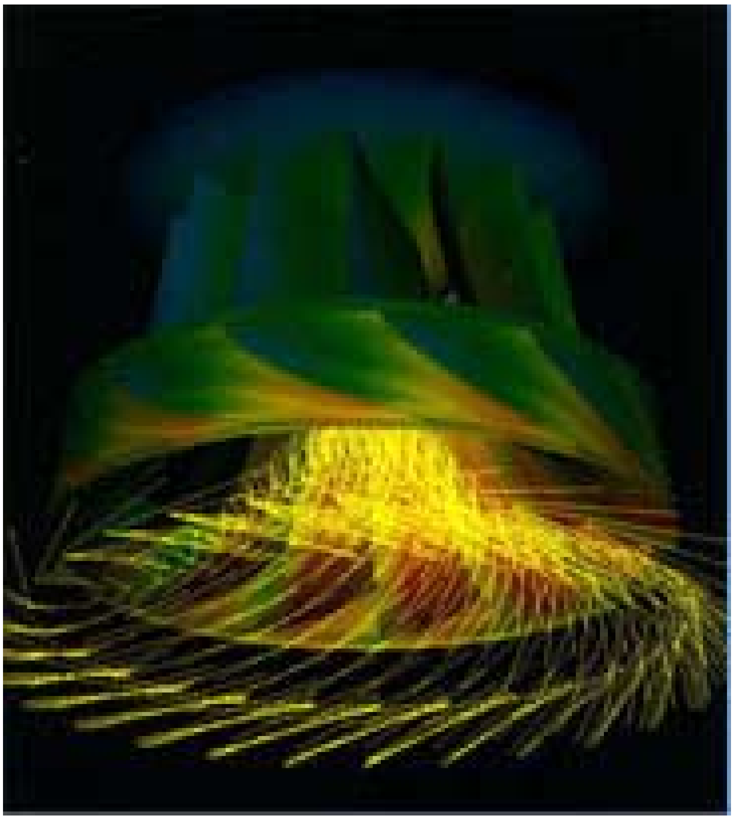
HYDRAULIC TURBINE RESEARCH PROGRAMME

Industrial collaborative R&D programme

Programme description 2000



Swedish National Energy Administration

ELFORSK
SWEDISH ELECTRICAL UTILITIES' R&D COMPANY

GE Hydro

WAPLANS
MEKANISKA VERKSTADS AB

A WORD FROM THE PROGRAMME MANAGER

The *Water Turbine joint research programme* is being financed by Swedish National Energy Administration, hydro electric power companies through Elforsk, GE Energy (Sweden) AB and Waplans Mekaniska Verkstad AB. The Programme finances research and development performed in collaboration between industry and universities in the field of water turbines. Its objective is to meet the need for specialist skills that will be required in order to carry out optimum renewals in future despite the fact that the rate of new construction and renovation is today very slow.

The programme is now in its second three-year phase. This commenced on 1 November 1999 and is continuing till the end of 2002. Its budgetary framework amounts to SEK 20.5 million. This is an expansion in comparison with the previous phase, partly so as to be able to start additional projects within the prioritised areas and partly in order to develop the structure of the programme along the lines of proposals from the evaluation of the first phase. Elforsk is administering the programme and a programme board with the following members has been appointed:

Ulf Wollström, Vattenfall Vattenkraft (Chairman)
Lars D. Jansson, Birka Kraft
Roger Nöjdh, Sycon Energikonsult
Christer Söderberg, Sveriges Energiföreningars Riksorganisation
Håkan Gustavsson, Luleå University of Technology
Hermod Brekke, Norwegian University of Science and Technology
Svante Leonsson, GE Energy (Sweden)
Lars Eric Killian, Waplans Mekaniska Verkstad (coopted)
Maria Malmkvist, Swedish National Energy Administration (coopted)
Cristian Andersson, Elforsk, swedish electrical utilities´ R&D company (Programme Manager, coopted)

It is gratifying to be able to state that commitment to the programme represents wide and deep knowledge through collaboration in programme reference groups and through people from the industry acting as industrial support for doctoral candidates. These contributions come from the founders of the programme, but also from Möller/Undenäs Turbin AB, Turbin- & Regulatorservice AB, ITT Flygt AB and SWECO Industriteknik.

In this pamphlet you can read about the programme and its projects. It contains a report on the objectives of the programme, how it is meeting its targets, results and position reports showing the current status of the research projects. It also contains references to published scientific results.

Cristian Andersson, Programme Manager, Water Turbine Engineering, Elforsk AB

CONTENTS

1 THE RESEARCH PROGRAMME

1.1 Relevance to industry and energy

Every scenario emphasises hydroelectric power as a basis for future electricity supplies in Sweden. The Swedish power industry is facing a long period of renewals since a major part of the power stations are built during the 40s, 50s and 60s. Modernisation offers some potential for increased energy production - 0.5-2.0 TWh.

During the expansion of hydroelectric power there was extensive activity at various universities that acted as a support for manufacturers and purchasers of turbines. When expansion ceased during the 60s university activity also stopped. Major structural rationalisations have been made on the manufacturing side. Today the fully deregulated electricity market is pressing the power companies for even more precise prioritisation of renewals. The new conditions of the electricity market are also affecting manufacturing companies.

GE Energy (Sweden) AB and Waplans Mekaniska Verkstad are today the major turbine manufacturers in Sweden. In order to be able to exploit the opportunities presented by the renewal process and to meet the requirement for specialist skills that will be needed, despite the fact that the pace of new building and renovation is today very slow, the first phase of the programme was initiated and implemented in 1996-1999. The reasons for running the programme remain in every material particular and the interested parties have therefore decided to progress to a second phase.

1.2 Objectives

The strategy is to ensure the education of doctoral candidates in the field of water turbine engineering at technical universities in Sweden. This should be done in co-operation with industrial environments to develop skills in the field.

The overall aim of the programme is to contribute to further development of hydroelectric power as a renewable energy source. The programme should also work to make it possible to supply the power generating and manufacturing industries with skills in the water turbine field in the long term.

The target is for the programme to result in at least one licentiate and one doctorate in the field every two years and to aim for balance between examining industrial licentiates/doctorates and university doctoral students. At present nine doctoral candidates are being funded by the programme of which three are industrial. Up to now one doctorate and three licentiates have been awarded.

1.3 Areas of activity

When the programme was being planned there was relatively little research and development activity by Swedish generating and manufacturing industry in the water turbine field. Activities within the programme have manifestly increased these efforts. The following constitute the main features of the programme:

Experimental and computational fluid dynamics (CFD). Being able to determine the efficiency of a water turbine with high accuracy is important for both new investments and reinvestment. The profitability of reinvestments is often determined by gains in efficiency. The degree of efficiency is traditionally determined by model experiments by which the efficiency attained is scaled up by model rules. Greater accuracy can be achieved by using new computational models. An active association of computational development and experimental activity with modern methods of measurement, model tests and full-scale trials provides the opportunity for a sound industrial basis together with a high scientific standard.

Cavitation mechanisms in water turbines are of interest for both their design and operation. Knowledge of cavitation can be decisive in achieving an optimised design without the risk of erosion, vibration and noise. It is in many cases also of importance to be able to predict what risks of erosion will arise if the turbine works outside its ideal operating conditions. Knowledge of the risks of erosion in different operating conditions can be decisive for the total cost of operating and maintenance.

Tribology aimed at environmentally friendly lubricating oils and non-oil-lubricated bearings in turbine applications. Hydroelectric power stations have low emissions into the surrounding environment. In rare cases, however, oil leakage does occur. With smaller quantities of oil in the system and more environmentally friendly oils the risks to staff and the surrounding environment are reduced. Efforts are being concentrated on problems of how the lubricating properties of environmentally friendly oils (as substitutes for traditional mineral oils) are affected by conditions in the external environment - e.g. moist air.

1.4 Results

Results attained in *Experimental and computational fluid dynamics (CFD)* include the following:

- The use of modern measurement techniques such as Laser Doppler Velocimetry (LDV), velocity measurement using lasers, has been further developed.
- An experimentally and internationally applicable ‘benchmarking case’ for CFD in Kaplan-type turbine draft tubes has been developed.
- An international workshop, *Turbine 99 – workshop on draft tube flow*, was arranged in 1999 at which experimental data from the programme formed the basis for the comparison of the calculations of different international groups. The workshop has meant that the programme’s activities have been calibrated against the international leading edge in the field. At the *IAHR Symposium on Hydraulic Machinery and Systems* (2000) the workshop was quoted in a number of papers which indicates that an important goal was achieved - to contribute to increased collaboration in the development of better calculation methods.
- The numerical conditions for the calculation models have been established and the physical weaknesses that still remain in calculating the flow in a turbine draft tube have been identified.
- A powerful calculation tool has been developed for calculating the turbulent flow in water turbines whose complicated geometry requires special properties of the calculation tool. In particular they require many boundary layers and very fine resolution of certain parts of the area under calculation.
- Calculations have been carried out on a Kaplan-type turbine including leakage at the blade tips. The results agree well with observations of flow in Kaplan runners.

- Calculations on a Francis turbine (the GAMM turbine). The calculations were carried out with the same requirements for the resolution for the turbulent boundary layers as the Kaplan turbine. The results agree well with detailed documented measurements. The calculations were carried out in collaboration with IMH-IMHEF-EPFL in Lausanne.
- A numerical tool (finite element generator, solver and visualisation routines) that can be used to calculate the flow through small water turbines has been developed.
- It has been pointed out to the smaller turbine manufacturers that measurements that they have themselves carried out to good accuracy can be calculated by the programme and that parametric studies reproduce the tendencies measured in test rigs to quite a reasonable extent.
- The results are so promising that the smaller turbine manufacturers have expressed an interest in progressing to an implementation phase so as to use the tool as a complementary feature of the design and evaluation process.

The results obtained on *cavitation mechanisms* in water turbines include the following:

- Experiments carried out identify that the cavitating overflow vortex caused by leakage through the gap between the blade ends and the wall of the chamber exhibits a regular pattern. Since the vortex cavity ends up on the blade surface, it may very well be one of the causes of the erosion that can arise in that particular area. Likely reasons for the regular pattern could be vortices or wakes from guide vanes or supporting struts arriving at the impeller wheel.
- Experiments also show clear signs of reentrant jets in boundary layer cavities and cloud, bubble and vortex cavities breaking away from boundary layer cavities. Such cavities can collapse very violently close to the blade surface with consequent erosion damage. Focussing of reentrant jets may very possibly be one of the mechanisms that cause cavities to break away.
- In cavitation calculations it has been assumed that simulation of reentrant jets is important in order to be able to identify the risk of cloud cavitation and hence the risk of erosion. Continued experimental studies will provide a physical basis for the development of these methods.
-

As to *Tribology* aimed at environmentally friendly lubricating oils and non-oil lubricated bearings in turbine applications, results include putting into operation a working test rig in which the load and sliding speed can be controlled and temperature, wear and friction checked.

