Application of computational fluid dynamics to the design of pico propeller turbines

R.G. Simpson, A.A. Williams

Nottingham Trent University

Abstract

A research project is currently being undertaken in collaboration with Practical Action (ITDG) to develop a standard design procedure for pico propeller turbines that can be manufactured locally in developing countries. A 5 kW demonstration turbine has been set up at a test site in Peru and Computational Fluid Dynamics (CFD) has been used to obtain overall performance data for the turbine and to assist in the design of a new rotor. It was found that an incorrect matching between the turbine rotor design and the available flow rate at the site significantly affected the turbine operation and in order to provide an acceptable performance it was possible to adjust just the runner design and operating speed of the turbine. The paper will present the initial CFD and field test results, and discuss the process by which computational fluid modelling has been used as an appropriate design tool.

Introduction

Very small hydroelectric power schemes, with outputs of less than 5 kW, can provide cost-effective power to many remote rural communities. More than 1000 'pico hydro' schemes have been installed at high head sites in recent years, mainly in Nepal[1] where batch-produced Pelton turbine-induction generator units have been manufactured locally. Such schemes are cost-effective where standardised equipment is available[2].

Low head hydro sites (2 to 10m) have an even larger potential for providing electricity in rural areas of developing countries but the harnessing of this potential is severely hampered by the lack of an appropriate turbine design[3]. Fixed geometry propeller turbines are one of the most cost-effective turbine options for low head pico hydropower. The leading market and manufacturer of small, low head hydropower units is China[3]. However, these turbines have a poor record of reliability and to date, none of the current designs is available for local manufacture in other countries[4].

With this in mind, a research project was undertaken in collaboration with Practical Action Peru (ITDG) to develop a standard design procedure that will result in cost-effective propeller turbines that can be manufactured locally, and yet have good performance. Previous research at Nottingham Trent University has shown that one of the most promising options for low head pico hydro is a fixed geometry propeller turbine with a spiral casing[5]. This has the advantage over a tubular design in that the runner bearing arrangement can be overhung and no water lubricated bearing is needed reducing cost and maintenance requirements. Furthermore, there is a possibility that such a design can directly drive a generator with the turbine runner fixed to a shaft extension, hence, increasing the mechanical

efficiency of the system and reducing cost and maintenance requirements associated with a transmission belt.

The research project involves a combination of field testing, accurate laboratory testing and current computational modelling methods. Computational Fluid Dynamics (CFD) has become a useful and complementary tool to the designer and can be used for simulating, designing and analysing complex three-dimensional flows in turbomachinery. With the emergence and improvement of the commercial CFD software industry, computational modelling is being used much more by engineers with applications spanning a very wide area of engineering. To date, there appears to be a limited amount of research into the use of CFD for analysing the design of small propeller turbines suitable for decentralised applications.

The demonstration turbine

The demonstration turbine for the project is located at a small farm in the northern highlands of Peru and is shown in Figure 1. The turbine has a horizontal axis and consists of a spiral casing to introduce swirl, six fixed radial guide vanes, a simplified four-bladed rotor fabricated from sheet metal and a conical draft tube with a 90° elbow. The flow enters the spiral casing through a short penstock attached to the bottom of the forebay tank and the total gross head for the site is approximately four metres. The turbine shaft is connected via a pulley and V-belt to an Induction Motor as Generator with an Induction Generator Controller.

Figure 1 – Demonstration turbine in Peru

Preliminary testing of the turbine revealed several problems. During operation, water was being emptied from the forebay tank resulting in a much lower head than the available four metres and the turbine was not producing sufficient power to get the generator up to operating voltage. It was decided to use Computational Fluid Dynamics (CFD) as a tool to assist in diagnosing the problem and to formulate a solution.

Original rotor design - preliminary CFD results

The commercial suite of software, ANSYS CFX, was used for all aspects of the CFD study including modelling, meshing, solving and post-processing of the results. The ANSYS CFX solver uses an element-based finite volume approach to solve the full Navier Stokes equations to second order accuracy[6]. Computations were performed in parallel on a dual processor workstation with AMD Opteron 246 processors (2.0 GHz, 2GB RAM).

