Performance prediction of the high head Francis-99 turbine for steady operation points

E. Casartelli, L. Mangani, O. Ryan, A. Del Rio
Lucerne University of Applied Science & Arts, Department of Mechanical Engineering, Horw, Switzerland
E-mail: ernesto.casartelli@hslu.ch

Abstract. Steady-state numerical investigations are still the reference computational method for the prediction of the global machine performance during the design phase. Accordingly, steady state CFD simulations of the complete high head Francis-99 turbine, from spiral casing to draft tube have been performed at three operating conditions, namely at part load (PL), best efficiency point (BEP), and high load (HL). In addition, simulations with a moving runner for the three operating points are conducted and compared to the steady state results. The prediction accuracy of the numerical results is assessed comparing global and local data to the available experimental results. A full 360°-model is applied for the unsteady simulations and for the steady state simulations a reduced domain was used for the periodic components, with respectively only one guide vane and one runner passage. The steady state rotor-stator interactions were modeled with a mixing-plane. All CFD simulations were performed at model scale with an in-house 3D, unstructured, object-oriented finite volume code designed to solve incompressible RANS-Equations. Steady and unsteady solver simulations are both able to predict similar values for torque and head in design and off-design. Flow features in off-design operation such as a vortex rope in PL operation can be predicted by both simulation types, though all simulations tend to overestimate head and torque. Differences among steady and unsteady simulations can mainly be attributed to the averaging process used in the mixing plane interface in steady state simulations. Measured efficiency agrees best with the unsteady simulations for BEP and PL operation, though the steady state simulations also provide a cost-effective alternative with comparable accuracy.

1. Introduction
Accurate and trustworthy numerical predictions of turbine performance in hydropower generation are more important than ever in the current competitive energy production environment. New developments in CFD codes provide quick, accurate and improved flow field predictions for hydropower applications. The Francis 99 workshop presents an opportunity for academic and industrial researchers to engage in research of the Tokke high head turbine model. After the first successful workshop in 2014, this second workshop allows to undertake more in-depth research and with an extended range of tasks for the participants of this event [10].

In this study, steady state and unsteady simulations are performed on the turbine in PL and high-load operating points as well as at design conditions. The computational domain reaches from the spiral casing inlet to the draft tube outlet, where a full 360°-model is used for the unsteady simulations and a passage model for the steady state computations. An in-house developed code is used here which is a block coupled incompressible solver embedded
in the OpenFOAM framework [7]. Simulation results are compared to measurements which are provided, together with the computational grids. For the steady state simulations using the passage model, a fully implicit mixing plane is applied for the rotor-stator interfaces [4]. The main focus of this study is to evaluate the performance of the steady state coupled solver used here in combination with the mixing plane, which promises to be an efficient method for hydro turbine investigations and can achieve considerable speed-up compared to conventional segregated solvers [1]. The unsteady simulations serve as numerical reference cases which require considerably more computational effort.

2. CFD-solver technology
2.1. Governing equation
The simulations are based on the Unsteady Reynolds Averaged Navier-Stokes (URANS) equations, which govern the dynamics of incompressible, viscous flows. The differential form can be written in an Arbitrary Lagrangian Eulerian (ALE) framework as follows:

\[ \nabla \cdot u = 0 \]  
\[ \frac{\partial u}{\partial t} + \nabla \cdot (u_r u) = -\frac{1}{\rho} \nabla p + \nabla \cdot (\nu_{eff} \nabla u) \]  

where \( u \) is the absolute velocity, \( p \) the pressure and \( \rho \) the density of the fluid. The convecting flux is written in terms of the relative velocity \( u_r = u - u_{ALE} \), where \( u_{ALE} \) is the velocity associated to the dynamic motion (either rigid and/or deformable) of the physical domain \( V = V(t) \) and it is evaluated enforcing the so-called Geometric Conservation Law (GCL). For instance, for a purely rigid rotation with constant angular velocity \( \Omega \), it corresponds to \( u_{ALE} = \Omega \times r \). Finally the effective kinematic viscosity is the sum of the laminar and the turbulent contributions \( (\nu_{eff} = \mu + \mu_t) \).