1.5 Co-operation and use of resources

It is true of all activities within the programme that practical verification of theoretical studies in experimental and pilot installations forms an important part. Because numerical simulation may be assumed to be a future tool in fluid mechanics the long-term work is aimed at developing appropriate numerical methods. The experimental studies provide a physical basis for development.

This was also one of the points under the magnifying glass during the international evaluation of the first phase of the programme. The evaluation has formed the starting point for the changes in the direction and organisation of the programme made before the second phase. This applies, for example, to:

- clearer prioritisation of activities that use existing experimental resources,
- special resources for network building and implementing periods of doctoral study abroad,
- programme funding for overheads for university co-operation and international co-operation of experts in reference groups and on the programme board,
- greater activity within the reference groups of the constituent projects,
- co-ordination of doctoral courses within the programme.

Clearer prioritisation of activities that use existing experimental resources

So far as the exploitation of experimental resources is concerned, this has manifestly been improved within the programme. In order to encourage the opportunities for synergistic effects and collaboration between the projects, extra resources have been allocated for co-ordination, joint planning and carrying out of joint experiments. The aim is for a common denominator between certain of the projects to drive collaboration between them as well between projects and industry.

This applies, for example, to the scale (1:11) model of the turbine at Hölleforsen power station. It is being used both in *Turbine 99 – workshop on draft tube flow* (see section 2.1 and 2.2), and for studies of transient effects in Kaplan-type water turbines originating in guide vane cavities (see section 2.3).

This will henceforth apply to a greater extent to Porjus U9 and its geometrically similar model turbine at Vattenfall Utveckling AB in Älvkarleby, which will be able to bring together several projects. This includes, for example, evaluation of model experiments for estimating losses (see section 2.3) and studies of scale effects on the course of cavitation (see section 2.4).

Experiments on cavitation have also been carried out at GE Energy (Sweden) AB in Kristinehamn (see section 2.4). Experiments relating to studies of loss mechanisms in the inlet part and spiral will also be carried out at GE Energy (see section 2.3).

Network building and implementing periods of doctoral study abroad

As an example of what is meant by network building and international cooperation, a doctoral student has spent three months with Prof. Francois Avellan at IMH-IMHEF-EPFL in Lausanne Switzerland as a consequence of contacts made during the evaluation. Another example is Prof. Hermod Brekke's (Norwegian University of Science and Technology) co-operation on the programme board.

The basis for broad international co-operation has also been laid by the international workshop. As a result of the exposure gained from *Turbine 99 – workshop on draft tube flow* an international network of contacts has been developed and will be further strengthened at a follow up to the workshop which is being held in June 2001 in Älvkarleby. This has been allocated a number of ERCOFTAC scholarships and has been endorsed by IAHR.

An important part of the programme is collaboration with industrial partners and between the doctoral students on the programme. Besides common denominators, in the form of experimental resources for certain projects, close collaboration between two of them is being promoted by collaboration between industrial and university doctoral students. This has proved successful in terms of transferring results and industrial contacts in a natural way. In the second phase, the industrial tie up is being strengthened by GE Energy and Waplans Mekaniska Verkstad participating each with their own industrial doctoral student.

Activities within the reference groups of the constituent projects

A programme seminar at which the doctoral students presented their projects and the results achieved was held in Porjus at the beginning of June 1999. The next programme seminar will be run in 2001. In order to expand the scope of contacts and increase activity within the project's reference groups, the opportunity was also taken during 2000 to coordinate reference group meetings. The results of this were favourable and the initiative in future will alternate between the parties carrying out the work.

Co-ordination of doctoral courses within the programme

So far as doctoral courses are concerned, Professor Håkan Gustavsson of Luleå University of Technology has been put in charge. The objective of joint doctoral courses is to reinforce the basis for co-operation in projects and the tie up between experimental and theoretical methods.

During summer 2000 a doctoral course *Experimental methods in fluid dynamics* was held at Vattenfall Utveckling AB in Älvkarleby and in November 2000 a general turbine engineering course, *Hydro Power Engineering,* was held in Porjus. Both had a good attendance of doctoral students on the programme and have contributed to increasing the common knowledge base and further developing the network of contacts between projects. The courses have also led to wider exploitation of facilities and equipment available nationally.

Experimental methods in fluid dynamics provided a general presentation of experimental methods with specific examples of physical factors and measurement techniques in fluid dynamics. The course focused on measurement in incompressible flows and raised issues of visualisation, pressure measurement, flow measurement, Laser Doppler Velocimetry (LDV) and Particle Image Velocimetry (PIV). It also dealt with experimental planning, model experiments and data collection/evaluation. Three pieces of laboratory work on visualisation methods, calibration and LDV measurement were included. The course was designed and organised by Urban Andersson and Rolf Karlsson of Vattenfall Utveckling AB.

Hydro Power Engineering dealt with subjects including pressure measurements, different types of loss (flow, bearing and ventilation losses), vibration and noise together with cooling. The course consisted of three elements - study visits (Jokkmokkskolan, Harsprånget and Porjus), theory and measurement. These aimed to put basic knowledge of hydroelectric engineering into a practical form and to use measurement techniques studied in *Experimental methods in fluid dynamics*. The course was designed and organised by Michel Cervantes of Luleå University of Technology.

2 STATUS REPORTS FOR THE RESEARCH PROJECTS

2.1 Turbin 99 – Angle resolved experimental investigation of flow field in a draft tube with kaplan runner - Status report October 2000

Scientific supervisor:
Prof. Rolf Karlsson
Vattenfall Utveckling AB
S - 814 26 ÄLVKARLEBY &
Royal Institute of Technology
FaxénLaboratoriet
S – 100 44 STOCKHOLM

Project leader:
Lic.Eng. Niklas Dahlbäck
Vattenfall Utveckling AB
S - 814 26 ÄLVKARLEBY
niklas.dahlback@
utveckling.vattenfall.se

Industrial doctoral student:
Lic.Eng. Urban Andersson
Vattenfall Utveckling AB
S - 814 26 ÄLVKARLEBY
uban.andersson@
utveckling.vattenfall.se

Achieved and planned academic exams

A thesis for the degree of Licentiate Engineering was presented in March 2000 [5]. A thesis for the degree of Doctor of Philosophy will be defended in the autumn of 2001.

Background

The strategic choice of project activities is based on the following:

- The economical benefits of redesign in medium head hydro power stations is often better at high loads than at best efficiency. In Sweden this implies extra interest in draft tube loss mechanisms.

- Modern technologies, such as LDA (Laser Doppler Anemometry) and CDV, are crucial tools for development in this field. They also give a high technology touch to an old technology field.

- Good collaboration between experimental activities at industry partners and theoretical activities at universities give possibilities to industrial relevance together with a high research level.

- Results from the field may also be interesting for other applications in other industrial branches.

- Good experimental results presented in an international forum give a base for international competence exchange.

Project description

The project contributes to the overall goal of the hydraulic turbine programme by increasing the base for development of competence in hydraulic turbine technology. The goal for this project is to perform an experimental test programme to create an internationally used benchmark test for CFD simulation of Kaplan turbines and draft tubes and develop experimental techniques used in turbine testing.

The benchmark test will be presented at two workshops with international participation. The first workshop was held at Porjus Hydropower Centre in June 1999. The second workshop will be held at the Älvkarleby laboratory in June 2001. The experimental data form this project is used as a base for comparisons of simulations performed of several international CFD groups.

Up to the first workshop, three different test periods were performed in the fully equipped experimental turbine test rig at Vattenfall Utveckling in Älvkarleby. The test periods determined relevant boundary condition data (inflow velocity distributions together with turbulence data and outlet pressure conditions) and data describing the draft tube flow behaviour (velocities and pressures) as a base for evaluation of CFD simulations.

Results from the project has been documented in workshop proceedings [2],[3],[4]. Results from the project together with conclusions from the workshops has also been presented at IAHR symposium of Hydraulic Machinery 1998 and 2000 [1], [7].

Figur 1: The tests were performed in a 1:11 model of Hölleforsen.

At the IAHR symposium presentation 2000, several comments on needs of more experimental data was expressed. This project seemd to be the only one of its kind open for everyone.

The last phase of the experiments will focus on:

- Additional LDA measurements and detailed wall pressure measurements to further improve the databank for the benchmark case
- Additional visualisations in the draft tube bend where there was found large discrepancies between different simulations in the first workshop.
- Angle resolved measurements, i.e. investigate the velocity fields in rotational direction.

In collaboration with this project one diploma work has been performed and one is planned. The first, by Tomas Lundvall, did investigated the quality and dynamic response of a pressure measuring system [6]. The second, by Staffan Jonsén, will contribute with a CFD simulation to the second Turbine 99 workshop.

International and industrial collaboration

This project and two other projects headed by Luleå University are contributing to the same goal. This means that there has been a close co-operation between the Division of Fluid Mechanics, Luleå University of Technology and Vattenfall Utveckling AB throughout the whole project. Frequent meetings have been held to plan future work, learn from each other and encourage and inspire new ideas. The Doctoral students involved are John Bergström , Urban Andersson and Michel Cervantes.

Data from the first experiments have been discussed with GE Energy (Sweden) AB, and common meeting with several of the other projects within the program has been held. In Spring 2000 a special course [8] in Experimental Hydraulics was arranged in Älvkarleby for all doctoral students within the Turbine Programme. This was a highly appreciated activity.

The international co-operation has started of well due to the first and the up-comming international workshops.

Status

The first goals in the project; the International workshop (*Turbine 99)* and a licentiate thesis have been achieved. The experimental work will be further developed through a test period before the second Tubine 99 workshop in June 2001. During this test period mesurements in cooperation with the project *Numerical investigation of turbulent flow in Kaplan water turbines* (see chapter 2.3) will also be performed.

The time schedule until summer 2002 also comprise preparation of contributions to the workshop proceedings, analysis and documentation, and the final exams in prior to the presentation of the thesis for Ph.D. which will be ready in autumn of 2001.