The fluid volume was separated into four separate flow domains. These consisted of the spiral casing, guide vanes, rotor passage and draft tube (see Figure 2). All domains were in the stationary frame of reference except the rotor passage, which was solved in a rotating frame of reference at the operating speed. In order to save on computational time, only one of the four rotor passages were modelled and a periodic boundary condition was applied. Frozen rotor interfaces were applied between the stationary and rotating domains. In this way, the two frames of reference connect such that they each have a fixed relative position throughout the calculation. This model produces a steady state solution and does not model transient mixing effects between the stationary and rotating components[6].

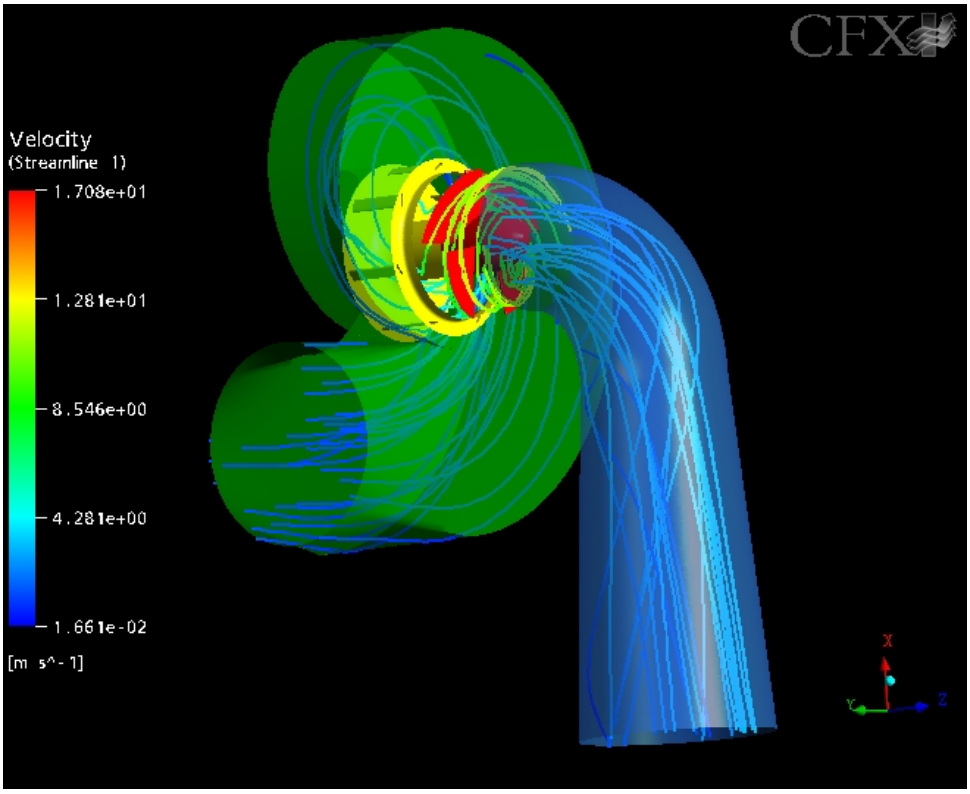

Figure 2 – Flow domains used for preliminary CFD simulations

The total number of nodes for the entire mesh was 926,000 with 1.44 million elements. An unstructured tetrahedral and prism mesh was created for the spiral casing and draft tube domains. A structured hexahedral mesh was used for the guide vane domain and rotor passage. A total pressure inlet boundary condition equivalent to four metres gross head was used and the outlet boundary condition was set to an opening with an average relative pressure to atmosphere of zero. These boundary conditions were found to be more stable than

other configurations tested and had the advantage of allowing simulations at an effective constant head. The rotational speed of the rotor passage was varied between 400 to 1600 rpm in order to get a turbine characteristic curve for torque, power, flow rate and efficiency over the full speed range. The recommended turbulence model[7] used for the simulations was the k-ε (k-epsilon) model, which within CFX uses a scalable wall function to improve robustness and accuracy near the wall.

