



# Technical Paper

## Wastewater pumps

### Session 2-3

## Application of CFD to the Hydraulic Performance Prediction of Single Blade Impeller Pumps.

**Author:**

**Mr. Richard McEvoy**  
Postgraduate Researcher  
University of Limerick  
Limerick, Ireland

**Co-Author 1:**

Dr. Brian de Souza  
Research Fellow  
University of Limerick  
Limerick, Ireland

**Co-Author 2:**

Dr. Andrew Niven  
Principal Research Investigator  
University of Limerick  
Limerick, Ireland



## Summary

This paper demonstrates the ability of Numerical Simulation to predict the hydraulic performance curves of single-blade-impellers. A low specific speed ( $N_{sd} = 58$ ) impeller was modelled using the commercially available Ansys-CFX-11.0 Navier Stokes solver. A hybrid tetrahedral was used globally throughout the pump domain while locally; a prism mesh was used to provide resolution of the boundary layer.

Numerical simulations were run for a total of ten revolutions, at which point the solution was found to have converged to a periodic stable solution. Examination of the flowfield showed that the single-blade-impeller generates a strongly transient pulsating flow. The impact of grid refinement was investigated and found to have minimal influence on the hydraulic performance results ( $<1\% \eta$ ). The numerically simulated head/flow curve was compared to that measured experimentally and was found to over-predict the head generated. The over-prediction was associated with the presence of tipgap clearance in the experimental test. This study provides an example of how it is now possible, through the use of Computational Fluid Dynamics to model the single-blade-impeller flowfield.