strong interaction between the runner and the Kaplan DT.

These simplifications allow using far less computational resources compared to TRS simulations but compromise on accuracy by not capturing transient effects or by not resolving the interaction between the DT and upstream components.

It is desirable to find a way to do relatively fast transient simulations of a DT without compromising on accuracy. One way is to use one or a few runner blades and guide vanes (GV) and let the software copy the results to give a solutions for the whole assembly. This way the components directly upstream from the DT could be simulated with a far smaller mesh. The problem has been that unequal pitch between GVs and runner prevents this from being done. ANSYS has a solution to this problem in CFX called *transformation methods* which makes it possible to simulate parts of the runner and GV regardless of unequal pitch.

Research has been done on transformation methods by Connel et al. [6] and Zori et al. [7]. The profile transformation (PT), fourier transformation and time transformation methods were used to simulate a high pressure power turbine stage and a low pressure aircraft engine turbine stage [6], and PT and time transformation methods were used on a transonic compressor stage [7]. The results were compared to TRS simulations done on the same geometries. Connel et al. and Zori et al. conclude that each of the transformation methods has its positive and negative sides, but all of them reduce the computation time significantly while having small impact on the accuracy compared to TRS simulations.

The present work investigates the inlet conditions the PT method can give for a hydro power DT and the computational resources needed for this method compared to TRS simulations. Simulations using the PT method were conducted for the Francis-99 geometry. The results were compared to a simulation of the DT alone, a TRS simulation and experimental laser Doppler velocimetry (LDV) and pressure measurements. Additionally the computational time for the PT simulation was compared to the TRS simulation.

III. EXPERIMENTAL SETUP

The Tokke power plant was built in the early 1960s and consists of 4 Francis turbines which produce a total power of 430 MW [8]. The turbines operate at a head of 377 m and a flow of 32 m^3/s at the BEP [9, p. 30]. The model used in the present work is going to be used in the upcoming Francis-99 workshops and will from now on be called the Francis-99 model. It is scaled down by 1:5.1 compared to the turbines used in Tokke. This model has a slightly different runner design compared to the full size Tokke runner because it was designed by NTNU while the prototype runner was designed by Andritz. Both the Francis-99 model and prototype have a spiral casing with 14 stay vanes, a wicket gate with 28 GVs, a Francis runner with 15 blades and 15 splitters, and an elbow type DT. The BEP of the Francis-99 turbine has been defined with a GV angle of 9.84° , a rotational speed of 5.59 Hz, a



Figure 1. Setup of the Francis-99 model at NTNU



Figure 2. Lines used for LDV measurements and points used for pressure measurements [11]

mass flow rate of 203 kg/s and a net head of 11.91 m [10]. The Francis-99 test rig installed at NTNU is represented in figure 1.

LDV measurements on the Francis-99 model DT were performed by Sundstrom et al. [11] using the closed loop configuration of the NTNU test rig (from now on referred to as LDV measurements). The measurements were done along two horisontal lines in the DT cone as shown in figure 2. Section 1 and 2 were placed 64 and 382 mm below the DT inlet respectively where the radius was 177.5 and 196.2 mm in section 1 and 2, respectively. The axial velocity was defined as positive upwards (towards the runner) and the tangential velocity was defined as positive in the counter clockwise direction (seen from above). Sundstrom et al. [11] do not mention the circumferential position of sector 1 and 2. The measurements at the BEP were done at 16 points from the DT wall to the centre of the DT and all the velocity measurements were time averaged over a 720 s period. The measurements were done slightly off the BEP values defined for the Francis-99 workshop; a mass flow rate of 207 kg/s instead of 203 kg/s, a runner rotation of 5.74 Hz instead of 5.59 Hz and a head of 12.77 m instead of 11.91 m were used [11], [10].

Pressure measurements have been done by Trivedi et al. [1] on the Francis-99 model turbine and are available at the Francis-99 website [10]. These measurements were done using the open loop configuration of the NTNU test rig and 2 points in the DT called DT21 and DT11 defined on the Francis-99 website were used. The placement of DT11 and DT21 can be found in figure 2. The experimental measurements were done at a sampling rate of 2083.33 Hz (\approx every 4.80E-4th second or 0.97° runner rotation) over a time period of 10 seconds. The pressure was measured as total static pressure while there was atmospheric pressure in the DT tank with a water level approximately at the same level as the turbine centreline. These measurements were done at the BEP as defined for the Francis-99 workshop [10]

A TRS simulation of the Francis-99 model was conducted by Chirag Trivedi [1]. The simulation includes the geometry from the spiral casing inlet to the DT outlet. The simulation done by Trivedi had a GV angle of 9.84°, a rotational speed of 5.52 Hz, a flow rate of 0.199 m^3/s and a net head of 11.88m. This simulation will from now on be referred to as the TRS simulation.