Figure 3 shows the results from the original rotor simulations plotted for a constant speed of 600 rpm. Maximum turbine efficiency was predicted to be approximately 55% with a head of 3.1 metres, flow rate of 256 l/s and corresponding power output of 4.2 kW. However, available flow rate at the site was measured to be approximately 180-220 l/s and therefore, a more accurate operation point would be at a head of 2.0 metres, flow rate of 210 l/s and output power of 1.7kW. These results therefore confirmed the problems encountered during initial operation of the turbine with the water in the forebay tank emptying from a potential four metres gross head to only two metres.

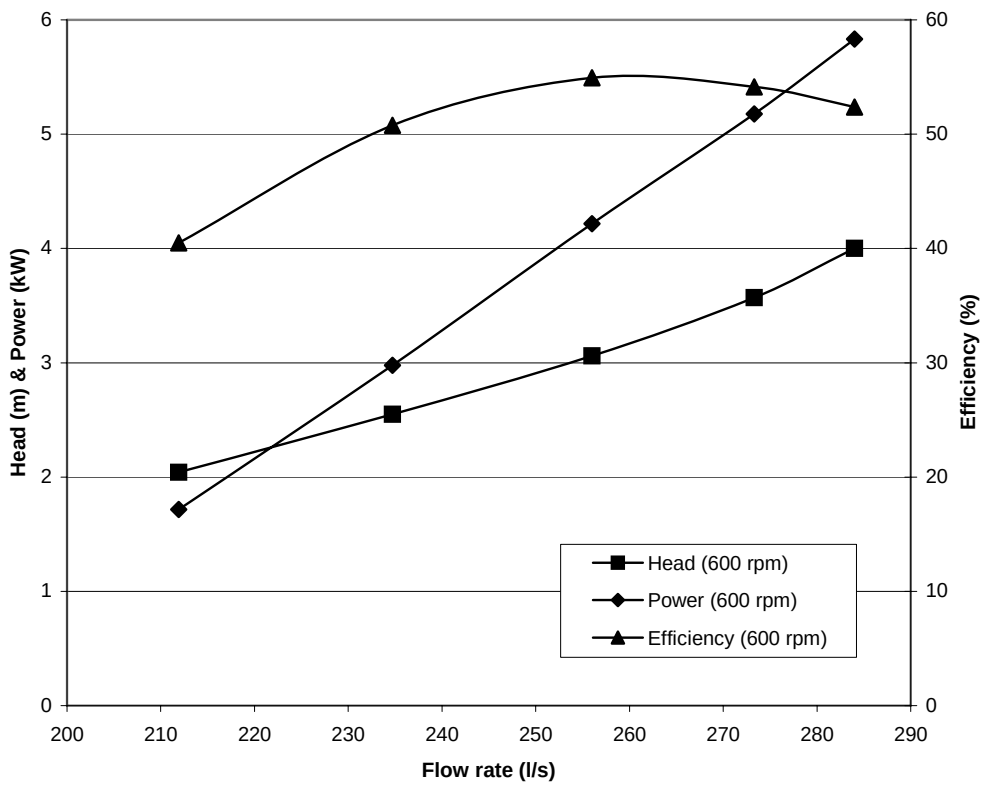

Figure 3 – Preliminary CFD results for original rotor at a speed of 600 rpm

From further inspection of the original rotor design it was concluded that this was due to a mismatch between the rotor blade angles and the available flow rate at the site. The original rotor blades had a very steep angle (relative to the tangential direction) and a low solidity ratio and in order to improve the turbine operation it was decided to design a new rotor with flatter blades and a higher solidity ratio.

New rotor design – CFD simulations

The new rotor blades were designed using conventional theoretical methods from a standard text[8]. Figure 4 shows a comparison between the blade angles of the original design and the new rotor design.