Scientific results

1. Andersson U. and N. Dahlbäck (1998) Experimental evaluation of draft tube flow. – A test case for CFD-simulations In*: Proceedings of the XIX IAHR Symposium on Hydraulic Machinery and Cavitation, vol1,* 268-275.
2. Andersson U. and R. Karlsson (1999) Quality aspects of the Turbine 99 draft tube experiments In*: Proceedings from Turbine 99 – workshop on draft tube flow*, Technical report, Luleå University of Technology, Sweden
3. Andersson U. (1999) Turbine 99 – Experiments On Draft Tube Flow (Test Case T) In*: Proceedings from Turbine 99 – workshop on draft tube flow*, Technical report, Luleå University of Technology, Sweden
4. Gebart B.R., Gustavsson L.H. and Karlsson R.I. (2000) *Proceedings of Turbine-99 – Workshop on draft tube flow in Porju*s, Sweden, 20-23 June 1999, Technical report from Luleå University of Technology, Luleå, Sweden
5. Andersson U. (2000) *An experimental study of the flow in a Sharp-Heel Draft Tube*, Licentiate Thesis 2000:08, Luleå Unversity of Technology, Sweden
6. Lundvall T. (2000) *Dynamisk respons vid tryckmätningar i modellförsök med vätskeströmning*, Examensarbete 2000:22 HIP, Luleå Unversity of Technology, Sweden
7. Gebart R., H. Gustavsson and R. Karlsson (2000) Report from Turbine-99 – Workshop on draft tube flow in Porjus, Sweden, 20-23 June 1999 In*: Proceedings of the XX IAHR Symposium on Hydraulic Machinery and Systems, CFD-F10*. Technical Report 2000:11, Lueå tekniska universitet.
8. Andersson U. and R. Karlsson (2000) Experimentella metoder inom strömningsläran, Kurspärm, Vattenfall Utveckling AB, Sweden

2.2 Turbin 99 – Follow-up, experimental study of scale-up formula for Kaplan turbines and inlet condition to the runner - Status report October 2000

Scientific supervisor/Project leader:
Prof. Håkan Gustavsson
Luleå University of Technology
Dept Fluid Dynamics
S - 971 87 LULEÅ
gustav@mt.luth.se

Industrial doctoral student:
M.Sc Sebastian Videhult
GE Energy (Sweden) AB
Box 1005
S - 681 29 KRISTINEHAMN
sebastian.videhult@ps.ge.com

Doctoral student:
Lic.Eng. Michel Cervantes
Luleå University of Technology
Dept Fluid Dynamics
S - 971 87 LULEÅ
michel.cervantes@mt.luth.se

Achieved and planned academic exams

During the first phase of the project, John Bergström from Luleå University of Technology obtained his degree of Licentiate of Engineering in March 1998 and the degree of Doctor of Philosophy in March 2000 [1]. Urban Andersson, Vattenfall Utveckiling AB, obtained his degree of Licentiate of Engineering in March 2000, see section 2.1. In the present phase, Michel Cervantes is expected to obtain his PhD degree and Sebastian Videhult his licentiate degree in October 2002.

Background

The purpose of the project is to develop computational fluid dynamics tools and methods to improve the effifiency in hydro turbines. At the beginning of the first phase of the project the goal was set to predict the effifiency in a complete hydro turbine with less than 2% discrepancy.

The simulation of the draft tube flows has been investigated in Bergströms thesis through calculations of increasingly complex test cases. Initially, simple test cases were studied where reliable experiments were available. The results show that the simple turbulence models commonly used in engineering give relatively large discrepancies compared to experiments.

At the end of the project, the flow of the Turbine-99 draft tube test case was studied, cf. also 2.1, with unsatisfying results at the beginning. The discrepancy compared to the experiments was coupled to the space discretisation, i.e. the grid quality. An optimised grid with approximately 700 000 cells coupled to a Reynolds Stress Model (RSM) predicted satisfactorily the pressure recovery, a representative engineering quantity of the draft tube.

The requirement on the grid fineness when optimising a whole turbine system implies that a simplified approach is still needed, where each part constituting the turbine system is treated separately. As a consequence, a simplified model of the flow trough the runner has been developed in the first stage of the project.

A separate study was done with a 'Very Large Eddy Model' based on a transient Reynolds stress model. Very good agreement with experiments was obtained and the low frequency component of the flow could be studied at the same time. The flow separation at the outlet of the draft tube showed a great influence on the result. The method based on Reynolds averaged (VLES) for transient calculations seems to have a promising future as the computer capacity increases. Such a method will certainly become a standard tool for engineers.

Bergströms and others results show that CFD is already a powerful tool to predict relevant engineering quantities for some part of the turbine system and soon will be the technique used to quickly determine the overall characteristics of a turbine. The actual discrepancy, between model and experiment, lies mainly in the description of the flow close to the wall and the influence of instationary effects. More work is necessary in this area to get a reliable description.

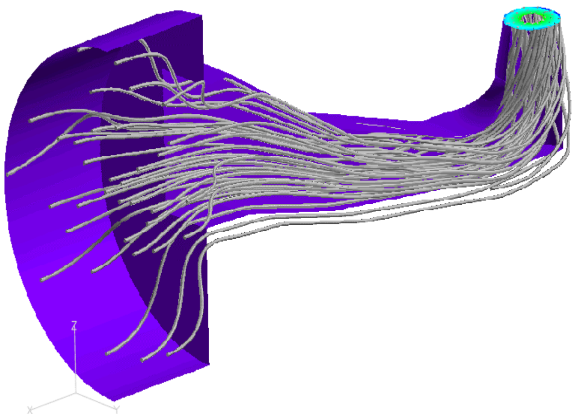


Figure 2: Streamlines in the Turbine-99 draft tube. Simulation done by J. Bergström, Fluid Mechanics, Luleå University of Technology

Improved models is of special interest for testing. Tests of runners have been reduced considerably. but more work is still necessary in the prediction of flow losses. This is the motivation of a part in the second stage of the project where unique opportunities are offered at Porjus and Älvkarleby to compare full-scale and model tests.

One important part in the first phase was the coordination and planning of the Workshop, Turbine-99 workshop on draft tube flow, in Porjus, and the compilation of the different contributions. The technical report of the workshop is now available as a Technical Report from Luleå University of Technology [2]. The positive response to the workshop indicates the achievement of the goal to contribute the increase cooperation in the development of turbine simulations.

The results of the workshop also helped in the planning of the second part of the project. Among other things, the results of some contributions have shown an unexpected distribution of the pressure at the inlet of the draft tube, which depends strongly on the inlet radial velocity. The investigation of the phenomena will be part of the follow-up part in the new project.

The results from the work shop also enables specifications of the requirements on the simulations of other parts of the turbine with better precision. This is the case of the inlet flow to the runner, which shows significant non-uniformities. A modified design may certainly improve the runner conditions. In summary, the following themes have been identified as central for continuation of the project:

- Follow-up of Turbine-99 with studies on the pressure at the inlet of the draft tube and the influence of the radial velocity distribution at the inlet of the draft tube.
- Scale-up formula for the flow close to wall, especially in the draft tube.
- Inlet flow to the runner.

Project description

The second part of the project consists mainly of the three themes named above. In addition, time will be required in the preparation of the second workshop. A separate simulation will be submitted to the workshop.

1) Turbin-99 follow up

One of the problems discussed in detail during the workshop was the pressure distribution at the inlet of the draft tube. The pressure recovery is calculated with the pressure on the wall, and errors give rise to a direct error in the global result. Among others, Skåre et al. [2] observed a low-pressure area in the calculated pressure profile, which is in contradiction with the analytical solution in a standard diffuser. In order to clarify the discrepancy, a test case is studied with the help of the Squire-Long equation, which describes swirl flow in a contraction. The equation takes care of the rotation that the flow has at the inlet of the draft tube and describes the complex interactions between the different forces acting in the system. Special interest is given to the influence of the radial velocity on the flow and the pressure distribution. The result may hopefully give some insights into the results obtained by some participants at the Turbine-99 workshop.

2) Scale-up formula

One of the important advantages with reliable simulations is that model testing can be substantially reduced and optimisation can easily be predicted. One of the important problems with model test is to scale-up the data from the model to the prototype. Several formulas have been proposed since the beginning of the century, e.g. Camere (1909), Moody (1925), Hutton (1954), JSME (1989) and IEC 60193 (1999). The international standard IEC 60193 is actually the most common in Europe.

Despite the evolution, discrepancies still exist between the predicted and the measured efficiency on prototype. The discrepancies are the consequences of both the scale-up formula and the measurement conditions on prototype. For example, the scale-up formulae of the IEC 60193 is based on the dependence of friction losses on Reynolds number where the different constants appearing have been experimentally determined with the hypothesis of smooth walls.

The object of the work is to investigate the different fluid processes responsible for losses in a hydropower plant such as wall processes, kinetic losses and transient phenomena in order to improve the scale-up formula. An important instrument to achieve thus goal will be the unique conditions offered by the Porjus Hydropower Centre (PHC) and its research prototype, combined with Vattenfall Utveckling AB (VUAB), Älvkarleby, where the corresponding model can be run.

3) Inlet flow to the runner

Whereas most of the research on turbines has been focused on the runner and later also on the draft tube, little work has been done in the spiral and the inlet flow to the runner.

In order to achieve an overall optimisation of the system, the inlet flow to the runner and the spiral need to be treated with the same accuracy as the other components. These components are interesting in several aspects. Previous experiments show that energy is lost in these components and identification of flow losses is of importance for an optimisation.

The spiral is designed to distribute a regular flow to the runner. Previous work show that the flow leaving the spiral is irregular both vertically and azimuthally and further distorted by stayrings and guide vanes. The irregular flow distribution most likely obstructs an optimal extraction of mechanical energy. Therefore, it is of interest to find the reasons of the irregularities in order to make it more regular. This work is made at GE Energy and involves both simulations and experiments.

Status

1. In the continuation of Turbine-99, the Squire-Long equation has been studied with the experimental boundary conditions, but with different radial velocity at the inlet of the draft tube. These calculations give the pressure distribution at the inlet of the draft tube, which gives the conditions for separation, especially on the cone. The results are expected to be reported at the beginning of 2001.

2. For the part on the scale-up laws, a literature study has been done, where the difficulties to clarify the fluid processes for scalable respectively non-scalable losses are evident. For the experimental part, the possibility and requirements material for Laser Doppler Velocimetry (LDV) measurements at Porjus have been investigated. A calibrating box has been designed and a measurement system purchased. Transient pressure measurements will be performed during spring 2001.

3. The study of the inlet flow to the runner and the spiral will be made with the help of computer simulation (CFD) and experimental verifications. The experiments will be made at GE Energy in Kristinehamn as well as in Ävkarleby. The computer simulation will be done at GE Energy and Luleå Technical University.

4. One of the most important conclusions of the turbine-99 workshop is that grid quality and small difference in the boundary conditions have a crucial influence on the computational results. Boundary conditions and grid have to be precisely specified to evaluate reliably the different turbulence models. This is the starting point for the next workshop, planned in June 2001, where the second case (R case) will be simulated. Measurements are presently on going at Vattenfall Utveckling AB. The call for the second workshop can be reached at www.luth.se/depts/mt/strl/turbine99.