IV. NUMERICAL MODEL

A. Transformation method

Several techniques are available to use for saving time and computational resources when performing transient simulations of turbine components. One is to use only some blades of the turbine and let the software copy the results around to emulate the whole rotating and stationary blades assembly. This is often not possible because of unequal pitch between the components. In some cases this is solved by altering the simulated geometry to achieve an integer pitch ratio, but this approach affects the validity of the analysis. In the case of the Francis-99 model geometry which has 28 GVs and 15 runner blades and splitters, the geometry could not be divided at all because of unequal pitch between the GVs and runner.

The transformation methods in ANSYS CFX aim to achieve periodicity despite using one or two blade rows with unequal pitch between rotating and stationary blades. Periodicity between runner and GV must be achieved to make it possible to use the transient rotor-stator interface. There are three transformation methods available in CFX.

The PT method makes it possible to use only one rotating blade and one stationary blade regardless of the pitch between them by scaling the flow profile across the interfaces while keeping the correct blade geometry and pitch ratio intact [12, ch. 6] [6]. The PT method uses instantaneous periodicity on the interfaces where unequal pitch is present and no special time scaling for the blade passings is used. The latter introduces a distortion in frequency disturbances crossing the rotor-stator interface and will therefore not be able to predict flow imposed by rotor-stator interactions [13, p. 2]. The overall performance and flow is usually well predicted [6, p. 3]. The time transformation and Fourier transformation methods are more advanced. Both methods are able to represent the correct blade passing frequencies between stationary and rotating domains. The time transformation method is similar to the PT method but prevents frequency distortions by scaling the *time* proportionally to the scaling of the flow profile [12, ch. 4.1]. The Fourier transformation method decomposes the pitch-wise periodics and the frequencies on the rotor stator interface, and stores them using Fourier series. The stored frequencies are then used to reconstruct the flow with the correct rotor stator frequencies [12, 4.2]. The present work is limited to the PT method because of time limitations.

The mesh connection used in the PT simulations is the general grid interface (GGI) which is able to connect domains despite non equal meshes at the interface. This mesh connection is in theory maintaining strict conservation, should not affect convergence and takes pitch change into account by scaling the flow up or down [12].

B. Mesh

The geometry was divided in to three domains for the PT simulations. The first domain is a stationary domain containing one GV. One runner blade and one splitter constitute the second domain which is rotating, while the third domain contains the whole DT.

The mesh used in the present simulations is based on the mesh provided in the first Francis-99 workshop held in december 2014. The original mesh characteristics for the DT and runner is shown in table I. The mesh for the GV is specific to the present work. The runner contains 15 identical runner blades and 15 identical splitters placed alternatingly. All 28 GVs are equally spaced and identical. The mesh for both domain 1 and 2 had to be cut to prepare the mesh for use with the transformation method. All modifications were done in the meshing software ANSYS ICEM. For the runner this was done by sweeping the runner blade suction side surface 6° counter clockwise and 18° clockwise from its original position. This approach created an enclosure around one runner blade and one splitter while also ensuring a totally similar surface geometry on each side. This was important because the geometry would be copied around and would therefor have to fit perfectly into each other without overlap or holes. This approach also ensured that 24° was cut out of the runner which is exactly 1/15 of the runner. The runner blade does not extend to the outlet of the runner so this section had to be cut by extending lines from the already swept area along the hub down to the runner outlet. This resulted in a slight simplification to the runner geometry as it extended the hub to the runner outlet. The GVs were cut by defining a line between the blades and sweeping this line $\sim 12.86^{\circ}$ (1/28 of the circle). The surfaces created by sweeping were then used to cut and associate the existing blocking to create a mesh. The cutouts can be seen in figure 4 together with the wireframe of the whole assembly of runner and GVs.

An attempt was made to keep the blocking as close to the original as possible but the cutting and association to the new enclosure was not a straight forward process and caused a reduction of quality of the mesh both for the runner and DT sections, especially at the outlets. The quality of the original and modified mesh is shown in table I. Quality (of element) and Min angle (of element) are two of many



Figure 3. Overview of the parts simulated



Figure 4. Wireframe and cutout of runner and guide vanes

ways to measure mesh quality in ICEM. A perfectly regular mesh element would have a quality value of 1 while the perfect value for the angle would be 90° for a hexahedral mesh. The modified mesh node numbers for the GVs and runner are for the complete geometry, not for the individual cut-outs. The max y+ value for the original mesh are taken from the TRS simulation done by Trivedi [1] while the max y+ values for the modified mesh are taken from PT simulation 2. The increase in max y+ from the original DT mesh to the modified DT mesh can be explained by high local velocities at the inlet that are not present in the TRS simulation as the first layer thickness is unchanged.

