Stationary and Transient Numerical Simulation of a Bulb Turbine

Helmut Benigni, Helmut Jaberg
Institute for Hydraulic Fluidmachinery
University of Technology Graz
Kopernikusgasse 24 IV, A-8010 Graz
Austria
www.hfm.tugraz.at

Abstract: In this paper, the commercial 3D Navier-Stokes CFD-solver Ansys CFX was used to investigate the flow through a horizontal shaft bulb turbine in a stationary and transient way. The draft tube is one of the most critical parts for the performance of bulb turbines and highly influences the efficiency of the whole configuration with a level of several percent of the total efficiency. A calculation of the existing situation was the basis for a modification and adaptation of the draft tube geometry to realize a shorter machine construction and also to compare the results with hydraulic model tests from a closed loop turbine test stand. For all influences on the draft tube rotor stator effects are decisive – in reality as well as in numerical simulations. In a first part, several modifications were analyzed, e.g. inlet variations, as those cause different space requirements of the generator as far as the flow rate is concerned, the operating points differ more than 300%. For the flow through the runner the meridional velocity was analyzed on 5 different planes around the runner. In front and also after the runner the flow over the radius was nearly on an equal level – this means that the hydraulic is good and that no fluid transport in radial direction is needed.

In a second part, an investigation of different influences of rotor stator interactions was carried out, based on a large full 360°-model of the whole machine configuration of this 3-blade runner and 16 guide vane turbine. The simulations were done after tests with different grids, turbulence models, grid interfaces and settings. To compare the simulation results with experimental test results, the histogram pressure method for the sigma calculation developed by the Astroe-team was used. For the flow through the runner the meridional velocity was analyzed on different planes around the runner. In front and also after the runner the flow over the radius was nearly on an equal level – this means that the hydraulic is good and that no fluid transport in radial direction is needed. After the hub a zone of very low velocities was estimated. For best efficiency in the draft tube a good balance between separation and reattachment of the flow is of importance.

For stationary calculations remarkable discrepancies in the draft tube flow distribution are known, which was also proved and analyzed by means of a transient simulation. There could be found separations for a short draft tube. We could verify and check the test data with the simulation for the client and give him a good impression of the complex physics of the draft tube flow and also of the other components of the machine. A further optimization of the whole geometry and more complex transient simulations are possible, as well as an adaptation of the turbine for different operation points and installation schemes. The model runner diameter was D=340 mm with a specific speed of approx. n_q=210 min^{-1} near the best point for the 3-blade runner configuration with 16 guide vanes.

Key-Words: bulb turbine, CFD, rotor stator, transient simulation, histogram analysis

1 Introduction

The investigation of the flow through a horizontal shaft bulb turbine and the comparison with the results achieved at a closed loop turbine test stand are the topic of the present paper. The draft tube highly influences the performance of bulb turbines. To save building costs (e.q. civil parts), the whole machine configuration was optimised and investigated in order to become shorter – this is of practical interest for the design of new machines but also for the rehabilitation of existing ones.

During the last years computational fluid dynamics (CFD) have been used routinely within the R&D process of hydraulic machinery. By means of CFD the development time of turbines can be reduced considerably and most of the time- and cost-intensive experimental investigations can be skipped. Normally, commercial codes Reynolds-averaged Navier-Stokes equations combined with a two-equation robust turbulence model are used for engineering applications. For time saving purposes, the stationary mode with interfaces between stationary and rotary parts as well as the use of a lot of averaging techniques are standard.In this work, the predictive quality of CFD has been carefully validated by the comparison of numerical and experimental data of the existing geometry. Reasonable correlation was the motivation to optimize the hydraulic shape by means of numerical flow simulation. To carry out single-phase calculations, but also to get accurate information on cavitation and sigma values, the