5. In the framework of the Hydro Turbine Research Program, a general course in turbine technique was held in Porjus during November 2000. Several PhD students of the program participated. The course focused on pressure measurements, different form of losses (fluids, bearing, ventilation), vibration, noise and surge. The course was designed and held by Michel Cervantes, Luleå University of Technology, and a report is expected in February 2001.

International and industrial collaboration

An international network has been developed through Turbine-99 work shop. The network will be further developed with the follow up workshop, in Älvkarleby in June 2001. A number of ERCOFTAC scholarships have awarded to the workshop and it has been sanctioned by IAHR. The scientific committee consists of Prof. Dr. K. Hanjalic, Delt University of Technology in Holland and Prof. V.C. Patel, Iowa Institute of Hydraulic Research (IIHR), USA.

The project involves a direct cooperation between Vattenfall Utveckling AB and Luleå University of Technology. The industrial coupling is strenghtened through the presence of a PhD candidate from GE Energy AB.

Scientific results

1. Bergström, J., 2000, "Modeling and Numerical Simulation of Hydro Power Flows", Thesis for degree of Doctor of Philosophy 2000:06, Luleå University of Technology.
2. Gebart B.R., Gustavsson L.H. och Karlsson R.I., 2000, 'Proceedings of Turbine-99, workshop on draft tube flow', Technical Report 2000:11, Luleå University of Technology.
3. Bergström, J.,"Time-phase Averaged Flow in Turbine Runners – An approximate Model for the Flow in Complete Turbine Systems", Presentation at the 3^{rd} Nordic ERCOFTAC Pilot Centre Meeting, Majvik, Finland, 26-28 August, 1998
4. Bergström, J., "Turbin 99", Presentation at Elforsk Seminar, Älvkarleby, May 25, 1998
5. Bergström, J., "An approximate CFD method for the interaction between stationary and rotating parts in hydraulic turbines", Proceedings of the XIX IAHR Symposium, Section on Hydraulic Machinery and Cavitation, Singapore, 9-11 September, 1998
6. Bergström, J. and Gebart, B.R., "Estimation of Numerical Accuracy for the Flow Field in a Draft tube", Submitted to International Journal of Numerical Methods for Heat & Fluid Flow, 1998
7. Bergström, J., "Simulation of The Flow Field in a Hydro Power Plant", Poster presented at the Nordic Graduate Student Workshop in Computational Science and Engineering, Linköping University, September 29- October 1, 1997
8. Gebart, B.R., "Errors from near wall treatment in computation of complex flows", Presentation at the ERCOFTAC Nordic Pilot Centre Meeting 4-5 September 1997, Ebeltoft, 1997
9. Bergström, J., "Turbulence modelling and numerical accuracy for the simulation of the flow field in a curved channel", 1997 ASME Fluids Engineering Division Summer Meeting FEDSM'97 June 22 - 26, 1997, Vancouver , Canada, 1997
10. Bergström, J., "Numerical accuracy and turbulence modelling in fluid dynamics simulations of hydro turbine systems", First Swedish CFX User Group Meeting, Royal Institute of Technology (Stockholm), June 4, 1996
11. Bergström, J., "Turbulence Modelling and Numerical Accuracy for CFD", Svenska Mekanikdagar 1997, Luleå, 16-18 mars, 1997
12. Bergström, J., "Numerical accuracy and turbulence modelling in fluid dynamics simulations of hydro power turbine systems", Seminar on Numerical Methods for Fluid Flow, April 30, 1996, CERN (Geneva), Switzerland, 1996
13. Bergström, J., "Developing flow in a curved rectangular duct (case 4.5) Comparison of Numerical predictions", Proceedings of the 5th ERCOFTAC /IAHR Workshop on Refined Flow Modelling, Vol I and II, CHATOU (Paris), April 25-26, 1996
14. Bergström, J., "CFD-simulering av strömningen i sugröret till en Francisturbin", Svenska Mekanikdagar 1995, Lund, 31 maj-2 juni, 1995
15. Bergström, J., "Simulering av vattenturbiner, Beräkningar på strömningen i sugröret i en Francisturbin", Examensarbete 1995:027 E, ISSN 0349-6023, Avdelningen för Strömningslära, Tekniska Högskolan i Luleå, 1995

2.3 Numerical investigation of turbulent flow in Kaplan water turbines - Status report October 2000

Scientific supervisor/Project leader:
Prof. Lars Davidsson
Chalmers University of Technology
Thermo and Fluid Dynamics
S - 412 96 GOTHENBURG
lada@tfd.chalmers.se

Doctoral student:
Lic.Eng. Håkan Nilsson
Chalmers University of Technology
Thermo and Fluid Dynamics
S – 412 96 GOTHENBURG
Hani@tfd.chalmers.se

Achieved and planned academic exams

The project started in spring 1997. A thesis for the degree of Licentiate of Engineering was presented in September 1999. A thesis for the degree of Doctor of Philosophy will be defended in the spring of 2002.

Background

The turbulent flow in the guide vanes and along the runner blades is of great importance for the performance of water turbines. Detailed analysis of the flow is needed in order to be able to study, and maybe improve, the flow conditions. We believe CFD, as a complement to experiments, is an appropriate tool to achieve this object.

In order to resolve the turbulent flow in the tip clearance (Kaplan turbines) and in the boundary layers, a low Reynolds number turbulence model must be used. Because of computational restrictions, most water turbine simulations usually use wall functions instead of resolving the boundary layers, which makes tip clearance investigations impossible.

Although the computations assume that the flow is periodic, which allows only one blade passage to be computed, and that the boundary conditions are assumed to be axisymmetric, these kinds of computations tend to become computationally heavy and numerically challenging. This, together with the complicated geometry requiring complex multiblock topologies, makes a parallel multiblock CFD solver a suitable tool.

Project description

A parallel multiblock finite volume CFD (Computational Fluid Dynamics) code CALC-PMB (Parallel MultiBlock) for computations of turbulent flow in complex domains has been developed and used for the computations of the flow through water turbines. The main features of the CALC-PMB CFD code are the use of conformal block structured boundary fitted coordinates, a pressure correction scheme (SIMPLEC), cartesian velocity components as the principal unknowns, and collocated grid arrangement together with Rhie and Chow interpolation.

The computational blocks are solved in parallel with Dirichlet-Dirichlet coupling using PVM (Parallel Virtual Machine) or MPI (Message Passing Interface). During the computations, the computational blocks are assigned to separate PVM or MPI processes. The level of parallelization is thus determined by the block size distribution and the distribution of the processes on the available processors. The parallel efficiency is excellent, with super scalar speedup for load balanced applications.

Two types of water turbine runners are investigated:

Kaplan water turbines:

The first part of this work is focused on tip clearance losses in Kaplan water turbines, which reduces the efficiency of the turbine by about 0.5%. The investigated turbine is a test rig, at GE Energy (Sweden) AB, with a runner diameter of 0.5m. It has four runner blades and 24 guide vanes. The tip clearance between the runner blades and the shroud is 0.25mm. These computations are performed in two steps. First the stationary guide vane flow is computed. Then the circumferentially averaged velocity field from the guide vane computations are applied as an inlet boundary condition for the runner computations. Four operating conditions with increasing blade loading are compared, with focus on tip clearance effects.

The second part of this work will focus on transient effects in Kaplan water turbines due to guide vane wakes. The investigated turbine is a 1:11 model of the Hölleforsen turbine, also used in the Turbine 99 workshop. These computations will be performed in two steps. First the stationary guide vane flow is computed. Then the periodic velocity field from the guide vane computations are applied as an inlet boundary condition for the runner computations.

Francis water turbines:

This part of the work investigates the turbulent flow in a Francis water turbine. The GAMM Francis turbine (reference calculation case at the GAMM workshop on 3D-computation of incompressible internal flows was held at EPFL in Lausanne in 1989) is used for validation of the CALC-PMB CFD code in the area of hydraulic machinery. The investigated Francis runner has 13 runner blades. Detailed measurements of the flow in this turbine was performed in conjunction with the GAMM Workshop in 1989. This validation is performed in a collaboration with LMH - IMHEF - EPFL in Lausanne, Switzerland, where three months of the work was carried out. For this runner, the design point and four off-design points are computed and compared with measurements. At the runner inlet, the measured circumferentially averaged inlet velocity profiles are extrapolated using potential flow assumption. This removes some (most) of the axisymmetric inlet velocity profile assumption.

The computational results are in accordance with observations made by GE Energy (Sweden) AB and the GAMM measurements. Some results are presented below as figures with captions. Download the references presented below for a more thorough investigation and discussion of the results.

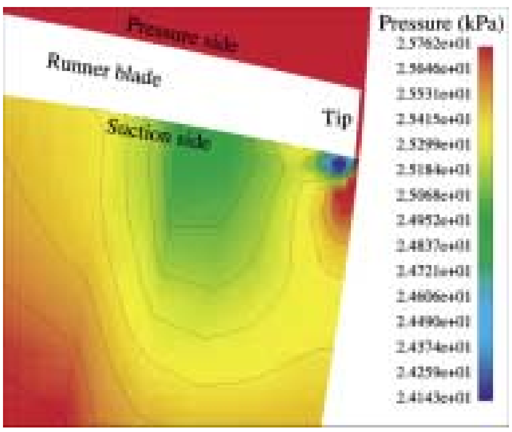


Figure 1: The static pressure distribution on a meridional cut through the center of the Kaplan runner blade. A local static pressure reduction is observed very close to the tip on the suction side of the blade. A local static pressure increase is observed where the tip clearance jet scrapes off the shroud boundary layer. The level of the pressure is not ajusted to a real case.

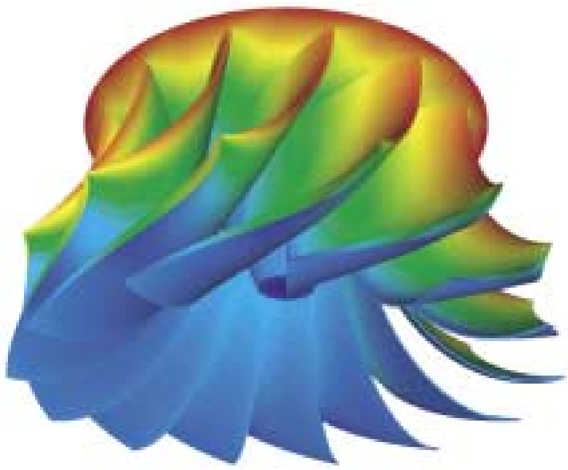


Figure 2: Surface static pressure distribution on the GAMM Francis runner.

