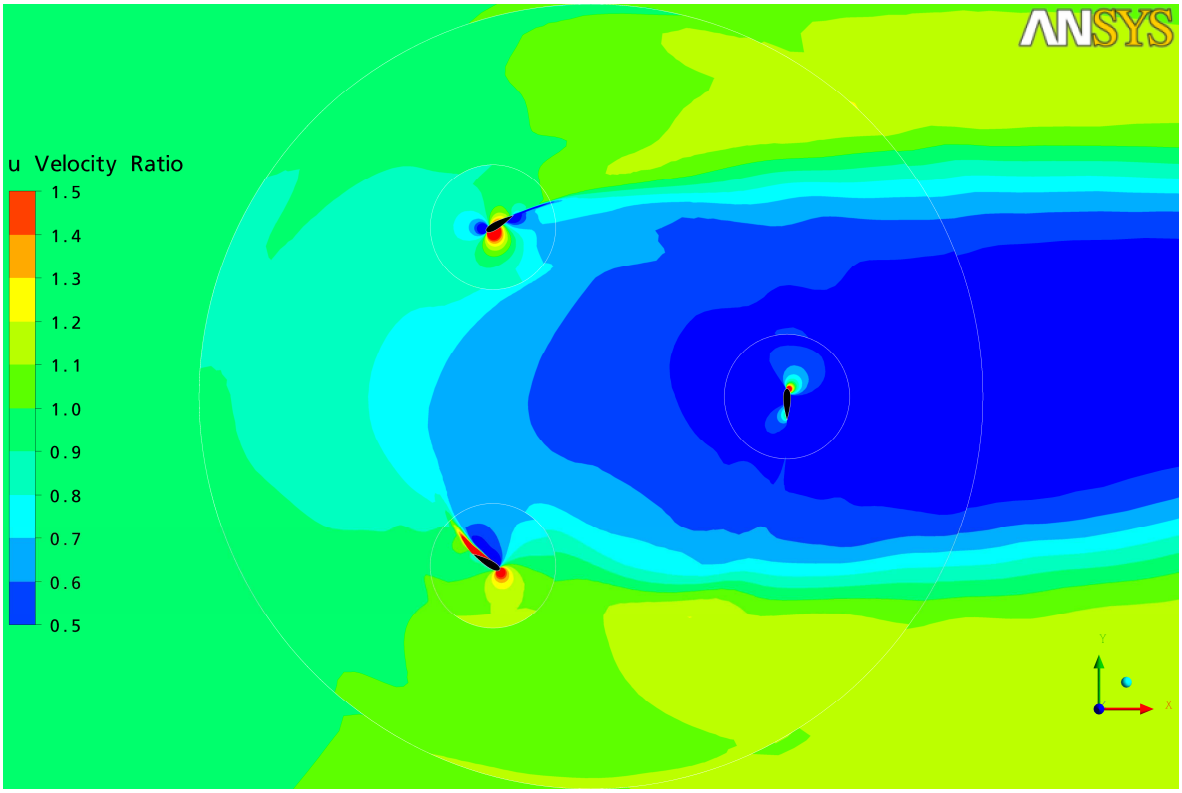
Turbine analysis using computational fluid dynamics

Gareth Gretton, Tom Bruce, David Ingram
Work stream 1: Physical and numerical convergence

Introduction

Computational fluid dynamics can be an immensely powerful tool for investigating the hydrodynamic behaviour of tidal current turbines. Before using CFD to investigate *physical* parameters, two studies should be carried out. First, the effect on the solution of a wide range of *numerical* parameters must be discerned in a process known as *verification*. Second, the numerical results are compared to those from a physical test case: *validation*. Following this, quantitatively accurate results with an estimable error can be generated.



Numerical parameters

What effect on the solution by altering:

- Turbulence model
- Spatial discretization: Δx, Δy, L
- Temporal discretization: Δt, t_{total}
- Iterative convergence (residual target)

Conclusions/Guidance

- SST k-ω and Spalart-Allmaras are good choices for the turbulence model
- $\Delta y_1^+ \sim 1$, $L = 40D$, $\Delta\theta = 2°$
- Run simulation for 60 periods on coarse mesh with loose iterative convergence
- Interpolate coarse mesh solution on to fine mesh and run for a further 3 periods with tighter iterative convergence

Institute for Energy Systems
The University of Edinburgh

EPSRC
Engineering and Physical Sciences Research Council