C. CFX configurations

Two simulations have been done with the PT method and the specifications are shown in table II. PT simulation 2 was performed to make sure the results were not affected by the initial conditions defined by a steady state simulation. PT simulation

Org. mesh	Nodes	Quality	Min angle	${\rm Max}\ y+$
GV		≥ 0.31	19	747
Runner	$5\ 172\ 300$	≥ 0.30	18	368
DT mesh	$3 \ 639 \ 241$	≥ 0.66	44	8
Mod. mesh	Nodes	Quality	Min angle	${\rm Max}\ y+$
GV	966 840	≥ 0.22	15	981
Runner	$5\ 709\ 060$	≥ 0.10	6	864
DT	$1\ 455\ 075$	≥ 0.66	45	91
		Table I		

10	
Mesh	QUALITY

1 was done to check if simulation beyond the first runner rotation is necessary for accurate results. Both simulations were done with a time step of 2.4846E-4 s corresponding to 0.5° runner rotation and results were recorded every 4^{th} time step. In PT simulation 1 one rotation of the runner was simulated. PT simulation 2 was performed for 7 rotations of the runner while the results were recorded for the last rotation.

Domain 3 was used to do a simulation of the DT without the runner and GVs (called DT-only). This simulation was configured the same way as PT simulation 1 for comparison purposes. The inlet boundary condition for this simulation was taken from the RSI simulation and was defined as a velocity field.

All transient simulations were done on a high performance computer at NTNU called Vilje. Details on the specifications of Vilje can be found in table II and on the NOTUR website [14].

1) Interface model and time step: The interfaces 1 and 2 in the transient simulations were both defined as transient rotor stator. This interface model updates the runner position and passes the information across the interfaces for every time step [7, p.2]. It is normally used for simulations with the full runner or a part of the runner with full periodicity intact and takes all transient effects between the rotating and stationary domains into account [12, sec. 5.3.3.1.4]. It is important to note that the circumferential normalised coordinates option had to be set to global at the interface between domains 2 and 3. This had to be done because the hub presented an area with no mesh at the centre of the runner outlet and CFX tried to fill this void by moving the runner towards the centre for the DT inlet. This in turn caused artificial flow in the centre of the DT cone and no flow at the DT walls. The global normalised coordinates option controls the interaction that happens in the interface by fixing both sides to their relative positions in physical coordinates, while still allowing rotation [12, sec. 13.1.4].

An adequate time step is important in transient CFD simulations. There is a general focus on getting the time step small enough to capture the transient phenomena occurring. In this case, the transient component of the simulation is the runner. The Francis-99 model runner has 15 full length blades, and 15 splitters which start at the same place as the blades but end at approximately half the length of the blades. This fact effectively gives different passing frequency at the inlet and outlet of the runner. In the present work the number of blades seen at the inlet of the runner (30) was used to define the passing period. The passing period is defined as the time it takes from one blade passing to the next. In this case the passing period was

$$\frac{1}{RPS \times \text{blades}} = \frac{1}{5.59 \times 30} = 5.963 \times 10^{-3} s \quad (1)$$

The default way of defining the time step when using transformation methods in CFX is to define the number of time steps per passing period. The time step has to be small enough to capture the transient variations of interest and small enough for the simulation to converge. Several time steps ranging from 5 to 200 per passing were tested and it was found that the results became unphysical (DT pressure recovery (PR) around or above 100%) for time steps above 50 per passing. The same problem was encountered by Cervantes et al. [15, p. 3] where the fluctuations in PR was attributed to rotating inlet boundary conditions changing the relative position to the nodes for every time step. The resulting interpolations for every time step caused fluctuations. This could also be the explanation for the PR fluctuations seen in the present work. Cervantes et al. [15] solved the problem by making sure that the relative position of the inlet boundary condition to the nodes newer changed when rotating. The time step of 24 per period ($\Delta t =$ 2.4846E-4s) was chosen to have a time step large

enough to make sure to stay out of the unphysical results while keeping it small enough to be able to capture the pressure pulsations.

The Courant number is commonly used as an indicator for numerical stability of a transient simulation:

Courant number
$$=\frac{v\Delta t}{\Delta x}$$
 (2)

The Courant number indicates how many mesh cells are passed by the fluid at one time step and it is desirable to have a value lower than 1; indicating that the fluid passes less than one cell in one time step [16, p. 34].