2.2. Numerical discretization
All simulations were performed using a so called coupled solver. The difference to the conventional segregated approach lies in the way the governing equations are solved. Speed and robustness are among the most important requirements for any software that has to be used for the time-accurate numerical analysis of complex highly unsteady phenomena. In the coupled solver the governing equations are coupled implicitly, since resolving efficiently the pressure-velocity coupling is essential for the performance of any CFD code. In comparison to the segregated solver of the SIMPLE family of algorithms, which sequentially solve each governing equation, the coupled approach benefits of higher robustness and speed. The coupled solver used for the simulations bases on the open-source CFD library OpenFOAM and was developed by Mangani et al. [7].

2.3. Turbulence modeling
All the simulations were performed using a SST \( k-\omega \) turbulence model with automatic wall treatment. The automatic wall treatment decides whether the boundary layer is being resolved or a wall-function is used. This type of turbulence model requires two auxiliary scalar equations (not coupled) in addition to the momentum and continuity equation, to compute the conservation variables of turbulent kinetic energy \( k \) and specific dissipation rate \( \omega \).

Although two equation turbulence models are known to have shortcomings at off-design conditions, it has been shown in the first workshop of Francis 99 [9], that such a model is capable to produce suitable results compared with the measurements, even for operation apart from the best efficiency point [1]. The choice of the turbulence model is based also on the fact that it is widely accepted in industrial applications [12] and corresponds to the philosophy of applied science.
2.4. Domain discretization

For the steady state and the unsteady simulation of static operating points, two different meshes were applied. For the unsteady simulations a complete 360°-model is necessary to capture the unsteady flow-behavior. The simulation domain ranges from the spiral case inlet, the stay vanes, the guide vanes and runner to the draft tube as shown in fig.1 (a). Rotating and stationary domains are matched using a rotor stator arbitrary mesh interface (AMI).

For the steady state simulations, a single passage for the guide vane and the runner is applied, see fig.1 (b). In this case the flow-parameters are interpolated between the stay vanes and the guide vane passage, between the guide vane passage and the runner passage and also between the runner passage and the draft tube. The interpolation is performed using the mixing plane method. For the simulation a fully implicit mixing plane was used [4]. Therefore, flow-parameters are averaged in the circumferential direction. This method bases on the observation, that flow information in circumferential direction mixes out faster than in Z-direction (for the interfaces between stay vanes and runner) or r-direction (for the interface between runner and draft tube).

The computational grids for both steady state and unsteady models are provided by the workshop committee. A mesh study is not performed here. Three operating points at different guide vane angles are analyzed. The guide vane opening angles are 9.84, 6.72 and 12.43 degrees at BEP, PL and HL, respectively. For further geometrical details of the model see [10]. The mesh size for the complete model is approx. 23 million elements. Except for the spiral case/stay vane domain, all parts have a structured mesh containing hexahedra elements. Information concerning mesh size and quality criteria can be found in table 1. All values are related to the model at BEP.

![Figure 1: Domain discretization](image)

(a) 360°-model.

(b) Reduced model.

Table 1: Mesh parameters for the complete 360°-model for the BEP.

| Mesh               | Element size | $Y^{+}_{AV}$ (BEP) | min. Angle | Volume change | Aspect ratio |
|-------------------|--------------|---------------------|------------|---------------|--------------|
| Global            | 22984188     | 42.2                | 2.2°       | 1.90          | 1.82         |
| Spiral & stay vanes | 3427708      | 30.8                | 2.2°       | 1.90          | 1.82         |
| Guide vanes       | 78841800     | 42.2                | 25°        | 1.20          | 0.25         |
| Runner            | 10780800     | 36.2                | 36.9°      | 1.23          | 0.19         |
| Draft tube        | 890880       | 35.5                | 36.7°      | 1.18          | 0.31         |
Table 2: Element types in the spiral case/stay vane domain*

| Tetrahedra | Pyramids | Wedges | Hexahedra |
|------------|----------|--------|-----------|
| 118,354    | 154,995  | 920,880| 2,233,479 |

The definition of the quality parameters is according to [2]. The reduced model for the stationary-case contains a quarter of the complete 360°-model elements. It has approx. 5 million elements. The rotor side spaces are neglected in both models so that disc friction as well as leakage flow are not directly available in the simulations. To compare the simulation with the measurements, the given experimental data for the friction torque from the test case are subtracted from the simulation output.