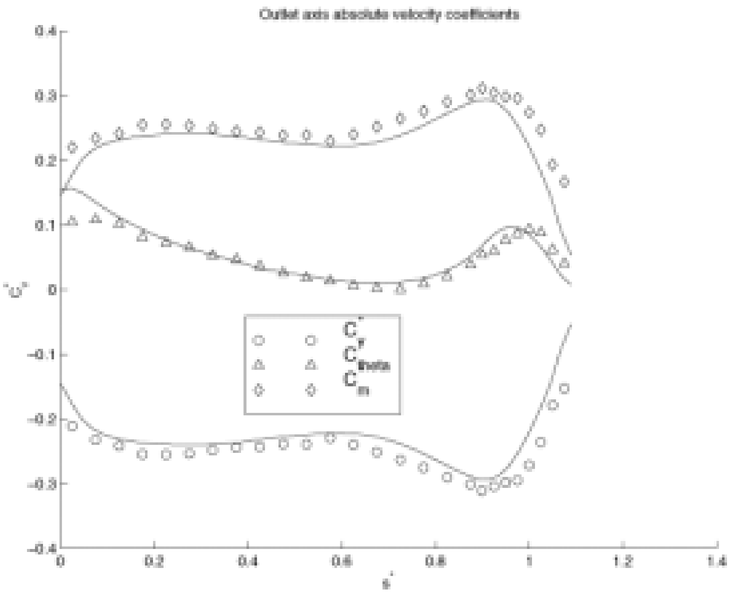


Figure 3: Outlet axis absolute velocity coefficients of the GAMM runner, preliminary computations vs. measurements. (x-axis: Shroud (0) to rotational axis normalized abscissa, y-axis: absolute velocity coefficient)

International and industrial collaboration

The work is carried out in cooperation with GE Energy (Sweden) AB. They have supplied the geometry of the Kaplan water turbines including guide vanes and runner blades. They have also contributed with their knowledge of the flow in Kaplan turbines.

Part of the work is carried out in cooperation with LMH - IMHEF - EPFL in Lausanne, Switzerland, where three months of the work was carried out. They have supplied the geometry and performed the measurements of the GAMM Francis turbine.

To analyse the results from the computations, a collaboration with a VR CAVE postprocessor developer at CERCA, is established.

Status

A thesis for the degree of Licentiate of Engineering was presented the 20:th of September 1999. A thesis for the degree of Doctor of Philosophy will be defended in the spring of 2002.

An efficient parallel multiblock finite volume CFD (Computational Fluid Dynamics) code CALC-PMB (Parallel MultiBlock) for computations of turbulent flow in complex domains has been developed and used for the computations of the flow through water turbines. The validation of the code for these applications is satisfactory.

The computations of the stationary Kaplan tip clearance computations and the validation of the code against the GAMM Francis runner are being concluded.

So far in this work, the upstream effects from the runner on the guide vane flow and the transient effects of non-axisymmetric runner inlet boundary conditions are excluded. In particular, the non-axisymmetric runner inlet boundary conditions are expected to affect the dynamics of the tip vortex because of the varying angles of incidence. In the future, a more advanced coupling between the rotating and stationary parts together with transient computations will be used to include the transient effects of guide vane wakes, varying angles of incidence and the instabilities of the tip vortex.

References / Scientific results

1. * H. Nilsson & L. Davidson: An implementation of a parallel multiblock extension to the CALC-BFC code using PVM for simulating the flow over sedimentary furrows, Diploma Thesis. Rept. 97/3, Thermo and Fluid Dynamics, Chalmers University of Technology, Gothenburg, 1997
2. * H. Nilsson & L.Davidson: A parallel multiblock extension to the CALC-BFC code using PVM. Internal Report Nr 97/11, Thermo and Fluid Dynamics, Chalmers University of Technology, Gothenburg, 1997
3. * H. Nilsson & L.Davidson: CALC-PVM: A Parallel SIMPLEC Multiblock Solver for Turbulent Flow in Complex Domains. Internal Report 98/12, Thermo and Fluid Dynamics, Chalmers University of Technology, Gothenburg, 1998
4. S.Dahlström, H. Nilsson & L. Davidson: LESFOIL: 7-Months Progress Report by Chalmers, Technical report, Thermo and Fluid Dynamics, Chalmers University of Technology, Gothenburg, 1998. Confidential.
5. * H. Nilsson: A numerical investigation of the turbulent flow in a Kaplan water turbine runner. Thesis for the degree of Licentiate of Engineering 99/5. ISSN 1101-9972.
6. * H. Nilsson and L. Davidson: A numerical investigation of tip clearance flow in Kaplan water turbines. Proceedings of HYDROPOWER INTO THE NEXT CENTURY - III, 1999, p.327-335. ISBN 0 9522642 9 3
7. * H. Nilsson and L. Davidson: A Numerical Comparison of Four Operating Conditions in a Kaplan Water Turbine, Focusing on Tip Clearance Flow. Published in the proceedings of the 20th IAHR Symposium, August 6-9 2000, Charlotte, North Carolina, U.S.A.
8. H. Nilsson, S. Dahlström and L. Davidson: Parallel Multiblock CFD Computations Applied to Industrial Cases. To be published in the proceedings of the conference Parallel CFD 2000, May 2000, Trondheim.

(* = download from http://www.tfd.chalmers.se/~hani)

2.4 Cavitation in hydraulic turbines - flow fields and mechanisms - Status report October 2000

Scientific supervisor/Project leader:
Associate. Prof. Göran Bark
Chalmers University of Technology
Naval Architecture and Ocean Engineering
S – 412 92 GOTHENBURG
bark@na.chalmers.se

Doctoral student:
M.Sc. Mikael Grekula
Chalmers University of Technology
Dept. of Naval Architecture and Ocean Engineering
S – 412 92 GOTHENBURG
mga@na.chalmers.se

Achieved and planned academic exams

A thesis for the degree of Licentiate Engineering is planned for March 2001 and a thesis for the degree of Doctor of Philosophy will be defended in March 2003.

Background

At the design as well as at the operation of hydraulic turbines, one has to consider both efficiency and cavitation. For example the radial blade loading, which affects both the cavitation and the efficiency, is settled at the designing. Increased loading at the blade root sections may give an increased efficiency but it may also cause cavitation and erosion. At such a weighing of counteracting factors, a realistic simulation of the cavitation process can be conclusive for reaching an optimised design. It is also important to predict the erosion risk that may occur if the turbine is operated outside its design condition. A design, optimised for operation at varying conditions, will most often have a lower maximum efficiency. Prediction of cavitation processes takes thus a significant place in the optimisation of the total economy for a turbine.

Erosion caused by imploding cavities is often associated with cloud cavitation. This type of cavitation consists of a cloud of cavitation bubbles, which for instance can origin from a relatively thin sheet cavity, which has developed on a runner blade. The importance of the bubbles is that they will collapse under mutual interaction and cause powerful pressure pulses.

The processes, which cause more or less smooth sheet cavities to transform to cloud cavitation, are not fully known although to all appearances one of the most important mechanisms is the re-entrant jet, which travels in the upstream direction from the downstream closure end of the sheet cavity.

A phenomenon newly studied in a neighbouring project shows that a smooth sheet cavity may collapse very rapidly and result in erosion without any substantial preceding transformation towards cloud cavitation. In many of the studied cases, it was possible to find minor cloud formations at the end of the collapse process. These small cloud formations are possibly enough to cause the collapse process to have the collective behaviour, which usually is associated with cloud cavitation.

Cloud cavitation may also be generated in other configurations than sheet cavitation and there are other generation mechanisms than re-entrant jets. As an example may be mentioned that relatively harmless bubble clouds originating from a sheet cavity can be transported into a vortex where they collapse rapidly and cause at least strong noise or, if the vortex cavitation collapses on the blade, possibly also erosion. For the simulation part of this project, it is assumed that simulation of the re-entrant jet behaviour is important for the identification of the risk for cloud cavitation generation and thereby the erosion risk.

Project description

The ambition of the project is that it should lead to knowledge, which in long terms makes it possible to make more precise predictions of the risk for erosion, vibrations and noise caused by cavitation in hydraulic turbines. One important sub-task is to generate general knowledge in this field as a support for present experimental and design processes. Since assessment of the risk for erosion is made on scale models at the design of hydraulic turbines, it is important to be aware of how the cavitation processes are affected by the geometrical scale of the turbine. As a support for this, we have planned to do a study in two steps of the scale effects on cavitation processes. Porjus U9 and its geometrically uniform model offer this opportunity.

Since numerical simulation can be assumed to be a future tool for cavitation prediction, the long-term work is to study and develop sufficient numerical methods for that purpose. The experimental studies generate the physical basis for the development of these methods.

The numerical method, which presently is planned to be used in the project, can hopefully be the first step on the way towards a new generation of computational methods. With the simulation methods of today, which mainly are potential flow methods, can low frequency noise be predicted reasonably well although high frequency noise and erosion is out of reach.

It should be mentioned that it will probably within foreseeable time not be possible to predict the rate of eroded material with numerical methods. However, it is within reach to be able at the design stage to tell whether a certain design can be expected to generate strongly erosive cavitation or not.

This project consist of two main parts:

A. Experiment

The aim of the experiments is primarily to make a documentation of how cavitation processes develops towards rapidly collapsing cavitation and in particular to erosive cavitation. Among others it is important to note the influence of re-entrant jets and in which extent they appear.

Based on the experiences from the first experiments and the discussions with our reference group the following experiments have been planned:

- *Complementary experiments* to have a closer look at shedding from sheet cavitation (see figure 1). We would also like to find out why the cavitating tip vortex is modulated. The modulation tends to cause the tip vortex cavitation to collapse on the blade and thereby possibly cause erosion. The first experiments indicated that wakes or vortices originating from the guide vanes cause this modulation (see figure 2).
- *Scale study, part one,* in model scale will be carried out in a model turbine at Vattenfall Utveckling AB in Älvkarleby. It is a geometrically uniform model of Porjus U9. The plan is to make high-speed filming, video recording, and noise measurements.
- *Scale study, part two,* is a full-scale study at Porjus U9. Like in the first part, we plan to use high-speed filming, video, and noise measurements.

Figure 1: The photograph shows suction side cavitation on a Kaplan runner blade. The hub is seen to the left in the photograph. Its rotational axis is parallel with the left margin of the picture. The blade leading edge is the almost horizontal shiny band in the middle of the picture. The flow is from above. There are two sheet cavities at the blade leading edge. Cavitation bubbles, which often are enclosed in vortex cavities or appear in big bubble clouds, are constantly braking off from the small sheet cavity (the one to the left in the picture). It cannot be excluded that a re-entrant jet is one of the mechanisms behind this. The opaque part in the downstream end of the large sheet cavity (the one to the right in the picture) is a typical sign on a re-entrant jet. Also from this cavity are bubble clouds braking off (not visible in exactly this picture). At the blade root, close to the hub, a root cavity is seen which reveals possibly some weak rotation..