2) Boundary conditions and turbulence model: The inlet boundary condition for the PT simulations was defined as a simple homogeneous velocity profile with parameters presented in table II. This simple way of defining the inlet boundary condition was chosen despite the fact that more realistic velocity profiles were available from the simulations done by Chirag Trivedi [1]. This was done to have as realistic conditions as possible considering that the inlet velocity profile for the GVs is normally not available when doing simulations on a new geometry. The outlet boundary type was set opening with a pressure of 0 Pa and a velocity direction normal to boundary condition. The boundary type opening, allowing flow in both

Table II CFX transformation method run definition

Parameters	Description
Simulated components	Domain 1: Stationary, one guide vane
_	Domain 2: Rotating at 5.59Hz clockwise (seen from above), one runner blade and one splitter
	Domain 3: Stationary, draft tube
	All domains have 0 atm as reference pressure
Grid type	Multiblock, hexahedral
Simulation type	Transient blade row, PT, time step size: 24 per period ($\approx 2.49\text{E-4s}$), 0.5° of runner rotation
	PT simulation 1: Initial time 0 s, total time: one rotation (≈ 0.179 s)
	PT simulation 2: Initial time 0 s, total time: 7 rotations (≈ 1.25 s)
Interfaces	For duplication of blade rows: Rotational periodicity, Mesh connection: GGI
	Interface 1 and 2: General connection, transient rotor stator with automatic pitch change,
	mesh connection: GGI
	Special for Interface 2: Circumferential normalised coordinates option set to global
Boundary conditions	Inlet: Mass flow rate: 7.25 kg/s for each GV, 203 kg/s total, turbulence intensity 5%
	Direction in cylindrical components: axial 0, radial $-1/3$, theta -1
	Outlet: Opening, pressure (0 Pa) and direction (normal to boundary condition)
	Walls: Smooth walls with no slip condition
Fluid	Water at 20° C
Solver control	Advection Scheme: High resolution
	Transient scheme: Second order backward Euler
	Turbulence numerics: High resolution
Convergence control	Maximum coefficient loops: 10, residual target (RMS): 1E-5
Turbulence model	SST
Run type	10 nodes with 8 cores each (Xeon E5-2670@2.60GHz), 32 GiB per node, double precision

directions, was chosen for numerical stability of the simulation.

The turbulence model was for all simulations set to the shear stress transport (SST) model. Work done on the Turbine-99 [17] for example by Simen Røst Breivik [3] and Björn Jedvik [5] has shown contradicting results on which turbulence model is best to use in a DT simulation. In the case of Francis-99 Chirag Trivedi [1] tested both the $k - \epsilon$ and the SST turbulence model and concluded that both worked well in TRS simulations done on the Francis-99 model. Simulations with the transformation methods by Connel et al. [6] in collaboration with ANSYS were solely done with the SST model. The use of the SST turbulence model was based on this, and the author's previous experience with SST. Further discussion on the SST turbulence model can be found in the work done by Moritz [4, p. 13] and Geberkiden [16, p. 41].

3) Initial conditions: The PT and DT-only simulations done in the present work use a steady state solution as initial condition. For the DT-only simulation, a steady state version of the simulation setup was performed. The steady state initial condition for the PT simulations used the same geometry as in the transient PT simulation but the interfaces 1 and 2 had to be defined as frozen rotor. This interface model fixes the runner in one position and passes the true flow up and down stream as if it was rotating but only for this position of the runner. There is no averaging of the flow for each position [12, sec. 5.3.3.1.2]. This gives a good initial condition but does not take any transient effects into account. The steady state simulation for the PT simulation was stopped after 500 iterations while the root mean square (RMS) residuals were oscillating at 10^{-4} - 10^{-5} most likely because of transient effects in the DT. The steady state simulation for the DT-only simulation converged (maximum residuals lower than 10E-6).

D. Evaluation methods

1) Pressure: The pressure from simulations was captured at DT21 and DT11 as defined in section III. The results were recorded every 4^{th} time step for one runner rotation. A fast fourier transform analysis was conducted to find the frequencies. All pressure results shown in the present work are made to fluctuate with a mean value of 0 for easier comparisons by subtracting the mean pressure value:

$$P(t) - \bar{P} \tag{3}$$

2) Velocity: The velocity profiles were captured by defining section 1 and 2 in the simulations at the same place and with the same positive directions as described in section III. 100 points were defined along both lines and velocities in the tangential and axial direction were time averaged using measurements from 18 time steps during one rotation of the runner both for PT simulation 1 and 2.

3) Pressure recovery: The DT is converting kinetic energy into pressure and thereby effectively lowering the pressure at the inlet [4]. Pressure recovery of the DT is a common way to estimate the DT performance. The pressure recovery shows the fraction of the kinetic energy that has been converted to pressure through the DT and is defined as

$$PR = \frac{P_2 - P_1}{0.5\rho v_{all}^2}$$
(4)

The PR presented in the present work are time averaged values for all time steps during one rotation of the runner except for the RSI simulation by Trivedi where only the value from the last time step was available.