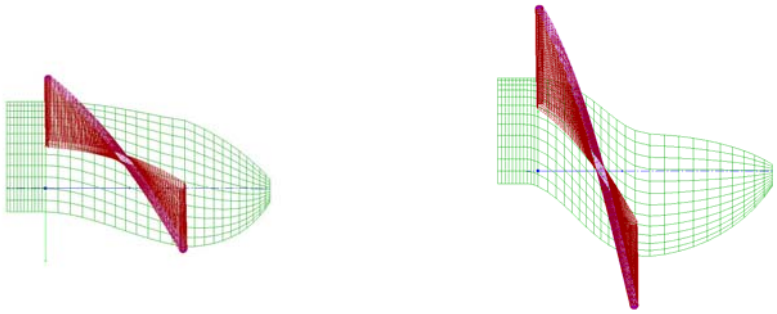

Figure 4 – Comparison between old (left) and new (right) rotor designs

For the new rotor simulations a mesh was created of the same size and quality as the original rotor passage and the CFD simulations were performed using the same boundary conditions and turbulence model described in the previous section. As expected it was found that the new rotor geometry produced a significant reduction in the flow rate needed for a given power output and the power curve demonstrated a much flatter characteristic over the speed range than the original rotor (see Figure 5).

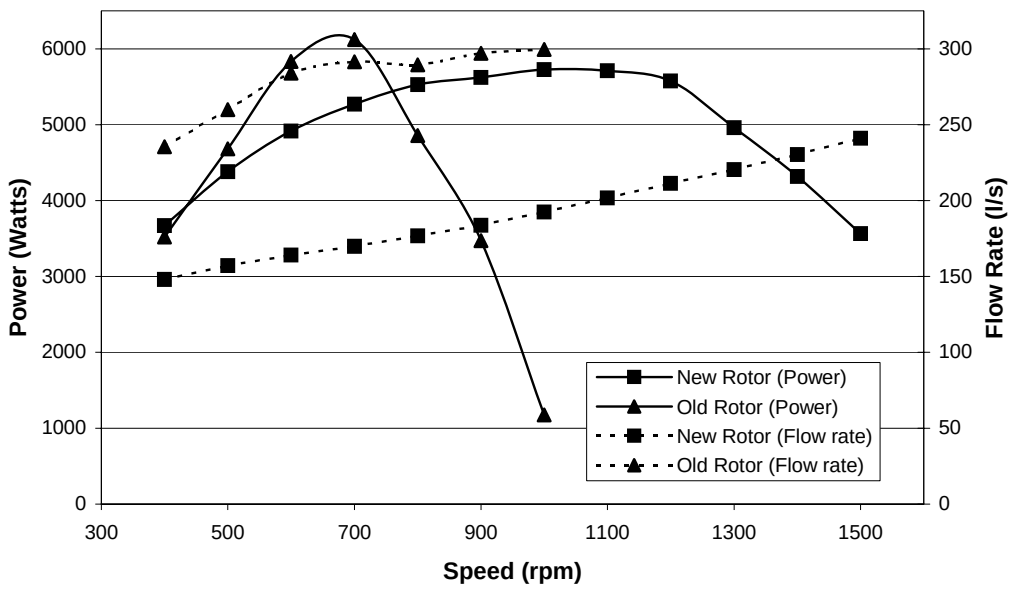

Figure 5 – CFD results for power and flow rate at constant head of 4 metres

Furthermore, the predicted best efficiency for the new rotor design increased to 80% at 800 rpm compared to the 55% at 600rpm predicted for the original rotor design.

Field tests and revised CFD simulations

A new rotor was manufactured in Peru by bending and twisting flat sheet metal into the required shape of the blades and then welding to the hub (see Figure 7). One side effect of this manufacturing technique was that the blades ended up with a slight S-shape due to the twisting.

Figure 7 – Simplified rotor manufactured by Celso Davilla, Peru

Data obtained from the turbine tests of the new rotor consisted of torque, speed, flow rate and total head. From these the mechanical power output and efficiency was calculated. A brake using a flat belt and spring balances was used to measure the torque and the speed was measured using a handheld optical tachometer. Gross head between the level of water in the forebay tank and tailrace was measured using predetermined height markings. By constructing a flume at the exit of the turbine it was also possible to measure the flow rate through the turbine.

In order to get a more accurate comparison between the field tests and the CFD results, a new CFD model was created taking into account parameters and geometry changes that were not included in the original simulations. These improvements included:

- Inclusion of the penstock flow volume.
- Revised geometry for the rotor blade using measurements from the finished product and incorporating the S shape of the blades.
- Revisions to the geometry for the spiral casing and guide vanes taken from measurements onsite.
- Inclusion of all four passages for the rotor domain. This provides a more accurate model taking into account the non-uniform nature of the flow from the spiral casing.
- Inclusion of a 3mm blade tip gap (approximately 3.5% of blade span length).