2.5. Numerical schemes
All the convective terms are discretized with a high resolution scheme. Therefore the flow solver itself switches between first and second order depending on the local flow-parameters [8]. The turbulence schemes are of first order and the temporal term is of second order (implicit Euler backward). To reach sufficient convergence 10 non-linear inner iterations are applied for the unsteady simulation, resulting in rms values for the flow solver residuals of at least $10^{-5}$. Initially the time step was chosen in a way, so that the temporal resolution of simulation was equal to $3^\circ$ of a rotor revolution. The value was then further reduced until the highest frequencies of the pressure signal in the simulation could be captured (approx. 10 samples per period were required). A sufficient sampling of the pressure signal had been achieved with a time step 0.75ms, which corresponds to a resolution of $1.5^\circ$ per each time step.

2.6. Initial and boundary conditions
For the steady state simulation no solution from a previous CFD simulation is used for the initialization. Therefore the simulation started from zero using constant values for $k, \omega, U$ and $p$ in the initialization. All unsteady simulations are initialized with a steady state Multiple Reference Frames (MRF) solution.

At the inlet of the computational domain the mass flow rate according to the operating point, a turbulent kinetic energy based on the turbulent intensity $I$ of 5%, which is a common value for hydraulic turbines [6] and a turbulent frequency based on $(\mu_t/\mu)$ around 10 for confined flow conditions [3] are imposed.

At the outlet of the computational domain, an average static pressure of $p_{\text{outlet}} = 0 \, \text{Pa}$ is applied, due to the incompressible flow type. The walls in the domains are non-rotating, except for the runner domain. At the walls, a no-slip condition is imposed. The runner domain rotates with constant speed around the z-axis.

3. Results
3.1. Integral values
3.1.1. Comparison between measurement and simulation
In order to evaluate the accuracy of the simulation, the two simulation-models with constant boundary condition are analyzed and compared with the steady measurements. In figures 2 (a)-(c) the values for the efficiency, the torque on the runner and the net head are shown. The measured torque represents the torque on the shaft. In consideration of the measurement set-up the following definition for the net head was used:

$$H = \left( \left( \frac{1}{A_{\text{in}}} \right)^2 - \left( \frac{1}{A_{\text{out}}} \right)^2 \right) \cdot \frac{Q^2}{2g} + \left( p_{AV,\text{in}} - p_{AV,\text{out}} \right) \cdot \frac{1}{\rho g} \quad (3)$$
\( H \) stands for the net head and \( A_{in} \) as well as \( A_{out} \) represent the cross sectional areas on the inlet respectively outlet of the turbine. Further \( p_{AV, in} \) and \( p_{AV, out} \) are the area averages of the static pressure on the inlet, respectively outlet, \( Q \) represents the volume flow rate across the turbine and \( g \) stands for the gravitational constant. It can be seen that the values for the

![Figure 2: Comparison of the global variables between measurement and simulation.](image)

head in all simulations have been overestimated compared to the measurements. A possible explanation for this behavior is, that at the inlet of the turbine, a volume flow rate with a constant velocity distribution is imposed. The inlet pipe is relatively short (see fig.1) and therefore the flow field does not have sufficient time to develop a fully turbulent velocity profile before entering the bladed region of the machine. Therefore, a steady state simulation of the BEP is conducted using a simulated fully developed velocity profile. The result in figure 2 (c) shows that a difference of 1% can be observed in the efficiency between profile boundary condition and constant value boundary condition, which brings simulation results and measurements closer together. Therefore it is presumed, that a difference can also be expected for the unsteady simulations. The difference in efficiency results from 1% less torque and 0.16% less net head using the fully developed velocity profile. However, the relative difference of 0.16% in head does not justify the gap between measurement and simulation. Further reasons which are not examined here such as mesh sensitivity and turbulence model influence would have to be analyzed in for closure.

For the torque it can be seen, that the stationary simulation with the mixing plane overestimates the value by a significant amount. The unsteady simulation is closer to the
measurements except for the high-load operating point. The resulting values for the efficiency of the unsteady simulation correspond well with the measurement. The error made for the BEP is just 0.61%. Even for PL the error is relatively small with 0.90%. Under high load conditions the simulation model has more difficulty to capture the flow behavior. For this operating point is the error for the efficiency 2.74%. The steady state simulations consequently shows larger differences to the measurement. The errors are 3.16%, 4.12% and 2.31% for BEP, PL and HL, respectively.