validated method of histogram analysis known from pump development by the Astroe team [4] was used .
The runner geometry given has been realized in several machine configurations all over the world in three- and four-blade runner configurations. For both installations different operating points were calculated in order to validate the existing geometry and the runner-guide vane correlation for all runner positions. Thus, a wide range of measured shells was investigated and not only the single operating points. The basis of those comparisons are measurements of the machine configuration on a closed loop turbine test stand. The draft tube of a hydraulic turbine is a significant part of the machine configuration, where kinetic energy is converted into static pressure – and this energy recovery strongly effects efficiency. In Fig. 1 the mass flow averaged pressure (static and total pressure in stationary frame) is lined out versus the length, where the domains of numerical simulation always run in a standardized way, between zero and one. In the section of the draft tube the increasing static pressure is clearly lined out. The operating point is near the optimum of the 3-blade runner configuration with the optimum guide vane position of 35 degrees, which is lined out in red. In general, the flow in the draft tube is characterized by self-excited unsteadiness, e.g. vortex shedding or vortex rope, as well as by externally forced unsteadiness induced by rotor blades [3]. The model runner diameter was D=340 mm with a specific speed of n_q=210 min^{-1} near the best point for the 3-blade runner configuration with 16 guide vanes.
The experimental data for the two configurations were carried out on a closed loop turbine test stand, which ensures efficiency with measurement errors smaller than +/-0.2 % and repeatable single efficiencies of about +/-0.1%. The closed circuit allows to adjust any absolute pressure (0.1 bar up to 5 bar absolute) in the tailwater tank, to simulate any cavitation conditions. The motor generator (turbine output) is equipped with a frictionless bearing assembly.

