ABSTRACT
The unsteady simulation of an entire Francis turbine is shown. The investigated turbine consists of spiral case, tandem cascade with 24 channels, runner with 13 blades and draft tube. The computational boundary conditions are applied at the spiral inlet and at the draft tube outlet. In the calculation no periodicity is assumed and no circumferential averaging is applied. Therefore all tandem cascade and runner channels have to be modeled. The computational algorithm including parallelization and rotor/stator interface is briefly described. The rotor/stator interface is realized by means of dynamic boundary conditions and allow non-matching grids. Selected computational results are shown for two points of operation.

INTRODUCTION
For design and research in hydro turbines the application of computational methods is steadily increasing. Nowadays it is fairly standard to simulate the different components separately. By assuming steady state uniform flow conditions in circumferencial direction only one channel of the tandem cascade (stay vanes and guide vanes) and of the runner has to be considered. Since very often there are strong interactions between the components (especially between guide vanes, runner and draft tube) it is inevitable to introduce this interaction into the simulation for accurate results. Many attempts have been made lately to do this by applying an averaging procedure in the circumferential direction. This still allows to consider only one channel of tandem cascade and runner and to apply a steady state computation, which results in a severe saving of computer resources. Different approximations are compared in [1]. The obtained results agree quite well with measurements and can be applied for turbine design, e.g. [2,3,4].

However, many problems arise due to dynamical effects and vibrations. Dynamical forces can not be obtained by the steady state approach. Also the measurement of the dynamical forces is quite complicate and requires a rather high measuring effort. Therefore it is desirable to obtain it by simulation. For the calculation of these dynamical effects it is essential to apply an unsteady calculation of flow including the dynamical stator - rotor interaction. Due to the non-uniform inflow from the spiral case and due to the unequal pitching of guide vanes and runner it is necessary to consider the entire turbine with all channels of the runner and of the tandem cascade. This results in an extremely high computational effort, which can only be performed in parallel on powerful computers with reasonable response times.

In this paper the unsteady simulation of an entire Francis turbine is shown. The investigated turbine consists of spiral case, tandem cascade with 24 channels, runner with 13 blades and draft tube. The computational algorithm including parallelization and rotor/stator interface is described and computational results are shown.
BASICS AND NUMERICAL PROCEDURES

The calculations are based on the Reynolds averaged Navier-Stokes equations. The turbulence is taken into account either by an algebraic model or by the standard k-ε model including the “hybrid” Kato-Launder correction \[5\]. The spatial discretization is obtained by introducing a Petrov-Galerkin FEM with tri-linear hexahedral elements. For the time discretization a 2\textsuperscript{nd} order three-level fully implicit scheme is applied. The implicit scheme has the advantage that the description of the time step is only dependent on the physics and not on the numerical mesh.

Because of the consideration of unsymmetrical flow in spiral case the inflow to tandem cascade and consequently the runner can be unsymmetrical as well. Together with the unequal pitching of guide vanes and runner, this leads to the fact, that no flow periodicity exists. Therefore the complete turbine including all flow channels in the tandem cascade and in the runner has to be considered. This represents the most general approach for the prediction of rotor/stator interactions, but it requires huge computational grids. Other methods e.g. applying time periodicity can reduce the computational efforts, they however include certain restrictions.

The whole calculation domain is divided into the four components (spiral case, tandem cascade, runner, draft tube). For each component an independent computational grid is generated. The different grids are slightly overlapping. They do not have to match at the boundaries. The calculation of the four components is carried out simultaneously. The connection between the components is realized by dynamic boundary conditions. In downstream direction the interpolated node values for velocities and turbulence quantities are exchanged. In upstream direction the pressure and the fluxes are prescribed in form of surface integrals, see fig. 1.

Each of the components is also calculated in

---

Fig. 1: Dynamic boundary conditions

---

Fig. 2: Solution procedure
parallel by domain decomposition, therefore the computational grids are divided into parts of equal sizes. The parallelization is performed in the linear equation solver. The runner is calculated in an rotating frame of reference. For details concerning the algorithm and the performance of the parallelization the reader is referred to [7,8]. The simulation procedure is summarized in fig. 2. It shall be mentioned that because of the rotating runner the exchange of the boundary conditions between the components leads to a dynamical connection of the different processors. This is schematically shown in fig. 3.

