Wake effects in tidal current turbine farms

A J MacLeod, **BEng(Hons)**
S Barnes, **BEng(Hons)**
K G Rados, **PhD**
Prof. I G Bryden, **BSc, PhD, CEng, CPhy, FIMechE, FIMarEST, FInstP**

The Robert Gordon University, Aberdeen

SYNOPSIS

In the current search for cleaner, cheaper solutions to the ever-increasing demand for energy, one of the most promising and often overlooked sources is that of tidal current energy. With several government-commissioned studies having concluded that there exists a European potential resource in excess of 12.5 GW, this is a significant and untapped source of power. Although work is well underway to develop tidal turbines in order to exploit this source, there has been no detailed study of optimal layouts for tidal turbine “farms”. The paper presented here describes the application of Computational Fluid Dynamics techniques to this problem in the modelling of tidal turbine wakes. Results for both one turbine alone and two turbines positioned in tandem are included.

INTRODUCTION

The oceans of the world have long impressed men with their vastness and power. Until recently, however, there has been little real success in tapping that power and using it to alleviate the huge and increasing energy demands of modern society.

In recent years, great advances in scientific and engineering knowledge relating to the offshore environment have been made, largely in the quest to recover oil from sub-sea fields. As easily accessible resources are exhausted, the need for means of exploiting reservoirs located in unhospitible waters has driven the development of new engineering methods for working in harsh conditions and deep water. These advances have allowed the development of tidal turbines; similar in concept to land-based wind turbines, but designed to operate submerged offshore to extract energy from tidal currents.

Tidal currents as a source of energy to generate electricity have many attractive features. They are reliable and predictable; one of the major failings of many forms of natural alternative energy is the lack of predictability, leading to periods when little or no energy is generated, either through lack of available motive power, or an equipment-damaging excess. Tidal turbines have the potential for minimising both visual and noise "pollution", and are likely to be located in areas where few people would be affected by their presence.

As with conventional wind turbines, the most cost-effective means of deployment is to locate several turbines together, thus minimizing many costs such as cabling and servicing. The grouping together of several turbines poses some problems, however. In particular, the wake of upstream turbines may interfere with downstream turbines, reducing the amount of energy produced. In the case of land-based wind farms, research has already been carried out to study the wake patterns generated by wind turbines and software has been developed to find optimal layouts for any location with a given number of turbines and annual wind conditions.

The main object of the present work was to carry out initial research into the development and interaction of wakes generated by tidal current turbines as a first step to producing software similar to that mentioned above. This has largely been achieved through the use of CFD, with additional water tank experiments for comparative purposes.

AUTHOR'S BIOGRAPHY

Angus John MacLeod has worked since his graduation as research assistant to Dr Kostas Rados at the Robert Gordon University, Aberdeen. His work has been centred on the project discussed in this paper.

Stuart Barnes has recently graduated from the Robert Gordon University, Aberdeen.

Dr Kostas Rados is employed as a lecturer in Environmental Engineering at the Robert Gordon University, Aberdeen.

Prof. Ian Bryden is currently the Associate Dean of the Faculty of Design and Technology. He has over 20 years of marine renewable energy research experience.

METHODS

The physical problem

The tidal current devices currently envisaged have a rotor diameter of around 20 metres, and are expected to operate in water depths of around 120m with current velocities in the region of 4 metres per second.

In order to reduce the problem to a manageable size, the simulation was broken up into several parts, which have been investigated in turn:

- The local tidal stream velocity field without turbines
- Modelling of a single turbine wake
- Modelling of the interaction between multiple wakes.

CFD modelling

The bulk of the present CFD work has been carried out using our in-house developed code, "3D-NS". This is a fully-elliptic turbulent 3D Navier-Stokes numerical solver with k-epsilon closure, and is capable of simulating clusters of turbines in any configuration.

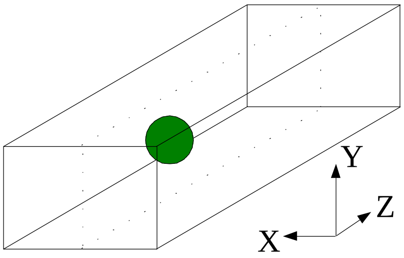

Fig 1- The computational domain

For the single-turbine case, a symmetry condition was applied (see dashed lines in figure 1) and only half of the physical domain modelled, thus taking advantage of the symmetrical nature of the problem and drastically reducing computational time. For multiple turbine cases, however, the code was modified to allow turbines to be placed in any configuration and therefore the whole physical domain was modelled.