4) Computational time: It is difficult to compare the computational resources of two simulations done on different computers with a different number of central processing unit (CPU) cores. Difference in clock speed of the CPU, random access memory (RAM), and in overlap between the computation domains because of different number of cores are some of the aspects making a comparison difficult. The most convenient way of comparing the computational resources used by two simulations is to compare the CPU time for one specific sequence from both simulations. The CPU time is defined as the time one CPU core is used for a computation task. CFX shows the elapsed total CPU time which is related to CPU time as shown in equation 5.

$$CPU time = \frac{Total CPU time}{number of CPU cores}$$
(5)

V. Results

A. Convergence

1) Mesh independence: A mesh independence test was conducted for the DT mesh. All tests were done with simulations at BEP with the inlet velocity boundary condition from the RSI simulation. The different meshes that were tested are shown in table III. Mesh 1 is the original mesh supplied for the



Figure 5. (a) Axial velocity profiles from sector 1

Francis-99 workshop. Mesh 2 and 3 were globally scaled down and have a mesh growth ratio of 1.5 from the wall. Mesh 4 has the same growth ratio but the first layer thickness (f.l.t) was increased to 1.0. Mesh 5 and 6 have a growth ratio of 1.15 and a f.l.t of 0.1 and 0.01 respectively to better capture the boundary layer at the wall.

All meshes allow a converged simulation (residual target max 1E-6) except for mesh 5 where the residuals started oscillating at values around 10E-5 and mesh 6 which had oscillating residuals above 1E-4. The fact that convergence gets worse when y+ decreases in a DT is a phenomenon observed and discussed by Moritz [4, sec. 4.1].

Mesh 3 was chosen for the simulations done in the present work since there were no real differences between the results from the simulations done with the meshes except lower convergence for higher resolution close to the wall (mesh 5 and 6). Furthermore, mesh 3 was the least computational expensive.

2) Transformation method convergence: The PT simulations inner loop converged after 4 - 5 itera-

Mesh	nr. of nodes	f.l.t [mm]	avg. $y+$	\mathbf{PR}
DT Mesh 1	$3\ 639\ 241$	0.32	25.6	81.638%
DT Mesh 2	$2\ 488\ 200$	0.32	25.6	81.567%
DT Mesh 3	$1 \ 455 \ 075$	0.32	25.6	81.577%
DT Mesh 4	$1 \ 455 \ 075$	1.0	80.4	81.887%
DT Mesh 5	$1 \ 455 \ 075$	0.1	8.1	81.217%
DT Mesh 6	$2 \ 579 \ 115$	0.01	1.6	- %
Table III				

DT MESH SEPCIFICATIONS FOR MESH INDEPENDENCY TEST



(b) Tangentilal velocity profiles from sector 1

tions. The RMS Courant number was 0.66 and the maximum Courant number was 22.38. The maximum Courant number is higher than desirable and found to be caused by the compromised quality of mesh at the outlet of the GV and the runner. A smaller time step could have lowered the Courant number but was not used because of problems with unphysical results as discussed in section IV-C.

The mass flow rate conservation in the interfaces 1 and 2 deviated with a maximum 0.12% with a mass flow rate in to the GV of 203 kg/s and a mass flow rate of 202.75 out of the DT. This is a large deviation compared to a deviation less than 10E-5% registered in the TRS simulations done on the whole assembly by Trivedi [1, Table 5]. The largest deviation was registered at interface 2. The GGI should provide full conservation across domaines but in combination with the use of only 1/15 of the runner it could be that the GGI was not so conservative as it should. The up and down scaling of the flow to take pitch change into account does not seem to be the cause as the flow varies very little (max 0.008 kg/s) for one runner rotation.

The comparison of the time averaged velocity profiles recorded during the first (PT simulation 1) and seventh (PT simulation 2) rotation of the runner can be seen in figures 5. The solution from PT simulation 1 is not fully converged, specially for the tangential velocity profile. This is a normal phenomenon but it could be discussed if this deviation is big enough to justify the computational resources needed for a simulation with several additional rotations of the runner.

The pressure recovery was measured and averaged over one whole rotation of the runner for simulation 1 and 2. The average PR for PT simulation 1 was 87.67% and 87.01% for PT simulation 2.

The presented results show the necessity of letting the transient simulation settle to get values that are not influenced by the initial conditions. It was not tested how many runner rotations were really necessary to achieve a fully converged results but it was assumed that seven rotations were sufficient based on how little the velocity deviates over one runner rotation in PT simulation 2 compared to PT simulation 1. The following presents only the results from PT simulation 2.