3.1.2. Comparison between the steady state and unsteady simulations

As outlined above, there is a significant discrepancy between the computed torque from the steady state simulation and the unsteady simulation for BEP and PL conditions. The reason can be traced back to the simplification made for the steady state model. An unsteady operation point is simulated as if it behaves steady. With the mixing plane approach mentioned above, the flow parameters are averaged along the circumference at the interfaces. For homogeneous flows, like for example for a turbine at the BEP, the influence is usually small [4]. Nevertheless in the simulation it can be stated, that even for the BEP, the flow field contains areas of flow separations at the stay vane trailing edge, which causes the differences in the torque. See fig.3. The separation zone influences the flow field and leads to a marginally different incidence angle at the runner. The relative difference in flow angle of the absolute velocities at the interface between runner and guide vanes is 2.6% between the two models at BEP. The resulting pressure profile on the runner blades consequently leads to a different value for the torque relative to the undisturbed flow field. The averaging process of the mixing plane reduces the intensity of this effect.

Figure 3: Detected separation zone for the unsteady simulation at BEP

(a) Development of the absolute total pressure on the mid-height (z=0m).
(b) vector-and conturplot of the absolute velocity at the stay vane trailing edge.

3.2. Pressure probes

Time-dependent pressure measurements are provided for this study, from two sensors placed in the draft tube short after the trailing edge of the runner and a third one in the vaneless space (fig. 4 (a)). To compare the measurements with the simulations, only the unsteady simulations are used. In order to visualize the dominant frequencies, Fast Fourier Transformations (FFT) have been performed.
In the vaneless space the dominant frequencies mainly correspond to the blade passing frequencies for all operating points. Therefore there are peaks at 15 and 30 times the runner frequency, corresponding to the 15 splitter blades and 15 runner blades. In fig.5 (a) and (b) data for PL is reported. It can be seen, that the simulation results contain the same dominant frequencies like the measurement, but not vice versa. Also, the time-signals show similar behavior. It can also be noted that the measured signals have a larger magnitude and the FFT show a much broader spectrum than the simulation. Part of the broader spectrum is caused by the system itself but also by the measurement noise. Because the simulation model does not consider the measurement set up of the turbine with all its armatures (compare figure 1 with figure 4 (b)), the corresponding frequencies are not represented in the FFT spectrum. Due to this reason and the strong band noise in the signal, in the following only the simulated pressure signals are analyzed.

(a) Location of the pressure sensors in the draft tube and vaneless space. 

(b) Model Francis turbine test rig [11].

Figure 4: Measurement set-up.

(a) Pressure signal in the draft tube (DT5).  
(b) Pressure signal in the vaneless space (VL2).

Figure 5: Comparison of the pressure signal (time signal and FFT) between the measurement and the unsteady simulation for PL.

In the following analysis of the pressure signal the normalized pressure was used to obtain a dimensionless representation for the spectrum. Therefore the vertical axis represents the
normalized pressure amplitude, which is defined as $\frac{p - p_{\text{average}}}{p_{\text{average}}}$. On the horizontal axis the frequency divided by the rotational frequency is used.

### 3.2.1. BEP and HL

For BEP and HL the dominant frequencies at the sensor position in the vaneless space are represented by the blade passing frequency, fig.6 (a) & (b). Contrary to the pressure signal in the vaneless space the frequencies in the draft tube are less dominant and therefore had to be recorded over a longer period of time, in order to get enough information to distinguish between the separate peaks.

![BEP and HL](image)

Figure 6: Pressure signal in the vaneless space for BEP and HL (VL2).

![FFT-analysis in the draft tube and vaneless space](image)

(a) FFT-analysis in the draft tube (DT5). (b) FFT-analysis in the vaneless space (VL2).

Figure 7: FFT-analysis of the pressure signals at PL condition.

### 3.2.2. PL

The dominant frequencies for PL condition also correspond to the blade passing frequencies in
the vaneless space, fig.7 (b). Even in the draft tube location these frequencies can be detected (fig.7 (a)). For the draft tube pressure signal there is also a remarkable peak at 155Hz, which corresponds to 28 times the rotation frequency. For an accurate identification of the origin of this frequency it is necessary to record additional data, which requires additional investigations.