It has to be mentioned, that because of the complex geometry rather large grids (millions of nodes) are necessary. Even for comparatively coarse mesh the unsteady simulation leads to a huge amount of data. The postprocessing of these data requires a very efficient and powerful hardware equipment. At the University Computer Center a VR-Cube with a powerful 14-processor workstation for postprocessing is available. This allows a quit fast and efficient evaluation of the simulation data. A look in the Cube is shown in fig. 4. In spite of the powerful tools, however, the data evaluation is still very time consuming. Consequently it would be desirable to have intelligent software routines, which automatically can investigate the data in order to detect critical flow regions (e. g. vortices). Otherwise with increasingly fine grids and finer time steps not all important information included in the simulation data can be recognized by the engineer. This will be a future field of research work.

**INVESTIGATED GEOMETRY AND COMPUTATIONAL GRIDS**

The investigated turbine is a Francis turbine consisting of the spiral case with 23 stay vanes and a tip nose. These are followed by 24 guide vanes and the runner with 13 blades. At the outlet a draft tube with two piers is located. The computational mesh of the turbine is shown in fig. 5. Different colors represent different processors for the calculation. For details also one channel of the tandem cascade is presented in the figure.
The whole computational grid consists of more than 2 million grid nodes. It is divided into 96 parts with the approximately equal number of nodes. During the calculation each part is run on its own processor. The simulation has been carried out on the CRAY T3E/512 of the University Computer Center. The time step for the calculation was chosen to obtain 1.15 degree of runner rotation per time step.

As boundary conditions a fully developed pipe flow at the spiral case inlet and free outflow at the draft tube outlet were assumed. The initial condition was obtained by a steady state calculation with a frozen rotor approach. It has to be mentioned, that the grids are rather coarse. Especially in the tandem cascade and in the runner finer grids are required. Investigations with finer grids are in progress.

**RESULTS**

The calculations were carried out for two different points of operation
- part load and
- optimum.

At first the overall overview of the results are presented for the part load point of operation. In fig. 6 the instantaneous pressure distribution in the tandem cascade and in the runner is presented for a certain time step. Clearly the reduction from the high pressure (red) in the spiral case to the low pressure (blue) at the runner outlet can be observed. At the runner inlet the stagnation point at the leading edge is also clearly visible. Looking at the stay vane inlet the unsymmetrical pressure distribution in circumferential direction can be seen, which results from the non-uniform flow distribution of the spiral case.
Looking on the pressure distribution at the stay vanes one can also observe a strong change along the height. At the top of the stay vanes the pressure is quite high whereas at the bottom the pressure is low. The reason is the strong turning of the flow from the radial to the axial direction. This behavior can also be observed in the velocity distribution, shown in fig. 7. There the magnitude of the absolute velocity for a certain time step is shown in the mid plane of the tandem cascade and at the outlet of the stay vanes. Again the acceleration of the flow on the bottom can be observed clearly, which results in the reduction of the pressure, fig. 6. The unsymmetrical inflow to the tandem cascade can be seen again, which results from the spiral case. Also very obvious are the wake behind the stay vanes. These wakes expand to the runner and causes fluctuations of the surface pressure and consequently also fluctuations of forces and torque.

Fig. 6: Instantaneous pressure distribution for a certain time step

Fig. 7: Instantaneous velocity distribution in the tandem cascade
In fig. 8 the overall flow behavior in the whole turbine shall be presented. For that purpose the velocity distributions on different planes are shown for a fixed time step. One can observe the acceleration from the spiral case into the tandem cascade and the runner and the deceleration of the flow in the draft tube. In the draft tube a very unsymmetrical flow distribution is obtained. The draft tube channel in the middle shows a very low velocity (black color) whereas the channel on the right hand side (in flow direction) has a high discharge (dark blue color).

Especially the draft tube flow strongly depends on the point of operation since the swirl rate behind the runner very much affects the flow behavior. In fig. 9 the instantaneous velocity distribution for optimum condition is shown. Here you can see that the discharge in the middle channel is increased, whereas the channel of the left hand side shows a reduced flow rate.

In fig. 10 the instantaneous distribution in the meridional mid plane is plotted for three different time steps, which means three different positions of the runner. Again it can be observed, that the flow is strongly accelerated towards the runner. This leads to a very fast equalization and consequently to a very fast fill in of the wakes behind the guide vanes. The result of this is, that only a very weak effect of the stator-rotor interaction can be observed. This would be much stronger in pure axial turbines (e. g. bulb turbines), where no strong flow acceleration towards the runner exists, see e. g. [6,9].