As part of the ongoing research several other factors are currently being investigated including roughness effects and leakage losses through the hydrodynamic seal.

Figures 8, 9, 10 and 11 show comparisons between the field test results, original and revised CFD simulations for the torque, power, flow rate and efficiency over the speed range.The revised CFD simulations are in closer agreement with the field test results than the original set of simulations for both Torque and Power in the low speed range (Figures 8 and 9). However, in the higher speed range (above 1200 rpm) both sets of CFD results tend to over predict the torque and power output. This may be due to the influence of stronger transient effects at the higher speeds, which will affect the accuracy of modelling in the draft tube and at the domain interfaces.

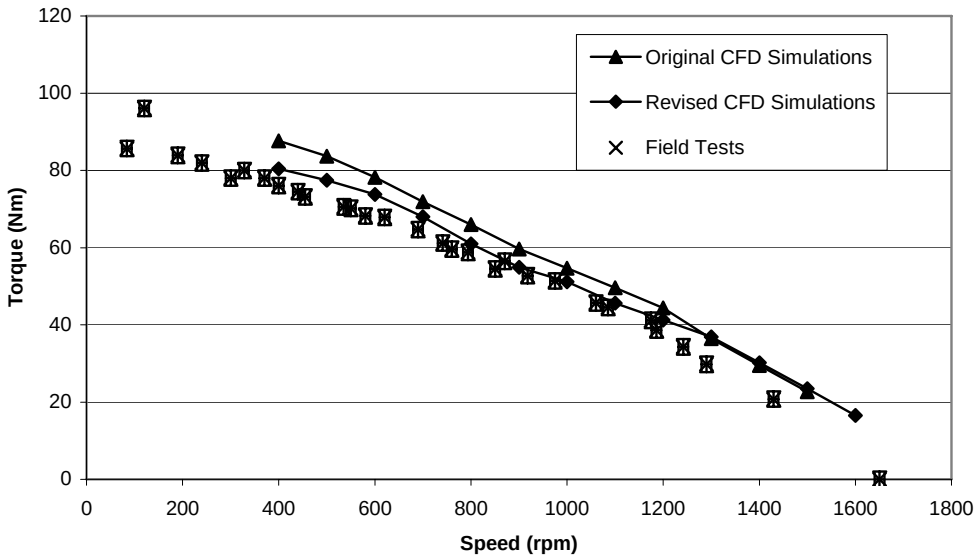

Figure 8 – Comparison of torque for a constant head of 4 metres

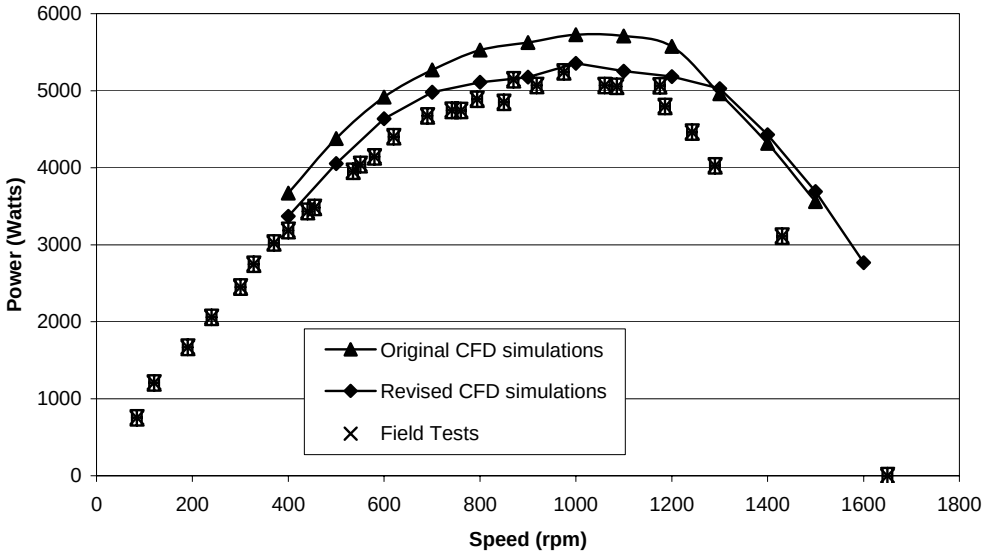

Figure 9 – Comparison of power output for a constant head of 4 metres

The flow rate predicted by both sets of CFD simulations does not agree very well with the field test results (Figure 10). As a result of the under prediction of the flow rate using CFD, the turbine efficiency is thus over-predicted by approximately 10% at the best efficiency point (Figure 11). This could be due to several reasons including inaccuracy of the measurement device onsite or inaccuracy in the computational model due to fluid leakage flows not being taken into account. Further investigation is currently underway to determine the effect of further changes to the CFD model.