Again, also for the pressure signal at PL condition the recording duration was relatively small, fig.5 (a)&(b). This is one reason why not all significant frequencies can be captured and represented by the FFT. In the time signal of fig.5 (a) another distinct oscillation can be recognized for the measurement as well as for the simulation. In the figure this oscillation is represented by the black solid line. This relatively slow oscillation of the simulated pressure signal has a frequency of approx. 1.2-1.4Hz which correlates to 20%-25% of the runner frequency. The frequency can be attributed to the vortex rope in the draft tube cone [1].

3.3. Velocity probes

In the second Francis 99 workshop, velocity measurements in the draft tube are provided. The measurements were realized with the use of Particle Image Velocimetry (PIV). Unlike in the first workshop, the velocity-components in axial and radial direction are given, instead of the tangential component. The measurement is given on three sampled lines, two in the horizontal direction and one in the vertical direction at the center of the draft tube cone, fig.8. For more detailed information about the measurement set up see [10]. Because of the small amplitudes of the radial velocity component only the axial velocity is analyzed in the following. Subsequently the averaged velocity component in axial direction is plotted over the sampled lines. Because the measurements are recorded with a relatively low sampling rate of 40Hz, the complete provided data set was used for the averaging over time. This corresponds to 200 discrete values for each point on the line. An evaluation of the signal for just one channel rotation (360°) could not be performed, because there are only seven samples per runner rotation and therefore the signal can not be captured properly. For this reason the simulated velocity signals were averaged over one runner revolution for each point on the line. For each point 240 discrete values are used for the averaging. The velocity components are normalized with the average meridian velocity (volume flow rate divided by the area at the draft tube entry). The horizontal values are plotted from left to right and the vertical ones from bottom to top.

3.3.1. BEP

Line 1&2 (horizontal):
For the BEP it can be shown, that the axial flow behavior can be captured with the unsteady and the steady state simulation. One reason for the good agreement of the steady state simulation with the measurements is caused by the axial symmetric flow behavior in the draft tube which is given at the BEP. For this case the averaging over the circumference of the draft tube entry leads to no additional error. An other reason for the good agreement between simulation and measurement is due to the relatively small velocity gradients along the measured line (fig. 9) at BEP conditions. Therefore it is easier for the simulation model to capture the flow field here, rather than in other operation points.

![Figure 9: Normalized axial velocity at BEP.](image)

3.3.2. HL
Line 1&2 (horizontal):
Compared with the BEP-case the operation point at high load condition shows much higher velocity gradients across the measured line, fig.10 (a). This leads to greater discrepancies between

![Figure 10: Normalized axial velocity at HL.](image)
measurement and simulation. Both, the unsteady and the steady state approach, have more problems to capture the flow field. Nevertheless both are capable to qualitatively reproduce the profile of the flow and also generate a solution on which the physical behavior is comparable to that of the measurement. Again the simulation model with mixing plane can benefit from axial symmetric flow behavior like the BEP.

![Normalized axial velocity at HL](image)

**Figure 10**: Normalized axial velocity at HL.

Line 1&2 (horizontal):
For PL condition a different flow behavior between measurement and simulation can be recognized. The simulation shows a backflow caused by the vortex rope in the draft tube. For both steady state and unsteady simulations the vortex rope is much more distinct than in the measurement. Remarkable is also, that the steady state simulation is capable to predict the draft tube flow with a similar accuracy as the unsteady simulation. The reason is, as already observed in the last workshop [1], that the vortex rope in the upper part of the draft tube is nearly axisymmetric. Therefore, averaging over the circumference again has a much smaller influence compared to a flow field with an excinicentric vortex rope attached to the runner.

Line 3 (vertical):
The backflow region, which can only be observed for the simulation results, relates to an offset and opposite sign for the axial velocity component compared to the measurement for the vertical line.