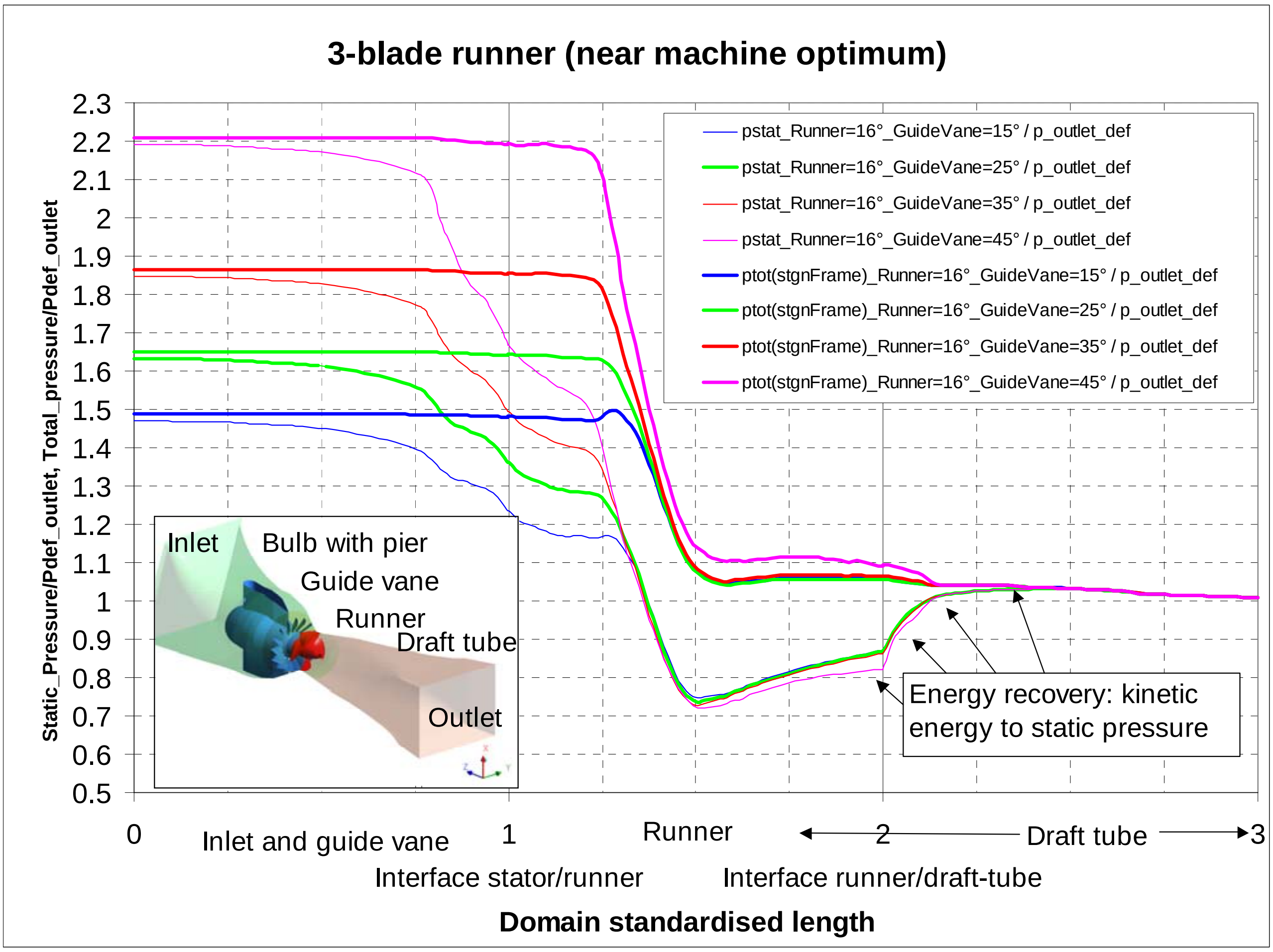

Fig. 1 – Total and static pressure along a 3-blade runner bulb turbine

2 Numerical Analysis

For the numerical simulation of the turbine the commercial CFD-software package ANSYS CFX 5.7 and CFX 10 were used. For the runner, the guide vane and the draft tube structural grids were generated, for the final complete machine simulation also an additional inflow area was modelled with an unstructured grid. For the turbulence modelling, finally, the SST-model

(Menter [7],[1]) was chosen for the stationary points of the shell and the transient calculations. The settings used were already tested at the institute several times before, in order to simulate hydraulic machines, and show results very close to reality [3]. With a distance of 1 or 2 degrees a total of approx. 30 (!) runner-position grids for the three- and four-blade configuration was generated. The high number of runner positions results from operating points ranging from the lowest (Q_{11} = 700 l/s) up to four times higher operating points (Q_{11} = 3750 l/s), which cover the whole range of runner adjustments. The enlargement of the blades on the hub, based on casting requirements, was modelled as well, although the effect on the simulation is low, as the bulk of the flow, where energy is converted, concentrates on higher radii. For all runner grids the tip clearence between the runner blade and the shroud was modelled and realised with 15 layers.