B. Transformation method speed

One of the main advantages of using transformation methods is to simulate only a small portion of a repeating geometry and thus reducing the simulation time. The TRS simulations by Trivedi [1] were done on 84 CPU cores while the PT simulations of the present work were done on 80 CPU cores. Both simulations use 720 time steps to cover one runner rotation and both simulations converged after 4-5 coefficient loops in every time step. The CPU time for one runner rotation for the TRS simulation was 67.5 hours while the CPU time for one runner rotation for the PT simulation was 2.7 hours. This means, when assuming equal computer setup, that the TRS simulation used approximately 25 times the computational recourses for one runner rotation compared to the PT simulation. The speed advantage of the PT simulation compared to the TRS simulation is significant but no surprise as the mesh simulated in the PT simulation is much smaller.

C. Transformation method accuracy

Comparisons of the pressure recovery, pressure measurements and velocity profiles will be presented to find the accuracy of the PT simulation results.

1) Pressure recovery in the draft tube: The pressure recovery for the PT, DT-only and TRS simulation was extracted as described in section IV-D3. The PR for the full simulation was 84.1%. The slight difference in mass flow rate in the TRS simulation compared to the PT simulation is not believed to have a significant impact on the PR. The PR of the PT simulation seems to over estimate the DT performance with a PR value of 87.0%. In theory the overall performance should be predicted well with the PT method but 3% points above the value from the full simulation does



Figure 7. FFT analysis of frequencies measured in PT simulation 2

not confirm this. The DT-only simulation gave a PR of 81.6% which under predicts the PR almost as much as the PT simulation over predicts it. This could be explained by the influence of a static inlet condition where all interaction between DT and upstream is not simulated. The PR from the DT-only simulation is in fact very close to the steady state simulations done with the same inlet conditions (81.6%) which shows that the DT at BEP without a rotating runner has few transient effects that influence the PR.

		DT	\mathbf{PT}	RSI	
	PR	81.6%	84.1%	87.0%	
Table IV					
PR E	XTRAG	TED FRO	OM THREE	E SIMULAT	IONS

2) Pressure measurements at DT11 and DT21: Only the results from DT21 will be shown as the pressure data from DT11 and DT21 were near identical except for the time shift of the pulsations because of their position.

A fast Fourier transform (FFT) analysis of the pressure recorded every 4^{th} seconds for one rotation of the runner shows frequencies of 86, 173, 259, 346 and 432 Hz, from now on called freq. 1 - 5 respectively. These measurements were done downstream of the runner and it is therefor expected to see at least frequencies relating to the blade passings (15×5.59 Hz= 83.9Hz) and the blade + splitter passings (30×5.59 Hz= 167.7Hz). These frequencies were observed but it is not clear if freq. 2 is the second harmonic frequency of freq. 1 or if it is actually related to the splitters. Freq. 3 - 5 were found to be the third, fourth and

fifth harmonic frequency of freq. 1 (refer to appendix C). It must be recognised that freq. 4 and 5 are relatively strong. The reason for this could be the same as encountered by Cervantes et al. [15] as discussed in section IV-C1. Figure 7 shows the result from the FFT analysis with the x-axis showing frequency values normalized with the runner rotation frequency (5.59 Hz). This makes it possible to see that freq. 1 and 2 correspond to the number of blades and splitters, respectively.

The frequencies observed by Trivedi in experimental measurements [10] were 15.95, 39.48, 100, 165.74 and 300 Hz where the first two are typical pressure fluctuations in the model setup at NTNU between the upstream tank and the runner and the DT tank and the runner. The 100 and 300 Hz frequencies are assumed to come from a frequency transformer used in the experiments. This leaves only the blade passing frequency of 165.74 Hz which was also observed in the present simulation.

In general it must be noted that the amplitude of the pressure fluctuations at DT21 were lower in the simulations than in the already low amplitudes in the measurements done by Trivedi. This can be seen in figure 6 which shows pressure values for a half runner rotation from PT simulation 2 and experiments done by Trivedi [1]. The low frequencies for the experimental values shown in the figure should be ignored (reasons mentioned above). The values from experiment and the transform simulation have some resemblance when looking at the high frequencies. It can also be seen in figure 6 that the values from simulation have a slightly higher frequency (as found by FFT).

Detailed flow phenomena like pressure pulsations can be affected by the distortion in frequency disturbances across the rotor-stator interfaces induced by the PT method. This could be one of the reasons for the inaccurate frequencies and amplitude of the pressure. One other cause for the inaccurate frequencies can be the relatively few values that were available for the FFT analysis.

3) Velocity profiles at section 1 and 2: The axial and tangential velocity profiles for both section 1 and 2 from PT simulation 2, DT-only, TRS simulation and LDV measurements are compared in figures 8 and 9. All values are normalised with the bulk speed at the DT inlet.