3.3.3. Vortex rope
In fig. 13 it can be seen, that for the locations of the velocity measurements the vortex rope for PL condition is almost in the center of the draft tube cone. The vortex has been found by applying the an isosurface with constant value for the Q-criterion on the simulation results. The Q-criterion is defined as positive second invariant for $\nabla \mathbf{u}$. It represents the local balance between shear strain rate and vorticity magnitude. Vortices are areas where the vorticity magnitude is greater than the magnitude of rate-of-strain [5].
Conclusion
The CFD simulations could successfully be conducted within the scope of the Francis 99 workshop. Steady-state case and the simulation with the moving runner both overestimate the integral quantities of the baseline measurements provided for this study in terms of torque and head. Since head increases the hydraulic power as much as higher torque increases the mechanical power, the hydraulic efficiency is only marginally different between simulations and measurements. However, unlike the head which is virtually the same for steady state vs. unsteady, the torque is dissimilar between steady state and unsteady throughout the study. A possible reason is found in the different rotor-stator interface treatment. The mixing plane at the steady state interface interferes with separation zones at the stay vanes upstream of the interface. This results in higher torque output for the steady state simulations, especially for PL and BEP.

Furthermore, FFT evaluations of the pressure signals from the simulations and from the measurements have shown that a large amount of frequencies are not reproducible with CFD due to effects caused by the setup of the measurement rig itself. As a result, only few frequencies can be compared among CFD and measurement. Mainly the blade passing frequency for BEP, PL and HL at 30 times the runner frequency and the frequency of the vortex rope in PL at 0.2 times the runner frequency can be detected with certainty. An additional frequency at approx. 28 times the runner frequency in the draft tube at PL can not be associated with any known physical effects and still has to be investigated in detail.

In addition, time-averaged values for axial velocity are evaluated here. The difference between the operating points can clearly be seen when considering the axial velocity on the three sampling lines within the draft tube. The axial velocity component on the axis of rotation confirms the presence of the vortex rope which is already indicated by the FFT evaluation of the pressure probes.

In conclusion, the different effects found here can be observed with both, steady state and unsteady simulations, which is especially noteworthy for the vortex rope due to its axial-centric properties for this PL operating point here. However, a small penalty in error of torque due to
the averaging process in the mixing plane of the steady state simulation is inevitable. Overall, BEP and HL agree best with the velocity measurements whereas the PL simulations are more critical in accuracy due to the increased complexity of the flow field. In essence, the steady state simulations of the three operating points, using a coupled incompressible solver, provide accurate information with manageable use of computational resources.

References
[1] B Aakti, O Amstutz, E Casartelli, G Romanelli, and L Mangani. On the performance of a high head francis turbine at design and off-design conditions. In Journal of Physics: Conference Series, volume 579, page 012010. IOP Publishing, 2015.
[2] Ansys. ANSYS ICEM CFD Help Manual. -, Canonsburg, - edition, 2013.
[3] Users Guide FLUENT. Fluent inc. Lebanon, USA, 2003.
[4] L Hanimann, L Mangani, E Casartelli, T Mokulys, and S Mauri. Development of a novel mixing plane interface using a fully implicit averaging for stage analysis. Journal of Turbomachinery, 136(8):081010, 2014.
[5] V Kolár. Vortex identification: New requirements and limitations. International journal of heat and fluid flow, 28(4):638–652, 2007.
[6] MV Magnoli. Numerical simulation of pressure oscillations in francis turbine runners. JASS Numerical Simulation of Turbomachinery, pages 32–50, 2009.
[7] L Mangani, M Buchmayr, and M Darwish. Development of a novel fully coupled solver in openfoam: Steady-state incompressible turbulent flows. Numerical Heat Transfer, Part B: Fundamentals, 66(1):1–20, 2014.
[8] F Moukalled, L Mangani, and M Darwish. The Finite Volume Method in Computational Fluid Dynamics: An Advanced Introduction with OpenFOAM® and Matlab, volume 113. Springer, 2015.
[9] NTNU and LTU. Francis-99 workshop. http://www.ltu.se/research/subjects/Stromningslara/Konferenser/Francis-99?l=en accessed august 2014.
[10] NVKS. Francis-99 second workshop. https://www.ntnu.edu/nvks/second-workshop accessed september 2016.
[11] C Trivedi, MJ Cervantes, BK Gandhi, and GO Dahlhaug. Experimental investigations of transient pressure variations in a high head model francis turbine during start-up and shutdown. Journal of Hydrodynamics, Ser. B, 26(2):277–290, 2014.
[12] JP Yan, U Seidel, and J Koutrak. Numerical simulation of hydrodynamics in a pump-turbine at off-design operating conditions in turbine mode. In IOP Conference Series: Earth and Environmental Science, volume 15, page 032041. IOP Publishing, 2012.