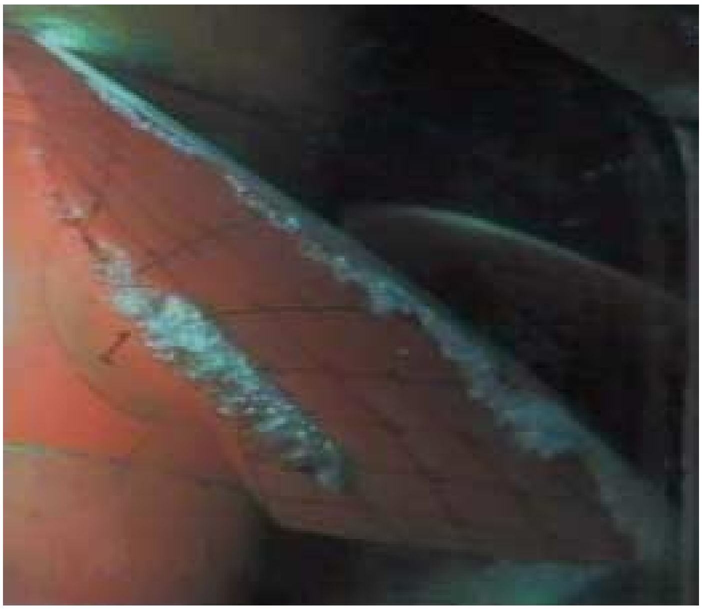


Figure 2: One of the runner blades is passing by the inspection window in the discharge ring. The turbine shaft is aligned vertically in this picture and the flow is from above. The leakage flow through the tip clearance rolls up in a vortex, which cavitates due to the low pressure in its core. After the vortex has formed a little bit downstream from the leading edge (up to the left in the photograph) it leaves the blade surface but for some reason it makes a turn and approaches the blade again. This course of events is repeated regularly over the whole chord of the blade. The modulation of the tip vortex is possibly caused by wakes and/or vortices originating from the guide vanes.

B. Numerical simulation

Still is lot of effort put into development of potential flow methods for cavitation simulation but the research front tends to be concentrated on Navier-Stokes methods. This should especially be relevant if simulation of re-entrant jets is regarded to be important.

Since we are interested in the unsteady behaviour, one of the main properties of the computer code must be that it is good at dealing with unsteady flows. Other important properties that must be managed are a model for cavitation generation, mass inertia of the surrounding water and re-entrant jets. The initial work will be limited to two-dimensional problems.

Even if the unsteady behaviour of the cavitation in an unsteady flow is simulated well the problem of how clod cavitation is generated may not have been solved although the simulation can be assumed to have come somewhat closer to the above given goal.

International and industrial collaboration

International cooperation within this project has not been planned in detail yet. However, the cavitation group has good international contacts and through the reference group are Knud-Aage Mörch and Hermod Brekke at NTNU involved in the project. Experiment has been carried out at GE Energy (Sweden) AB in Kristinehamn. At the planning of the next experiments, which consists of one complementary study and one study of scale effects on cavitation processes, discussions are conducted with Vattenfall Utveckling AB, GE Energy (Sweden) AB, Vattenfall Vattenkraft AB and the Department of Fluid Dynamics at Luleå University of Technology. Our selection of program for computations (FOAM) will lead to some contacts with FOA, department of Termo- and Fluid Dynamics at Chalmers and with the CFD-group of Professor Gossman at Imperial College of Science, Technology and Medicine in England.

Status

A literature survey has been carried out. The experiences from that survey concerning cavitation in general as well as cavitation in hydraulic turbines have been used as the base for the initial experiments. The conclusions from the literature survey are included in the report from the experiments [1].

A. Experiment

An initial experimental study has been made at GE Energy (Sweden) AB in Kristinehamn and is reported in [1]. Some cavitation processes needs to be studied closer. The plan is to do this in a complementary study, which will be coordinated with the first part of the study of scale effects. This second round of experiments will be coordinated with the velocity measurements that Michel Cervantes at Luleå University of Technology has planned to do in the scale model at Vattenfall Utveckling AB in Älvkarleby (see also section 2.2). This collaboration will not only save some work but also give the opportunity to relate the cavitation studies to velocity measurements. The second part of the scale study, which will be carried out in Porjus U9, is planned to be in the second part of this programme by an application of continuation during the first part of year 2001.

B. Numerical simulation

Based among others on the initial experiments we have selected to concentrate the work on prediction of sheet cavitation and the development of the re-entrant jet. There is cooperation with another project on the department in which tip vortex cavitation has to be simulated. The computational program FOAM (Field Operation and Manipulation C++ class library) is used in both these projects. Different methods will be tested for tracing the interface between a vapour cavity and the water. Cavitation simulation will first and foremost be calculated and verified for a two dimensional foil in a steady flow to be able to compare the results with the computational and experimental results of other research groups.

Scientific results

1. Grekula, M. Suction side cavitation in a Kaplan model turbine, review and experimental study. Tech. Rep. CHA/NAV/R-00/0070, Department of Naval Architecture and Ocean Engineering, Chalmers University of Technology Göteborg, 2000. ISSN 1101-0614.
2. Grekula, M., and Bark, G. Experimental study of cavitation in a kaplan model turbine. In Svenska Mekanikdagar 1999 (Stockholm, 7--9 jun 1999), KTH Högskoletryckeriet.

2.5 Transport of cavitation bubbles in hydraulic turbines - Status report October 2000

Scientific supervisor/Project leader:
Prof. Lazlo Fuchs
LundsIinstitute of Technology
Division of fluid mechanics
S - 221 00 LUND
laszlo.fuchs@.vok.lth.se

Industrial doctoral student:
M.Sc Henrik Lindsjö
Waplans Mekaniska Verkstad AB
S - 830 44 NÄLDEN
henrik.lindsjo@waplans.se

Doctoral student:
M.Sc Daniel Lörstad
LundsIinstitute of Technology
Division of fluid mechanics
S - 221 00 LUND
daniel.lorstad@vok.lth.se

Achieved and planned academic exams

Daniel Lörstad´s thesis for the degree of Licentiate of Engineering is planned to be presented in the end of spring 2001 and a thesis for the degree of Doctor of Philosophy will be defended in the spring of 2003. Henrik Lindsjö´s licentiate thesis is planned to be presented in the spring of 2002 and PhD in the spring 2004.

Background

The project aims at the development of a method to predict the transport of cavitation bubbles. Such a method will be a base for a tool to predict the bubble transport which may be used to increase the distance between the cavitation implosion points and sensitive parts of the turbine. It will also be possible to tell from where the erosive cavitation bubbles originates, which will lead to a possibility to deal also with the formation of the bubbles.

In order to obtain such a numerical tool, experimental data must be used for validation. Therefore the project contains both a numerical and an experimental study of the flow in a simplified hydraulic turbine. The numerical simulations are focused at investigating the bubble transport in turbulent flows in rotating turbines. The design of the experimental set-up has mainly been governed by the requirements set by the goal to evaluate the numerical methods. Of course, this implies a high accuracy in the measurements, but also that the boundary conditions (used as input in the numerical model) must be of high quality – both spatially and temporally. Hence a great effort is put into reducing the uncertainties and the sources of error in the comparison between numerical and experimental results. The results (including the full geometry) will be made available.

Project description

The project contains two main parts

1. The development of numerical models for bubble transport in turbulent flows in hydraulic turbines.
2. Validation of the numerical results

In the experimental part a study of the flow through a long, square channel in which a rotating turbine runner is placed will be conducted. Bubbles will be introduced upstream of the runner. The channel shape is quadratic in order to simplify the measurements (reducing problems due to refraction). The velocity field will be measured with Particle Image Velocimetry (PIV) and Particle Tracking Velocimetry (PTV). When both phases are studied simultaneously a PIV/PIV method will be used. As the project deals with the transport of cavitation bubbles (not the formation or the destruction), the cavitation bubbles will be replaced with air bubbles or bubbles created with electrolysis or ultrasonic sound, depending on the required bubble size. Different sizes of bubbles will be investigated. Figure 1 and 2 shows examples of measurements of bubbles conducted at Fluid Mechanics at LTH.

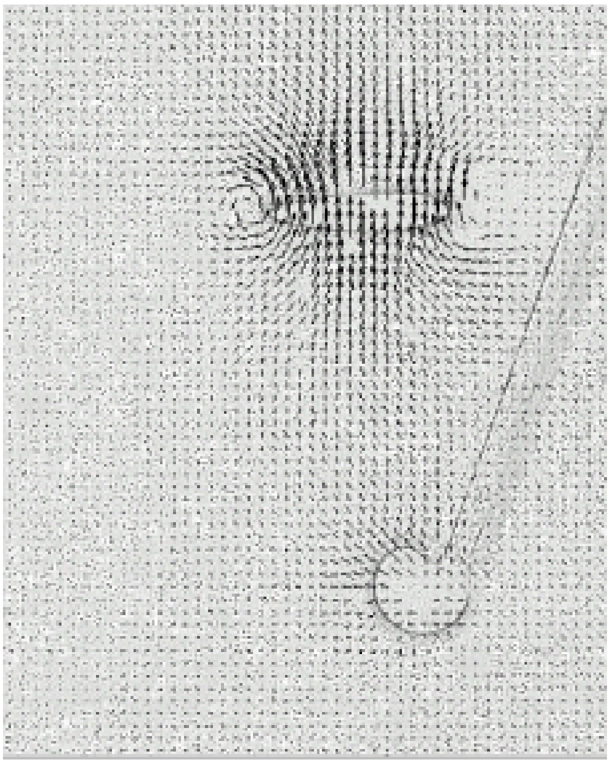


Figure 1: PIV (Particle Image Velocimetry) of the flow field around air bubbles (in water) ejected from the tube in the right part of the picture. An almost spherical air bubble is seen at the end of the tube and a deformed elliptical bubble is moving upwards.