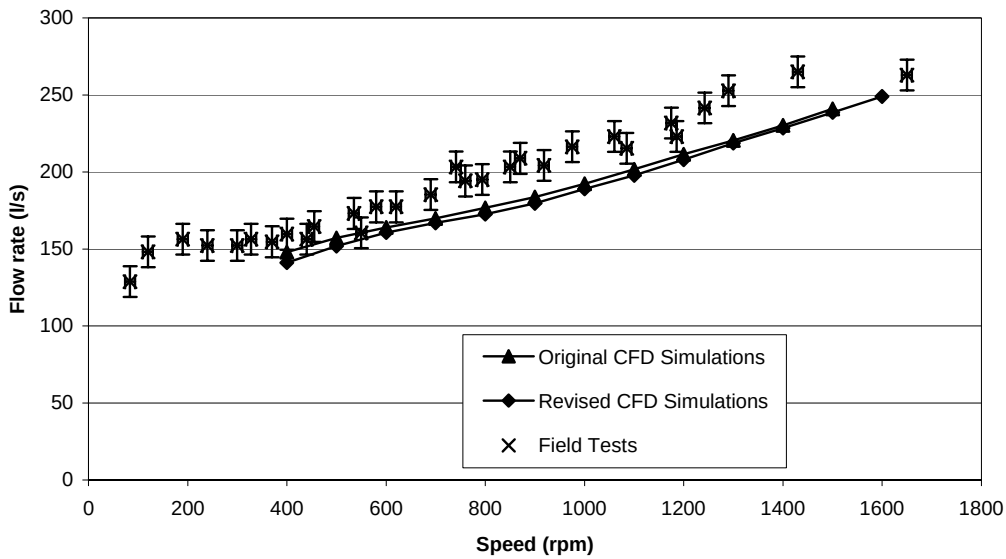

Figure 10 – Comparison of flow rate for a constant head of 4 metres

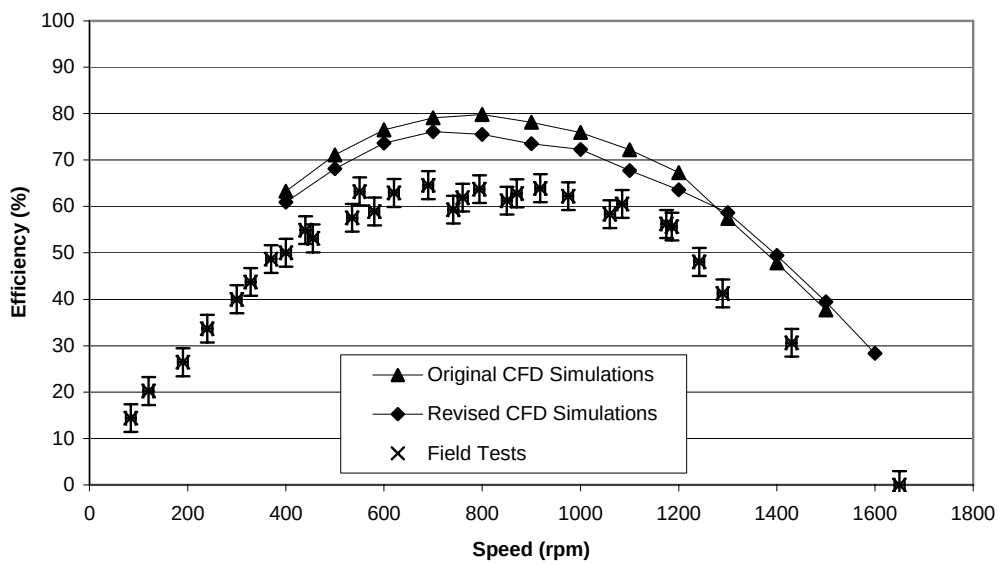

Figure 11 – Comparison of mechanical efficiency for a constant head of 4 metres

The overall total pressure losses for the two sets of CFD simulations are summarised in Table 1. These were calculated using the difference in average total pressure across each domain interface for the original and revised CFD simulations. In the case of the rotor total head loss calculation the total pressure difference across the domain was adjusted by subtracting the equivalent head that is being converted to shaft output power. From the table it can be seen that the total pressure loss for each component (excluding the rotor) has not changed significantly and the inclusion of the penstock only has a small effect on the overall pressure losses.

Description	**Torque (Nm)**	**Power (W)**	**Average total pressure loss**				
			Penstock	**Casing and vanes**	**Rotor**	**Draft tube**	**Exit**
Original Simulation	65.96	5526	0m	0.40m	0.06m	0.18m	0.17m
			0 %	49%	7%	23%	21%
Revised Simulation	60.97	5108	0.02m	0.43m	0.20m	0.16m	0.17m
			2%	43%	21%	17%	17%

Table 1 – Comparison of total pressure losses for the CFD simulations at 800 rpm

Loss in torque and hence output power for the revised simulation could be due to a number of factors including:

- modelling of the S-shape geometry of the blades,
- inclusion of all four rotor passages, and the
- reduction in the effective span of the blade when the tip gap is taken into account.

These factors result in a decrease in rotor efficiency and a subsequent drop in overall turbine efficiency. For example, the rotor efficiency for the original simulations is estimated to be 98% but it drops to 94% when the extra influences are taken into account in the revised simulation. Future research will also focus on the design of the spiral casing and guide vane arrangement, which appears to contribute close to half of the overall pressure losses for the turbine.

Conclusions