Computational Domain

The computational domain consisted of a rectangular “box” with an upstream distance (i.e. -Z direction) of five diameters from the first turbine and a downstream distance of fifteen diameters from the last turbine. In the X direction, there was a distance of seven diameters on either side of the turbine tips for the multiple-turbine model. The total distance in the Y-direction was varied to investigate various parameters, but most runs were completed with four and a half diameters from the turbine tip to the seabed and half a diameter from the turbine tip to the sea surface.

The computational domain included the turbine disks, which were approximated as semi-permeable membranes generating pressure jumps (see Eqn 1) to account for the thrust force of real turbine rotors.

$$\Delta P = C_t \times \frac{1}{2}\rho \times u^2 \tag{1}$$

The seabed has been simulated by a no-slip wall boundary condition, while a slip condition (slip wall) is currently applied at the sea surface. At the downstream outlet boundary, a conservation of mass condition was applied.

RESULTS

Single wake

Initially, runs modelling the wake of a single turbine were undertaken. As well as grid-independence checks, tests were carried out with the turbine submerged at different depths to determine what effects, if any, the "slip wall" boundary condition applied at the sea surface had on the wake formation.

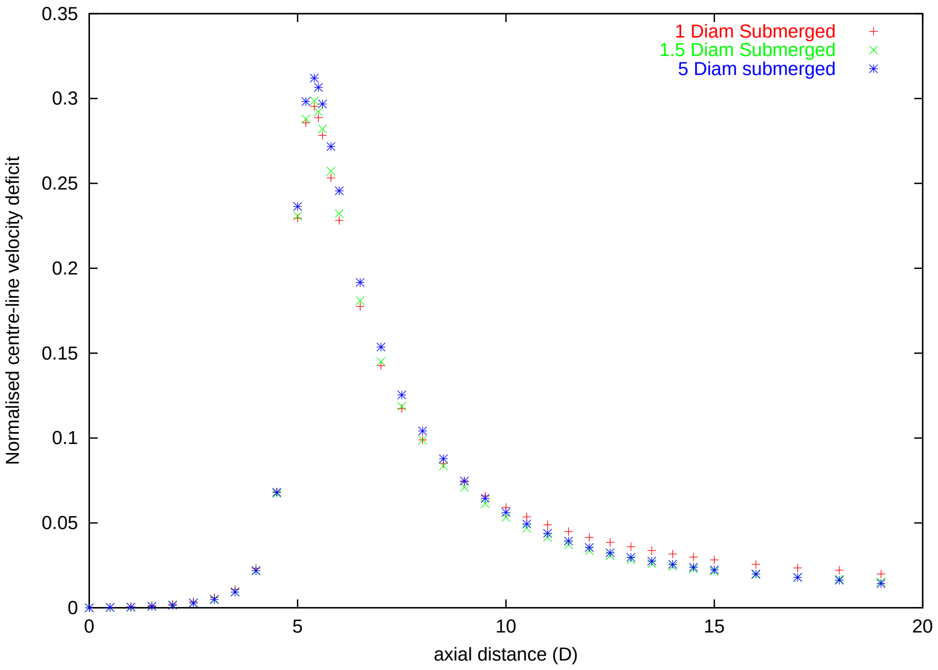

Fig 2 - Centreline velocity deficit for various turbine depths

It can be seen from figure 2 that the centreline velocity deficit is barely affected by the depth of submergence, even through a range of 1 to 5 diameters depth (turbine centre to sea surface).