The axial PT simulation 2 profiles have a lower velocity close to the DT centre and the velocity is higher close to the DT wall both for sector 1 and 2 compared to the LDV profiles. The lower velocities close to the DT centre could be explained by the simplifications done to the hub area of the runner. The extension of the runner hub prevents the water from flowing towards the centre of the DT and thereby causes the water to flow slower in this area. The higher velocity at the DT wall could mean that the roughness of the wall is defined too smooth in the simulations or it could mean that the LDV measurements are a bit offset towards the DT centre.

The tangential PT simulation 2 profiles differ from the LDV profiles but seem to follow the same basic form, especially considering that the measured LDV points close to the DT centre are relatively far apart. This fact could prevent the LDV measurements from showing a velocity peak at or around the same radius as the simulation profiles. The axial velocity profiles from PT simulation 2 have the largest deviation from the LDV measurements close to the DT centre. This could be caused by the simplification done at runner hub which adds some surface area in the middle of the runner outlet. This surface could impose more rotation of the water close to the CT.

The profiles from the TRS simulation are almost



Figure 6. Pressure measurements from DT21 for 180° runner rotation



Figure 8. (a) Axial velocity profiles from sector 1



Figure 9(a) Tangential velocity profiles from sector 1

without exception closer to the LDV profiles in both sector 1 and 2. The main reason for the deviation of PT simulation 2 from the LDV and TRS simulation in all cases is most likely the simplification done in the runner hub area. Other sources for the deviation can be the relatively rough estimate for the inlet velocity to the GVs as this can affect the flow more than initially thought. The reduced quality of the mesh at the outlet of the GVs and runner could also affect the flow in to the DT. The DT-only simulation gave slightly different profiles compared to the TRS simulation which confirms that the interaction between runner and DT affects the flow in the DT. The deviation between the DT simulation and the TRS simulation is



(b) Tangential velocity profiles from sector 2

expected to be larger for simulations at other turbine operating points where the transient phenomena are more prominent.

D. General discussion

The results show a significant reduction in computational resources needed for the PT simulation compared to the TRS simulation. The results seem to be less accurate than even a steady state simulation of the DT defined with inlet boundary conditions from the TRS simulation. The transformation method has worked well for gas turbines in other studies (see section II) suggesting that there are other reasons for the inaccurate results. The GV and runner mesh have a reduced quality and were not tested for mesh independency because of time limitations which could be a source of error. The simple definition of the inlet boundary condition at the GV inlet could be a reason for the inaccurate results. The relatively large deviation in the mass flow rate in interface 2 could probably also have been significantly reduced by defining a strict conservation criteria at the interface in CFX, which could enhance the accuracy of the results. It must also be kept in mind that the PT method is the simplest of the three transformation methods and it is very likely that the fourier transformation method could give better results for the present model.

One important aspect that needs to be discussed when considering the use of a transformation method is the availability of a mesh. The transformation method is convenient if the mesh has to be constructed from scratch anyway as only a small portion of the geometry has to be meshed. The choice is not so obvious if a mesh of the whole geometry is available in the first place. The cutting and modifications necessary to make the mesh work with the transformation method takes a considerable amount of time, and after that even more time will be spent optimising the mesh quality because it has been affected by the modifications.

Furthermore, these kind of methods that reduce the needed amount of computational resources could become less relevant as the available computer power steadily increases over time. The present work shows the potential reduction in simulation time and that a TRS simulation still takes a very long time to compute. It is therefore believed that these kind of methods will be useful in the foreseeable future, especially for computationally expensive simulations like fluid-structure interaction simulations.

The transformation method promises a great reduction in simulation time (which was confirmed) and accurate results (which was not confirmed). It is believed that transformation methods true potential was not shown in the present work, and that further work eliminating the mentioned sources of error could reveal how well this method really works.

The comparison of PT simulation 1 and 2 showed that a simulation of one runner rotation was not enough to give a converged solution. It could on the other hand be discussed if seven simulated runner rotations is excessive and that maybe even just two runner rotations would be enough.

VI. CONCLUSIONS

The goal of this thesis has been to find out if the PT method can save computational resources compared to a TRS simulation while still being able to give accurate results. An additional goal has been to check if additional computational time can be saved by only simulating one runner rotation instead of letting the simulation settle over several rotations.

The computational resources used by the PT simulation were significantly reduced because of the reduced mesh compared to the TRS simulation. This confirms one of the main advantages of the transformation methods compared to a TRS simulation. The accuracy of the PT simulation on the other hand was not as expected, as results did not match experiments or the TRS simulation. It was concluded that this deficit in accuracy most likely is caused by simplifications done at the hub of the runner, rough inlet boundary conditions, quality of the runner and the GV mesh and the use of the PT method rather than the fourier transformation method. It was additionally found that it is necessary to simulate more than just one runner rotation as the results deviated significantly from the simulation with seven runner rotations.