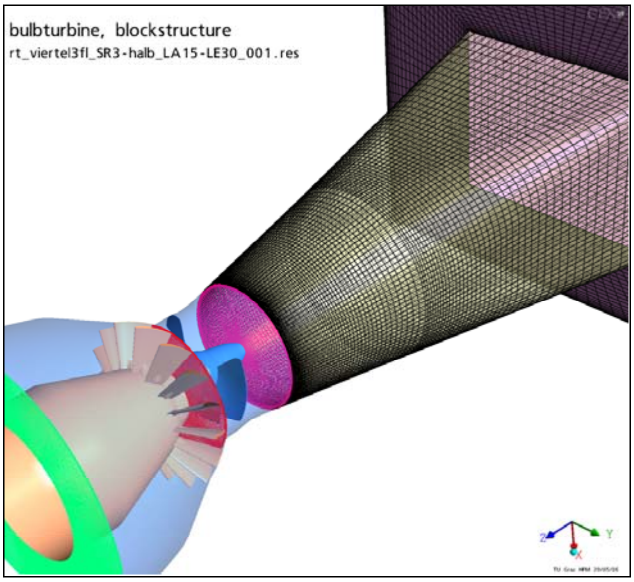

Fig. 2 block structure of the CFD - model

The turning of the guide vane was modelled with 5°-distances, starting with 10° and going up to 80°, and a grid for the blade geometry including the casted enlargement of the hub was generated. Only for very high angles the casted enlargement was neglected because of the needs of grid generation The draft tube was given special attention, as it is very important for the simulation of this type of machine. A fully scripted structured hexaeder grid was generated with TASCgrid V12.1 for both investigated draft tubes, including downstream water with a total of 600000 nodes. The downstream water causes an abrupt growth of the enlargement, an extension of the simulation area and thus well-defined flow conditions at the end of the draft tube, as the hard boundary outflow condition is set at the downstream water exit. So, there is no influence of the outflow condition on the flow distribution in the draft tube. The full machine simulation yields to a grid with 8025039 nodes at 9313384 elements. In Fig. 2 the block structure is figured out, where the red grid surface is the guide vane-rotor interface and the pink grid surface is the runner-draft tube interface.

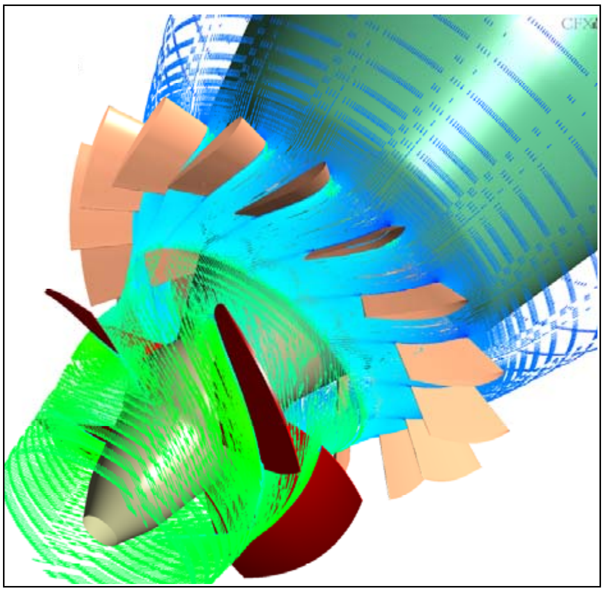

Fig. 3 Flow through the turbine

Unlike the simulation with the full 360° guide vane, the runner and the draft tube, the shell was carried out with a model with one guide vane (22.5°), one runner passage (90°) and half of the draft tube (180°). Thus, a maximum of possible periodic and symmetric boundaries was used. With these settings the influence of the pier is disregarded – grid size amounts to 1259265 nodes at 1195212 elements (only hexaeder).

Boundary conditions were set in the same way, according to the institute's experience with other simulations. As the solver gives mass flow highest priority, we used a mass flow boundary condition at the inlet and a pressure boundary condition at the outlet. So, mass flow was constant and the pressure head resulted from the calculations, which means vertical cuts through the shell. For the solver we used CFX10 – the latest software version.

For the sigma estimation the pressure on the blade was written out and spread on for histogram anlysis: This method was developed by Astroe and was cross-checked several times [[4]]. $p_{Histogram}$ is the value, when the pressure at a certain percentage of the blade surface exhibits pressures lower than $p_{Histogram}$. This value is transferred to a sigma denomination (see nomenclature). Theoretically, the conclusion for $\sigma_{HISTOGRAM} = -\sigma_{Installation}$ is, that this percentage of the blade surface is cavitating. Experimental values:

$\sigma_{CFD,\ HISTOGRAM,\ 0.005}$ is equivalent to 1.2 $*\sigma_{STANDARD}$.

3 Results

This operating point with Q_{11} = 1700 [l/s] has the highest measured efficiency of all calculated operating points of the 4-blade runner. In the diagram the efficiency measured is lined out with the grey line. It can clearly be seen, that the best behaviour related to the cavitation shows with heads lower than 1.2 of the design head.
Both, efficiency and sigma are very close to the experimental results and show the same tendencies towards the bordering regions of the shell. Best values are achieved with a runner position of 17 degrees.