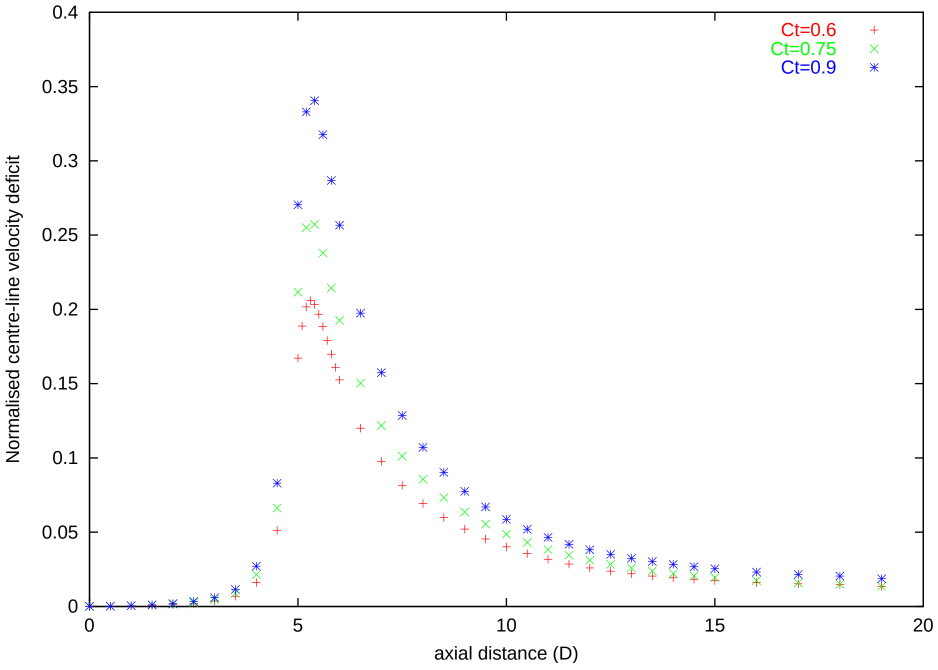

Fig 3 - Centreline velocity deficit for various turbine thrust coefficients

Figure 3 shows the expected behaviour when the thrust coefficient of the "turbine" is altered. The centreline velocity deficit caused by the turbine increases as turbine thrust coefficient increases.

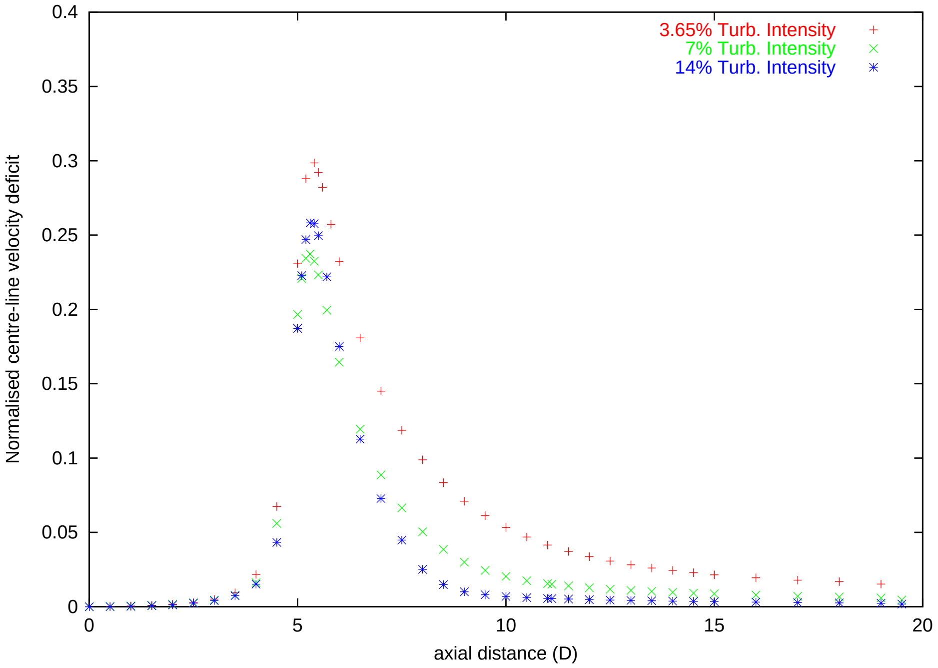

Fig 4 - Centerline Velocity Deficit for varying levels of ambient Turbulence Intensity

The plot in Figure 4 shows the centerline velocity deficit for three runs completed using various levels of ambient turbulence intensity. It can be seen that increased levels of ambient turbulence intensity lead to a faster recovery of the flow to it's original velocity. This is indicated by the slope of the velocity deficit lines.

Double Wake

All runs to date for the two turbine case have involved two turbines in a tandem arrangement. Two main cases have been studied; five diameters separation between turbines and eight diameters separation between turbines.