It is recommended that the research on this topic is continued as the present work is not conclusive about the usefulness of transformation methods in conjunction with hydro power turbines as a tool for achieving realistic DT simulations.

VII. ACKNOWLEDGEMENT

This research was supported in part with computational resources at NTNU provided by NOTUR.

The present work has been written as the master thesis required for the Master of Science degree in Mechanical Engineering at NTNU. I would like to thank Michel Cervantes and Ole Gunnar Dahlhaug for their supervision and inspiration throughout the work. I also want to thank PhD candidate Bjørn Winther Solemslie for always being helpful. A special thanks goes to all employees and my fellow students at the hydro power lab at NTNU for making my last year as a student a really memorable experience.

References

- C. Trivedi, M. J. Cervantes, B. K. Gandhi, and O. G. Dahlhaug, "Experimental and numerical studies for a high head francis turbine at several operating points," *Journal* of Fluids Engineering, vol. 135, 11 2013.
- [2] B. R. Gebart, L. H. Gustavsson, and R. I. Karlsson, "Proceedings of turbine-99. workshop on draft tube flow," Luleå University of Technology, Tech. Rep., 1999.

- [3] S. R. Breivik, "Project thesis cfd analysis of flow in a kaplan draft tube," 2011.
- [4] R. Moritz, "Project thesis analysis of a kaplan turbine draft tube," NTNU, Tech. Rep., 2013.
- [5] B. Jedvik, "Evaluation of cfd model for kaplan draft tube," Master's thesis, Luleå University of Technology, 2012.
- [6] S. Connell, M. Braaten, L. Zori, R. Steed, B. Hutchinson, and G. Cox, "A comparison of advanced numerical techniques to model transient flow in turbomachinery blade rows," 2011.
- [7] R. Blumenthal, B. Hutchinson, and L. Zori, "Investigation of transient cfd methods applied to a transonic compressor stage," *Proceedings of ASME Turbo Expo*, 2011.
- [8] Statkraft, "Statkraft information about tokke power plant, site loaded 7/05," 2014. [Online]. Available: http://www. statkraft.no/Energikilder/vaare-kraftverk/norge/Tokke/
- [9] Statkraft-Energi-AS, "Tokke-vinje reguleringen status 2005, site loaded 08/06," 2014. [Online]. Available: http://www.tokke.kommune.no/~/media/Tokke/ Dokument/Prosjekt/vilkarsrevisjonen/Statusrapport% 20Tokke%20Vinje%20des%202005_.ashx
- [10] "Francis-99 test case, site loaded 7/05," 2014.
 [Online]. Available: http://www.ltu.se/research/subjects/ Stromningslara/Konferenser/Francis-99/Test-Case-1.
 111520
- [11] L. J. Sundstrom, K. Amiri, C. Bergan, M. J. Cervantes, and O. G. Dahlhaug, "Lda measurements in the francis-99 draft tube cone," *IAHR Symposium on Hydraulic Machin*ery and Systems, 2014.
- [12] ANSYS, "Ansys help system," 2013.
- [13] C. Cornelius, T. Biesinger, P. Galpin, and A. Braune, "Investigation of transient cfd methods applied to a transonic compressor stage," *Jorunal of Turbomachinery*, vol. 136, 6 2014.
- [14] NOTUR, "Information about vilje hpc, site loaded 05/06," 2014. [Online]. Available: https://www.notur.no/ hardware/vilje
- [15] M. J. Cervantes, U. Andersson, and H. M. Lövgren, "Turbine-99 unsteady simulations - validation," 25th IAHR Symposium on Hydraulic Machinery and Systems, 2010.
- [16] B. M. Geberkiden, "An experimental and numerical investigation of a kaplan turbine model," Ph.D. dissertation, Luleå University of Technology, 2012.
- [17] M. Cervantes, T. Engström, and L. Gustavsson, "Proceedings of the third iahr/ercoftac workshop on draft tube flows. turbine-99 workshop 3," Luleå University of Technology, Tech. Rep., 2005.

Appendix

The appendix files can be found attached as a .zip file to the present work.

A. Convergence analysis

Excel document showing showing PR, analysis of velocity profiles and the data gathered for finding the right time step.

B. Pressure data

Excel document showing all pressure data gathered for DT21 and DT11 + the pressure data from experiments.

C. Pressure frequency analysis

Excel document showing harmonic frequencies for the pressure measurements at DT21.

D. Script for Vilje

PBS script showing a typical definition of a CFX run on the Vilje high performance computer.

E. Export-velocity-every-time-step

CFX post session file for exporting all time steps of the 100 velocity measurements along sector 1 and 2.