3.1 Flow through the runner

In order to get an impression of how good the runner works, the meridional velocity (c_m) was analised on different planes around the runner – to be presented in this paper for three different operating points (Q_{11}=800, Q_{11}=1700 and Q_{11}=2800) for the 4-blade runner configuration. The planes were situated in the different CFD-domains of the model (see Fig. 5 top left), the first plane (labelled with 0.97) was situated near the stator-rotor (frozen rotor) interface in the stator domain on the normalised domain length of 97%. The next three planes were situated in the runner domain at 10% (in front of the blades), 65% and 90% (after the runner) of the normalised domain length (labelled with 1.1, 1.65 and 1.9). The last plane was situated after the frozen rotor interface of the runner-draft tube connection in the draft tube domain at 3% of the domain length, labelled with 2.03. All the planes are located on standardised domain lengths that thus are not planar in radial direction. Each of the operating points has its local optimum, the flow rates differ by more than 300%. For all operating points a homogeneous velocity distribution in front of the runner exists, which indicates a good guide vane geometry and is lined out in Fig. 5 with the red and green lines corresponding to the coloured planes in the picture on the top of Fig. 5. This could also be detected when – for each turbine component – the efficiency in relation to the total efficiency was compared to other hydraulics.

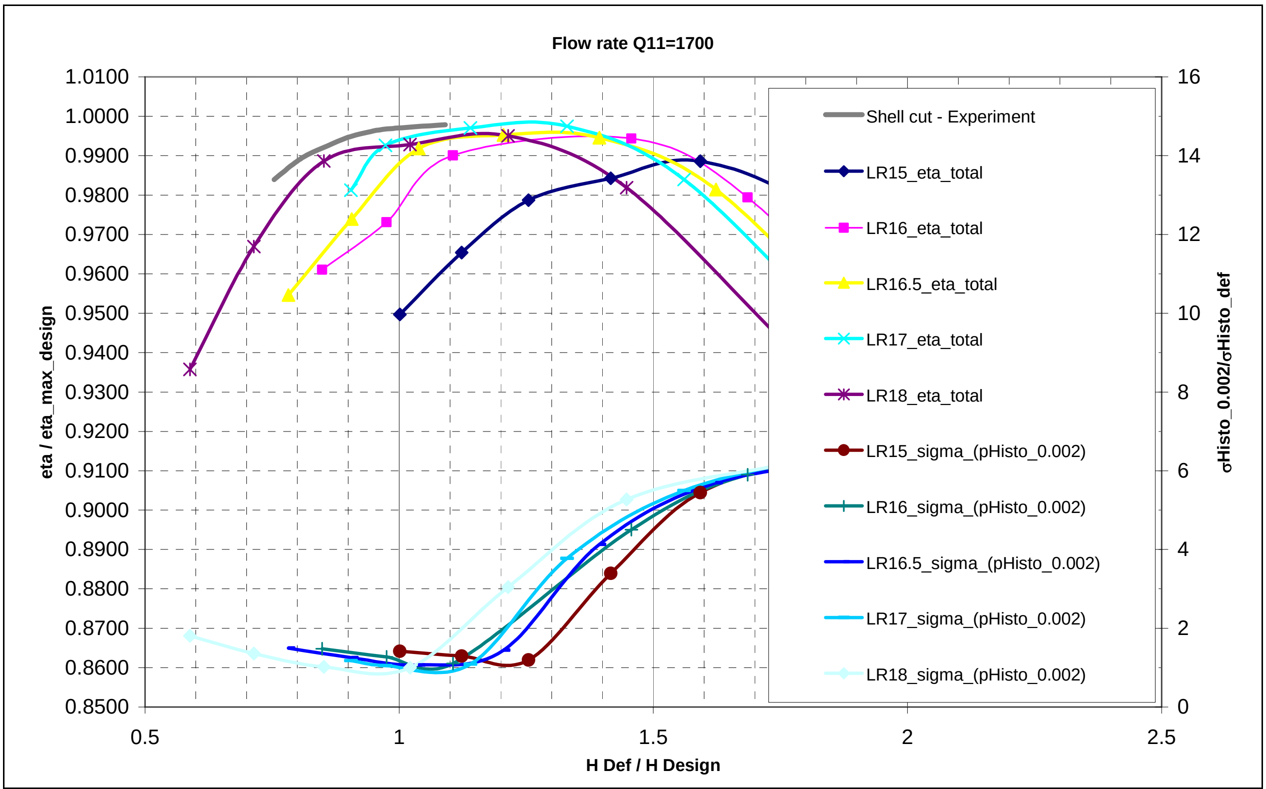

Fig. 4 – Normalised efficiency and $p_{Histogram}$ for the4-blade runner configuration (near shell optimum)