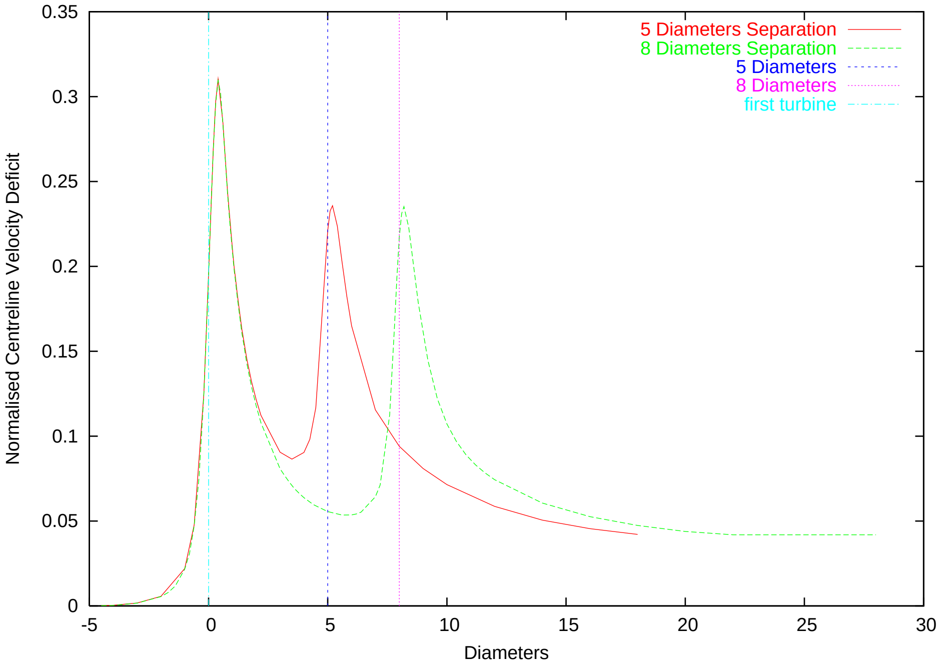

Fig 5 - Centerline Velocity Deficit for turbines at 5 and 8 diameters spacing

It can be seen that there is almost no difference in the centerline velocity deficit at the second turbine in the two cases shown in Fig. 5; thus, there would be little or no benefit in leaving eight diameters downstream distance between turbines, as this would increase cabling costs with no corresponding improvement in turbine power output.

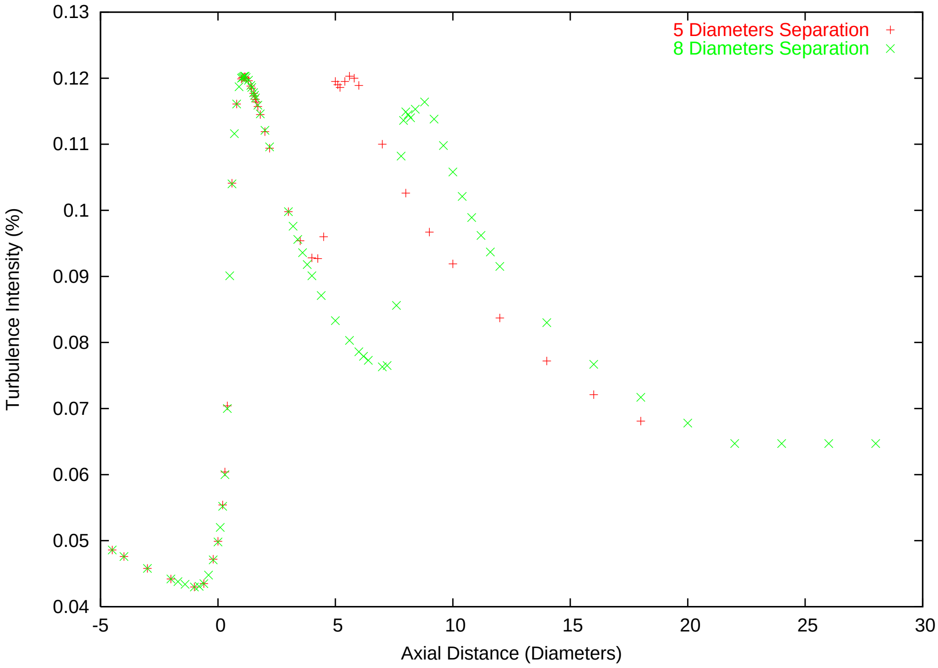

Fig 6 - Centerline Turbulence Intensity for turbines at 5 and 8 diameters spacing

Fig. 6 shows the predictable result that the turbulence intensity at the second turbine is slightly higher with five diameters spacing when compared to eight.

DISCUSSION

The results from 3D-NS show that the code is predicting much of the expected flow behaviour. Features such as the increased flow recovery rate with higher ambient turbulence intensity levels and the slower recovery downstream of turbines with higher thrust coefficients are predicted much as expected.

For full validation of these CFD results, however, comparisons must be made with the results of water tank scale model experiments currently underway. Unfortunately these were not available at the time this paper was compiled.

CONCLUSIONS

1. The wake formed downstream of a single tidal turbine submerged at various depths has been modelled using CFD techniques and water tank experiments.
2. The CFD results have shown (as expected) that flow recovery is faster in areas of greater ambient turbulence intensity.
3. The wakes formed downstream of two tidal turbines in tandem at two different separation distances have been modelled using CFD.
4. The CFD results have also indicated that for turbines with higher thrust coefficients, the wake recovery is slower.
5. Comparisons with measurements from water tank experiments are necessary to validate the results presented here.

ACKNOWLEDGEMENTS

This research has been wholly funded by the EPSRC (Engineering and Physical Sciences Research Council) whose support is gratefully acknowledged.