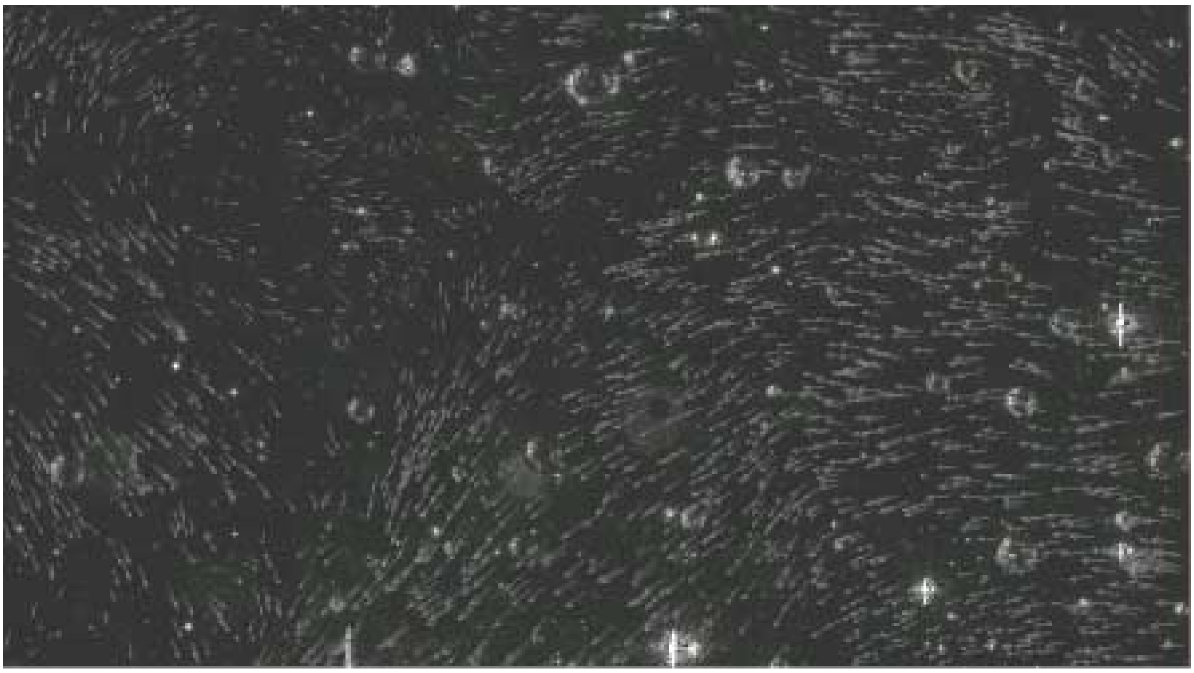


Figure 2: Air bubbles in water. The arrows indicate the velocity obtained with PTV (Particle Tracking Velocimetry) on fluorescence tracer particles (the particles are filtered from the picture).

To simulate the turbulent diffusion of bubbles (strongly dependent on the time dependent large scale structures in the flow) Large Eddy Simulation (LES) will be used.

International and industrial collaboration

Henrik Lindsjö and Daniel Lörstad are in contact almost on a daily basis and as the former is an ”industrial” doctoral student, based at Waplans, the industrial collaboration comes about naturally. Henrik also has close contacts with Herman Lindquist.

International collaboration is being built up with Prof. Wei Shyy, University of Florida, USA. Prof. Shyy has many years of experience from numerical applications alike those in the project and he is positive to collaboration. Daniel will visit Prof. Shyys department during a couple of month’s time during 2002.

Status

The numerical part will focus on:
1. Turbulence
2. Rotating turbine runner
3. Bubble transport

Daniel has mainly been working with the development of a numerical method that with higher accuracy deals with moving boundaries in Cartesian grids, wich can be used for rotating turbine runners. The model has been tested on simpler geometries in both laminar and turbulent flows and the results are promising. A Navier-Stokes solver for Cartesian grids has been chosen as it is fast, requires (relatively) little memory, uses high order discretisation (important for LES) and is applicable on all types of water turbine geometry’s. Daniel has also implemented models to simulate large and small bubbles. An article describing the model development regarding moving boundaries and large bubbles is being written and will be presented in a numerical journal.

Henrik has been working with the design and manufacture of the experimental set-up. The set-up is planned to be completed during autumn 2001. Henrik has also helped Daniel in the evaluation of Daniels numerical models by comparisons with Fluent, in the cases where Fluent has been applicable.

Daniel is at the moment working with the numerical description of the geometry defined by the experimental set-up while Henrik is working with the completion of the set-up. When that is ready they will together carry out the numerical computations and the measurements on the model. Firstly, evaluation without bubbles will be carried out, then with small bubbles and lastly with bubbles of different sizes.

Scientific results

1. Lörstad D., Fuchs L., LES of Turbulent Flow in a Draft Tube of a Water Turbine, 1999, Turbine 99, ERCOFTAC Workshop on Draft Tube Flow, Porjus Hydropower Center, Sweden.

2.6 Tribology in hydro turbines - maintenance and environmental adaptation - Status report October 2000

Scientific supervisor:
Prof. Erik Höglund
Luleå University of Technology
Div. of Machine Elements
S – 971 87 LULEÅ
erik.hoglund@mt.luth.se

Assistant project leader:
Sen. Lect., PhD, Elisabet Kassfeldt
Luleå University of Technology
Div. of Machine Elements
S - 971 87 LULEÅ
elisabet.kassfeldt@mt.luth.se

Doctoral student:
Jan Ukonsaari
Luleå University of Technology
Div. of Machine Elements
S - 971 87 LULEÅ
jan.ukonsaari@mt.luth.se

Achieved and planned academic exams

A thesis for the degree of Licentiate of Engineering is planned to be presented in januar 2002 and thesis for the degree of Doctor of Philosophy will be defended in januar 2004

Background

Mineral oils in water power plants bring environmental risks. A seal failure can cause oil leakage into water. The consequences of leakage are not the most conspicuous thing because the dilution from large amounts of water. In spite of this it's not satisfying that there is a risk of leakage and you must consider the sensibility of nature. Changing to environmentally adapted oils is given a strong motivation. Unfortunately is the knowledge about environmentally adapted oils limited in the aspect of performance and life endurance in this application.

There are many questions to be answered before the users are ready to change the oil type. For an example; how are the journal bearing- and component materials influenced? Do you have to change oil filter? Is the operation temperature altered? The problem concerns how a relevant demand specification should be formulated.

The project focuses on problems how lubrication ability of environmentally adapted lubricants is affected by conditions in the surroundings e.g. moisturised environment. Environmentally adapted synthetic esters and rape seed oil based lubricants tend to bind large amount of water. The limit of water contamination is today based on how mineral oils are affected and this is not comparable to environmentally adapted ones. Water in oil must be considered as pollution like particles and debris and that can affect viscosity and oxidation stability. These factors affect the lubrication ability and life. Corrosion is another factor to be considered.

Project description

The project starts by testing in controlled laboratory environment. A new test rig is designed to study different lubricants, bearing materials, loads, temperatures and sliding speeds in controlled environment. It's possible that moisture air and high pH-value will be considered. This question is initiated from ABB where the problem concerns extreme wear of seals.

The project will be carried on by a PhD student, senior researcher, technicians and a reference group.

The main aim in the project is to decide a relevant demand specification of environmentally adapted oils in water turbine application. The demand specification could consider properties that are decided from standard procedures, but also properties specific to this application.

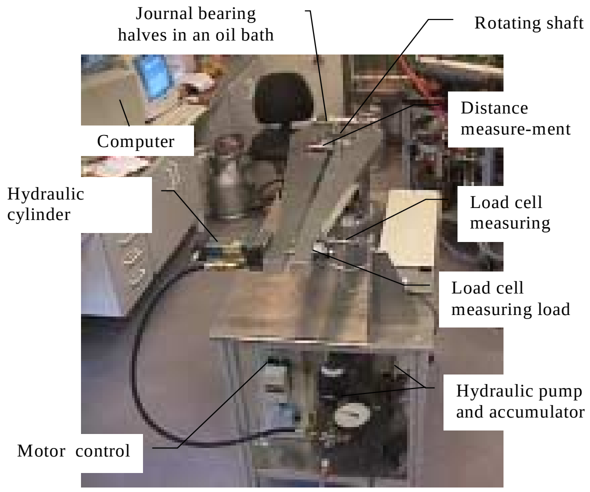
International and industrial collaboration

The Division of Machine Elements is constantly working together with other scientists at different universities, in Sweden as well as abroad. One objective within the PhD education is to give the possibility to study some time at a foreign company or university. This is also planned in this project.

National collaboration is done mostly with companies within the reference group, but also through collaboration in PhD courses.

Status

The work has been focused on developing a new test rig, designed to simulate the environment in a Kaplan turbine and its turbine blades support. The design of the test rig is in its final phase, see Figure 1.



From a number of ideas, constructions and drawings is now everything ready and small adjustments remain. During October 2000 is the first test results planned to be presented.

Figure 1: Test rig design

The test rig controls *temperature, wear, sliding speed, friction* and *load* of the two bearing halves placed in a fluid bath. Temperature, sliding speed and load are variable.

- *Temperature* is registered by thermo element in the journal bearing shells just below the sliding surface. One temperature gauge is for the oil temp in the bath. Cooling/heating of the oil bath/journal bearing is made by a system of vater flowing through the bearing housing and bath.
- *Wear during* running is recorded by a displacement meter measuring the decreasing distance between the two bearing shell halves. This gives no absolut value on wear but always an indication, see Figure 2.
- The *sliding speed* and movement pattern is made by the motor control.
- The *friction* between the journal bearing shells and the shaft gives a torque that is picked up by load cell 1. The measured force is translated into friction.
- The *load* is applied by a hydraulic cylinder which via the clamping arms squeezes the bearing shells to the shaft. The load is measured by load cell 2, see Figures 1 and 2.
- All parameters is controlled in a computer.

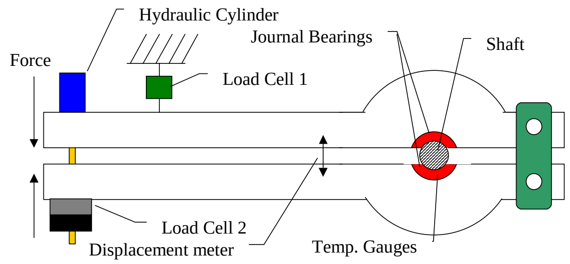


Figure 2: Measurement principals of the test rig.

Comparing studies are important to achieve knowledge about different test methods and the relevancy of them. Plint & Partner high frequency wear tester is a test apparatus where boundary conditions can be achieved. This apparatus is very good and fast for material and oil tests, and to study different wear mechanisms vs. friction. It is interesting to compare the new test rig with the Plint & Partner rig because it has a small contact area and the new rig relatively large. In this way we can learn more on edge effects in boundary lubrication.

A pre-study shows interesting results on wear mechanisms and friction [2]. Experience from these tests is more studies are made with other materials and oils.

Scientific results

1. M. Del Din and E. Kassfeldt ”Wear characteristics with mixed lubrication conditions in a full scale journal bearing” Proceedings of the 8th International Conferencs on Tribology, NORDTRIB ´98, June 7 - 10 1998, Aarhus, Denmark. WEAR 232 (1999) 192-198.
2. E. Kassfeldt, T. Norrby and J. Rieglert, ”Wear and friction in boundary lubricated contacts”, Tribology in Environmental Design 2000, Bournemouth, United Kingdom, 4-6 september 2000.