Fig. 8: Instantaneous velocity distribution for part load operation point

Fig. 9: Instantaneous velocity distribution for optimum point of operation

Fig. 10: Velocity distribution for three different positions (time steps)
The relatively weak interaction of guide vanes and runner can also be seen in fig. 11. There the variation of the axial force (fig. 11 top) and of the torque (fig. 11 bottom) on a single runner blade are plotted versus the rotation angle of the runner. It can be observed, that the axial force as well as the torque show fluctuations of approximately +/- 2%, which is very low, compared for example with the axial type turbine presented in [6].

Looking at the distribution of the axial force one can see that it shows an almost periodical behavior. Clearly two dominating frequencies can be evaluated. The lower one is caused by the runner speed. The higher fluctuations correspond to the number of guide vanes time the runner speed. In this case 24 small peaks can be counted during one revolution of the runner. The fluctuations caused by the interaction with the guide vanes are quite small. The reason for that is that the wakes behind the guide vanes vanish very early because of the accelerated flow. This could be observed already in fig. 10. The distribution of the torque shows nearly the same behavior than that of the axial force. The amplitude of overall fluctuation is slightly higher. However it is still very low compared e. g. with the axial turbine in [6].

For both quantities, the axial force as well as the torque, fig. 11 shows the distribution on a single runner blade. Looking at the total force or torque the fluctuations are much lower, since the forces on the 13 runner blades have different phases and consequently the sum of the forces (and torques) remain quite constant.

Again it has to be mentioned, that the presented calculations are performed on a rather coarse mesh. This means, that the wakes behind the blades are equalized to early. In addition it has to be emphasized, that the computational effort is quite high. Therefore it is costly to simulate long time periods. However this leads to a severe problem, because the flow in the turbine consists of a huge spectrum of frequencies and initial periods.

Concerning the rotor-stator interaction, for example, one obtains quite fast a “periodic” situation. For this it is enough to calculate 2-3 rotations of the runner. In regard to the vortex rope in the draft tube, however, the calculation period has to be much longer. It takes quite a long time until the vortex rope forms and reaches a “periodic” circulation. This requires the
calculation of at least 15-20 revolutions of the runner, which is extremely costly. For the
above shown results the calculation time therefore was too short to obtain the vortex rope,
which appeared in the model tests. That the calculation procedure is able to predict the vortex
rope, however, is shown in [6]. To overcome this difficulty it is intended to apply a multi-
scale approach with different time steps in order to reduce the computational effort. This
approach is in progress.

**CONCLUSIONS**

The unsteady calculation of the flow in an entire Francis turbine from spiral case through stay
vanes, wicket gates, runner down to the draft tube is presented. No periodicity is applied
neither in the stay vanes and wicket gates nor in the runner. Therefore all flow channels are
modeled. This results in huge computational grids and computational times, which only can
be handled by parallel computing. The coupling between the different components are
performed on sliding non-matching meshes with dynamic boundary conditions, whereas the
parallelization in each component is obtained by domain decomposition and application of a
parallel solver for the linear system of equations.

The calculations were carried out for a turbine with 24 stay vanes and wicket gates and with
13 runner blades and some results are presented. These calculations are intended to be a
feasibility study rather than to obtain major quantitative results, but these and similar
calculations for an axial turbine [7] show that the simulations are feasible and meaningful and
quantitatively accurate results concerning dynamical forces and loading can be expected.
Therefore more accurate solutions on finer grids are in progress. One major difficulty in the
calculation is the presence of quite different flow phenomena, which very much distinguish in
frequencies and initial periods. Applying a single time step for all the problems results in
extensive requirements of computer time. Therefore it is worked on a multi-scale approach.

**Reference**

    simulation, 18th IAHR Symposium Hydraulic Machinery and Cavitation, Valencia, 1996.
    between components, Modelling, Testing & Monitoring for Hydro Powerplants, Lausanne, 1996.
[4] Sick, M., Casey, M. V., Galphin, P.F.: Validation of a stage calculation in a Francis turbine, 18th IAHR
    Passage with Low-Re Two-equation Models, Computational Fluid Dynamics, Wiley & Sons Ltd., 1994.
    rotor interaction, IAHR WG "The Behaviour of Hydraulic Machinery under Steady Oscillatory
    Conditions", Brno, 1999.