After the runner there could be seen a cm-distribution with nearly the same value over the whole radius. This also means, that the runner works fine over its height and that only little fluid transportation in radius direction is needed, which can be found out by comparing the velocity in radial direction (not shown). After the hub, where the rotor-draft tube interface is located, the cm is coloured in yellow. In the middle of the draft tube velocities are very low – this area is also one of the most unsecure parts of the steady state simulation.

3.2 Stationary and transient results

The steady state simulation exhibits a wide range of regions with low velocity fields after the hub (see Fig. 5). One of the important aspects for shorter draft tube construction are separation zones and the correlated

efficiency losses. For the transient simulation presented in this paper the SST-model of Menter was used, with an implicit time discretion scheme and second order accurate time integration. A steady state calculation was used as initial guess for the transient calculation of the full 360° model simulation. After 6 revolutions of the runner the result was considered as periodic in time.
Experience from built-in situations shows, that separation effects of short building draft tubes (with large opening angles) have an influence on the efficiency up to several percent of the total efficiency. With a high discharge operating point in the transient simulation the calculation shows a separation near the wall. Also, strong changes of the flow distribution in the center can be seen in the transient simulation. In the center of the draft tube remarkable discrepancies could be found as compared to known experimental flow distribution of a comparable bulb turbine configuration [3]. The calculation still under predicts the mixing of the runner wake flow with the main flow. Reflections on this effect are given in [3] and [8].
The hydraulic design reaches best efficiency when the separation and reattachment of the fluid in the draft tube is in balance and so the diffusor has its best efficiency – the highest amount of transformation of velocity energy into static pressure. At modern low pressure turbines the amount of retransferable energy is about 0.5% to 5% of the hydraulic energy [5].
In Fig. 6 the transient data are lined out for head, flow rate and torque over the calculation time (revolutions). When transferring this data with a Fourier analysis a peak could found at runner frequency.