A fixed geometry propeller turbine with a spiral casing was designed and installed at a field site in Peru and field testing showed the turbine to have an overall mechanical efficiency of 65% after modifications were made. Results from a preliminary CFD study were used to analyse the original rotor design, which was found to have a low efficiency and the blade geometry did not match the flow conditions at the site. A new rotor was fabricated using a simplified method of manufacture and it was found to give acceptable performance. Some discrepancies between the CFD simulations and the field test results exist and work is continuing to improve both the CFD modelling and measurement techniques used at the site. The overall aim of the project is to use these results and future investigations to develop a manual and method for the design of pico propeller turbines for local manufacturers in developing countries.

References

1. CADEC (Community Awareness Development Centre). Micro-hydro yearbook of Nepal 2002, H.M.G. Nepal, Alternative Energy Promotion Centre, Kathmandu, 2002.

2. Maher, P., Smith, N.P.A. and Williams, A.A. Successful strategies for rural electrification using pico hydro power, *Proceedings of the 1st EU-China Small-Hydro Industry Conference*, Hangzhou, China, 1999.

3. Waltham, M. et al. Low head micro hydro potential: Final report to ODA under TDR contract R6482. Intermediate Technology Group, Rugby, UK, 1996

4. Williams, A.A., Upadhyay, D.R., Demetriades, G.M. and Smith, N.P.A. Low Head Pico Hydropower: A Review of Available Turbine Technologies, *World Renewable Energy Congress VI*, Elsevier Science Ltd, 2000

5. Demetriades, G.M., Williams, A.A. and Smith, N.P.A. A simplified propeller turbine runner design for stand alone micro hydro power generation units, *International Journal of Ambient Energy*, 17(3) pp 151-162, 1996.

6. ANSYS Canada Ltd, CFX-5 Solver Modelling, ANSYS Canada Ltd, Ontario, 2004

7. ANSYS Canada Ltd, CFX-5 Best Practices Guide for Turbomachinery, ANSYS Canada Ltd, Ontario, 2004

8. Nechleba, M. Hydraulic Turbines: their design and equipment, Artia, Prague, 1957