2.7 Development of a user-friendly CFD (computational fluid dynamics) system for analysis of small axial flow hydraulic turbines - Status report October 2000

Scientific supervisor/Project leader:
Prof. Torsten H. Fransson
Royal Institute of Technology
Department of Energy Technology
Chair of Heat and Power Technology
S - 100 44 STOCKHOLM
fransson@egi.kth.se

Doctoral student:
M.Sc. Nicolas Deschatrettes
Royal Institute of Technology
Department of Energy Technology
Chair of Heat and Power Technology
S – 100 44 STOCKHOLM
deschatr@egi.kth.se

Achieved and planned academic exams

A thesis for the degree of Licentiate of Engineering is planned to be presented in March 2001.

Background

Traditional water turbine design comprises several model or full scale tests of components and prototypes. Though computational fuid dynamics tools are available, they are not well established or considered accurate enough to replace traditional experimental methods in Sweden. Usually the CFD-tools available need to be handled by specialists.

Objectives

The aim of the project is to create a computational fluid dynamics (CFD) tool suitable for analyses of the flow in axial water turbines. The first detailed objective is to compose a tool (mesh generator, solver and post-processor) suitable for performing flow calculation in hydraulic turbines, and compare the results to existing measurements.

The second objective is to evaluate and adjust the ability of the tool to perform parameter studies with trendwise correct information so as to reduce the number of component tests.

The third objective is to evaluate and adjust the ability of computing parameter studies in order to replace some prototype testings.,The goal is that the tool easily can be handled as a complement in the current construction, design and evaluation process of the small turbine manufacturers involved in the project.

Project description

The software of the system is to be adapted from existing software. The pre-processor, the code and the post-processor are to be customized from existing programs in order to achieve the user-friendly aspects of the system, and to provide the most accurate numerical results as possible.

The CFD code must be powerful enough to take into account the phenomena that occur in the type of turbine system concerned by the project, in order to have the most realistic simulation. It has been decided that the code used shall be a 3D Navier-Stokes equations solver and preferably a general purpose code. The programs chosen should also be commercial software. This is to ensure the possibility of a user-support for the industrial partners as well as the reliability of the code.

The system is to be implemented on PC Hardware. PC's are nowadays as fast as, or even faster than, many UNIX Workstations, and their performances are increasing rapidly. The use of PC's allows as well to reduce the hardware cost, and the administration cost.

During the project, two different turbines will be studied.

The first study case is a fully axial turbine (se figure 1). The geometry is provided by ITT-Flygt. The second study case is a Kaplan Turbine (se figure 2). The geometry is provided by Undenäs Turbin AB. The CAD data has been generated from drawings by Undenäs. Measurements have been performed in the past on these turbines.

Figure 1: Fully axial turbine (first study case) and its numerical modelisation

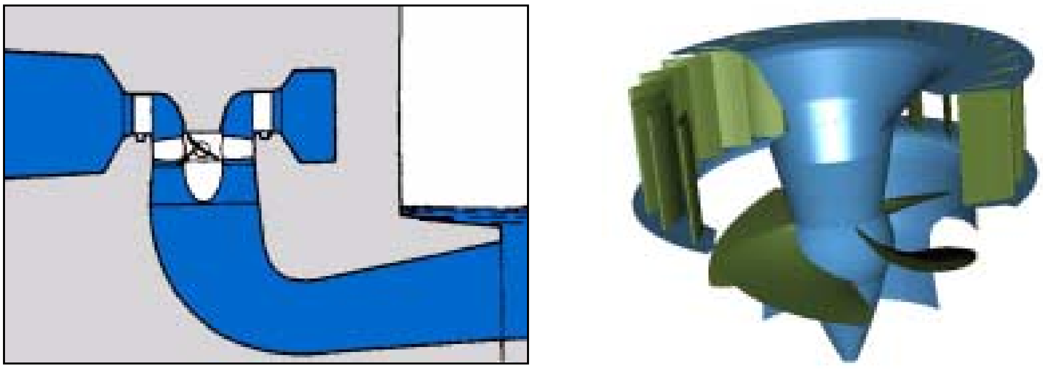


Figure 2: Kaplan turbin, (second study case) and its numerical modelisation

International and industrial collaboration

This project is performed in collaboration with several small water turbine manufacturers in Sweden. If the goal is fulfilled, the tool developed can be valuable to the collaborating partners: Hällaryds Turbiner, ITT Flygt, Svenska Undenäs Turbin AB, Turbin- & Regulatorservice AB, VBB Anlägning AB and Waplan Mekaniska Verkstad AB.

Status

The program used for the numerical calculations is CFX-TASCflow, a code originally developed by the Canadian company ASC. It is a 3D Navier-Stokes solver.

The mesh generation can be done by CFX-Build a general purpose mesh generator or CFX-Turbogrid. CFX-Turbogrid is a mesh generator dedicated to the generation of mesh in blade-rows. The post-processor is provided with the CFX-TASCflow solver.

The program CFX-TurboGrid uses templates for the mesh generation. These templates are describing the block structure, the node distribution and the connectivity between the blocks. One of these templates has been modified for the project use. This allows a better grid generation for the type of blades used in the axial water-turbine.

Macros have been added to the post-processor program in order to present the data the turbine manufacturers are interested in in a fast and easy way.

These data are:

- the velocity distribution in pre-defined planes
- the pressure distribution on the runner blade
- the separation location
- the low pressure locations
- the stagnation locations
- the efficiency, the generated power
- the overall pressure loss
- the mean (circumferencial) values of the velocity components.

The calculations are made on a dual-Pentium PC; the calculation time are reasonable and should be shorter as the calculation speed of this family of Processor has increased since the start of the project. The typical duration of one calculation is 12 hours (one night).

The two study cases have been studied, and the results are highly promising. The efficiency is predicted with an error smaller than 10% compared to the measured efficiency. The analysis of the flow pattern allows diagnosis on the turbine (for instance non-optimal blade orientation). The location of the cavitation is predictable even though cavitation is not modeled.

Simulation of off-design points has been done, and the conclusions were that even if the pattern of the flow is correct, the value of the efficiency cannot be predicted with this numerical method. This is probably due to the absence of cavitation modeling which allows pressure not physically possible (extremely low pressure). A post-processing solution to this problem is being sought at this moment.

In figure 3, it is possible to see the efficiency of the first turbine studied, the fully axial turbine, at the same rotational speed and same blade orientations, but with different mass-flows. The efficiency calculated from the result of numerical simulations has been plotted for comparison. The two curves differ by a shift in the horizontal direction (mass flow) which might be due to small errors in the files defining the geometry of the turbine. This shift also appears when comparing the power developed by the turbine or the head of the machine. The simulations also give slightly higher efficiencies, as expected.

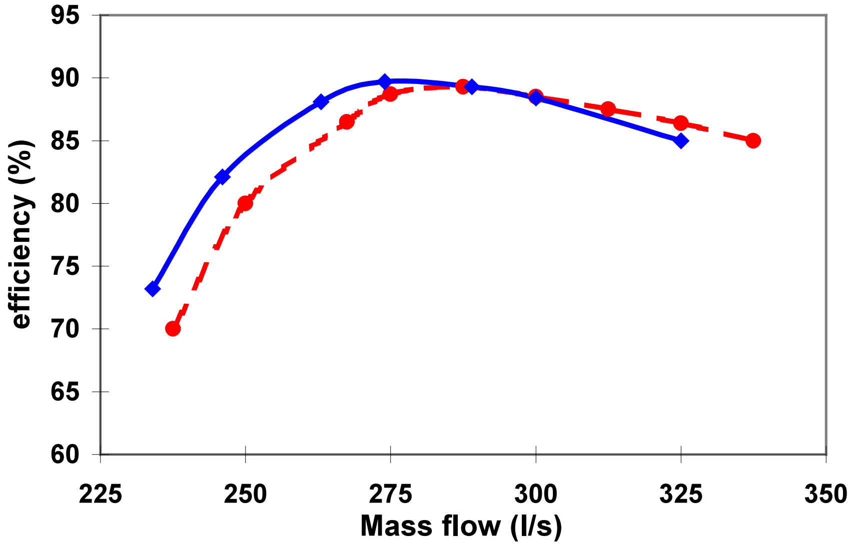


Figure 3: Comparison of the measured (red curve) and calculated (blue curve) efficiency, first study case.

Table 1: Comparison of engineering quantities for the Kaplan turbine (m = measured quantity)

n (rpm)	215	225	250
Q (m^3/s)	21,8	22,3	23
H (m)	8,54	8,63	8,08
H/Hm	1,095	1,106	1,036
Pm (MW)	1,49	1,51	1,51
P (MW)	1,625	1,655	1,46
P/Pm	1,091	1,096	0,967
ηm (%)	89,2	88,7	86
η (%)	88,85	87,66	80,06
η/ηm	0,996	0,966	0,931

In table 1, the power, head and efficiency are compared between the simulations and measurements on the second turbine case for thre different mass-flows. The calculation error for the efficiency does not exceed 7% in this case. Calculations on wider ranges of mass-flows will hopefully confirm these promising results.

Table 2: Influence of the Turbulence intensity (Tu) on the engineering quantities at the same duty points for the Kaplan turbine

Tu (%)	2,5	5	10
H (m)	8,65	8,64	8,63
P (MW)	1,659	1,657	1,655
h (%)	87,71	87,69	87,66

By modifying the turbulence intensity at the inlet of the numerical domain, the influence of turbulence on the overall characteristics of the turbine calculations has been checked. Table 2 shows that the influence of the turbulence intensity on the calculations was lower than ±0.1%.

Scientific results

1. Deschatrettes N., ”Development of a User-friendly Numericalsystem for the analysis of the flow in small axial water turbine”, Internal Report HPT, Department of Energy Technology, Chair of Heat and Power Technology, Royal Institute of Technology, Sweden, 2000

ELFORSK

SWEDISH ELECTRICAL UTILITIES' R&D COMPANY

Elforsk AB, S-101 53 Stockholm, Sweden.

Telephone: +46 8 677 25 30. Telefax: +46 8 677 25 35

Swedish National Energy Administration

Swedish National Energy Administration, P.O. Box 310, S-631 04 Eskilstuna, Sweden.

Telephone: +46 16 544 20 00. Telefax: +46 16 544 20 99

GE Hydro

GE Energy (Sweden) AB, P.O. Box 1005, S-681 29 Kristinehamn, Sweden.

Telephone: +46 550 84 800. Telefax: +46 550 18 998

WAPLANS MEKANISKA VERKSTADS AB

Waplans Mekaniska Verkstad AB, S-830 44 Nälden, Sweden

Telephone: +46 640 177 00 Telefax +46 640 177 80