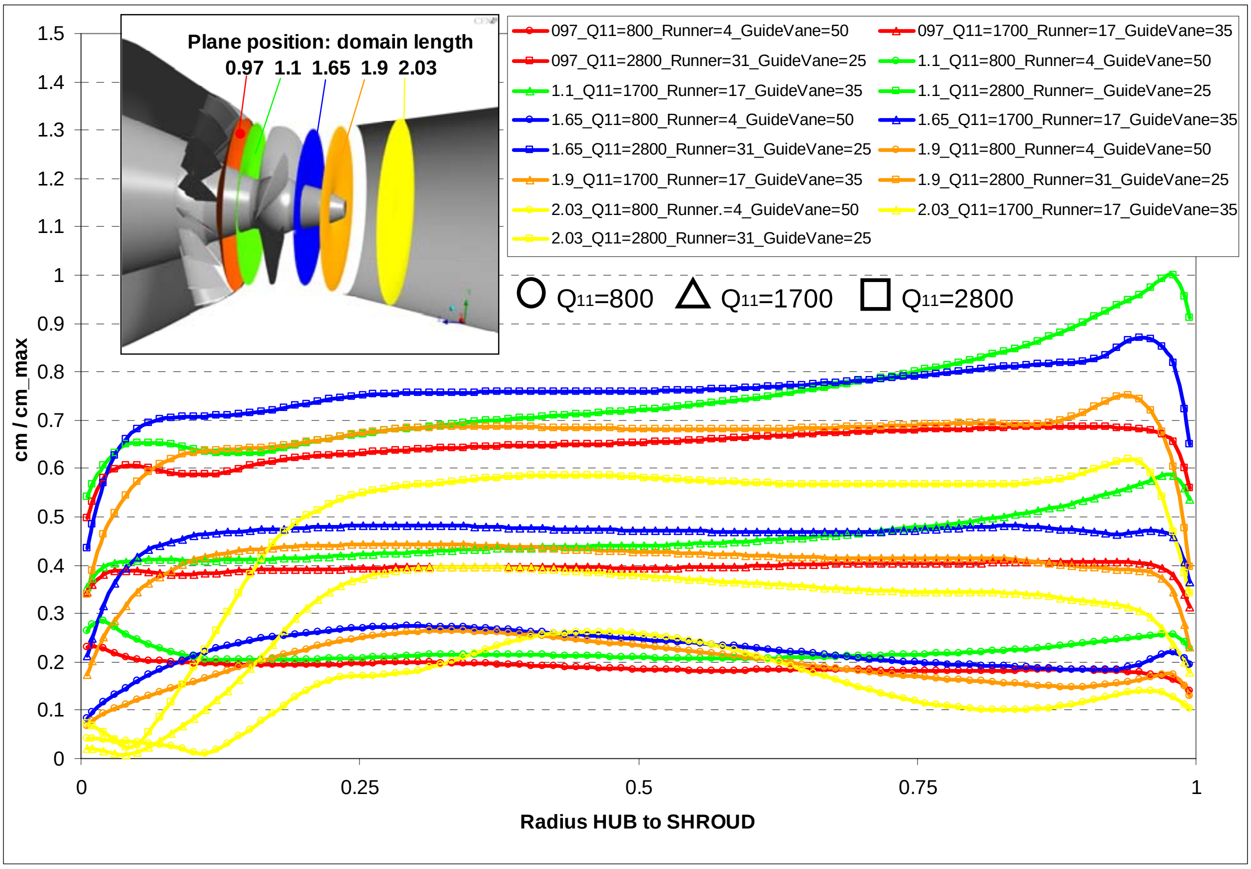

Fig. 5 – Velocity distribution around the runner, different operating points

4 Conclusion

With the help of commercial CFD-codes the investigation of an existing geometry of a horizontal bulb turbine was carried out. For the better understanding of the flow through the passages they were analised and modified, having in mind the idea to build a shorter turbine configuration, i.e. a shorter draft tube. For different operating points of the blade geometry, installed as 3- and 4-blade configurations, all significant hydraulic variables were calculated, including a histogram pressure analysis for the sigma estimation. An excellent correlation between experimental data, carried out on a closed turbine test loop, and the simulation data could be estimated for different operating points. For the separation zones near the wall of the draft tube also a transient calculation was carried out to proof the findings in comparison to other simulated hydraulics.

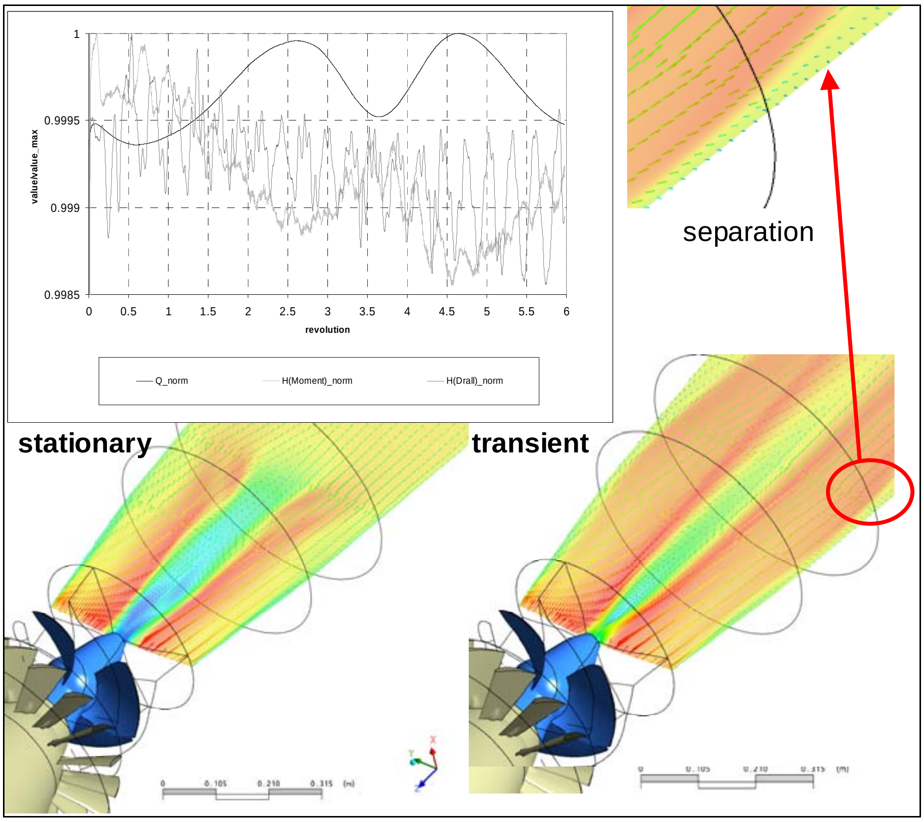

Fig. 6 – Velocity in a short draft tube, same operating point stationary and transient

Nomenclature:

Head

$$H_{Net\_Head} = \frac{1}{\rho \cdot g}\left\{p_{tot,Inlet} - \left[p_{stat,outlet} + \frac{\rho}{2}\left(\frac{\dot{Q}}{A}\right)^2\right]\right\} \tag{1}$$

Sigma

$$\sigma_{HISTOGRAM} = \frac{p_{Histogram} - p_{tot,outlet}}{p_{tot,inlet} - p_{tot,outlet}} \tag{2}$$

Specific speed

$$n_q = n \cdot \frac{\sqrt{Q}}{H_{Def}^{\ 0.75}} \tag{3}$$

Standardised discharge

$$Q_{11} = \frac{Q_{Model}}{\sqrt{H_{Model}} \cdot D_{Model}^{\ 2}} \tag{4}$$

Standardised rotational speed

$$n_{11} = n_{Model} \frac{D_{Model}}{\sqrt{H_{Model}}} \tag{5}$$

References:

[1] CFX, “Documentation of Ansys CFX5.7”, P*DF-files of the documentaion on installation CD, Ansys Inc., 2004.*

[2] Gehrer, A., Egger A., Riener J., “Numerical and Experimental Investigation of the draft tube flow downstream of a bulb turbine”, *Proceedings of the 21st IAHR Symposium on Hydraulic Machines and Systems, Lausanne, September 9-12, 2002.*

[3] Gehrer, A., Benigni, H., Köstenberger, M., “Unsteady Simulation of the Flow Through a Horizontal-Shaft Bulb Turbine”, *Proceedings of the 22nd IAHR Symposium on Hydraulic Maschines and Systems, Stockholm, 2004.*

[4] Gehrer, A., Benigni, H., Penninger, G., “Dimensioning and Simulation of Process Pumps”, *Karlsruhe Pump Users Technical Forum, Preprints 15-4, VDMA, 2004.*

[5] Giesecke, J., Mosonyi, E., „Wasserkraftanlagen”, ISBN 3-540-64907-7, 2. Auflage, Springer Verlag, 1998.

[6] Launder, B. E. , Spalding, D. B., „Lectures in Mathematical Models of Turbulence”, Academic Press, London and New York, ISBN 0-12-438050-6, 1972.

[7] Menter, F.R., “Two-equation eddy-viscosity turbulence models for engineering applications”, *AIAA-Journal, 32 (8), 1994.*

[8] Ruprecht, A., Maihöfer, M., Heitle M., Helmrich, T., “Simulation of vortex rope in a turbine draft tube”, *Proceedings of the 21st IAHR Symposium on Hydraulic Machines and Systems, Lausanne, September 9-12, 2002.*