A CFD Study on Design Iterations of the Hydrobox Energy Converter

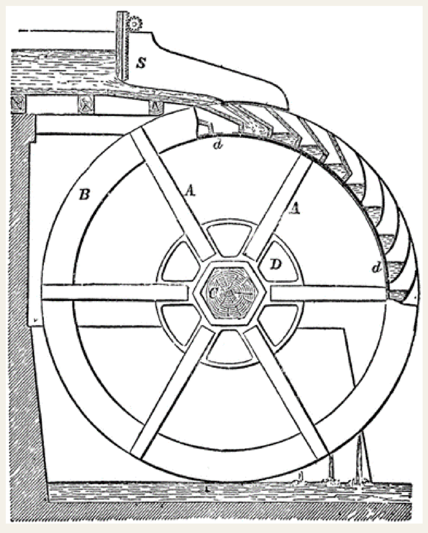
Wenguang Li, Kiran Ramesh*

School of Engineering, University of Glasgow, Glasgow, G12 8QQ

Introduction

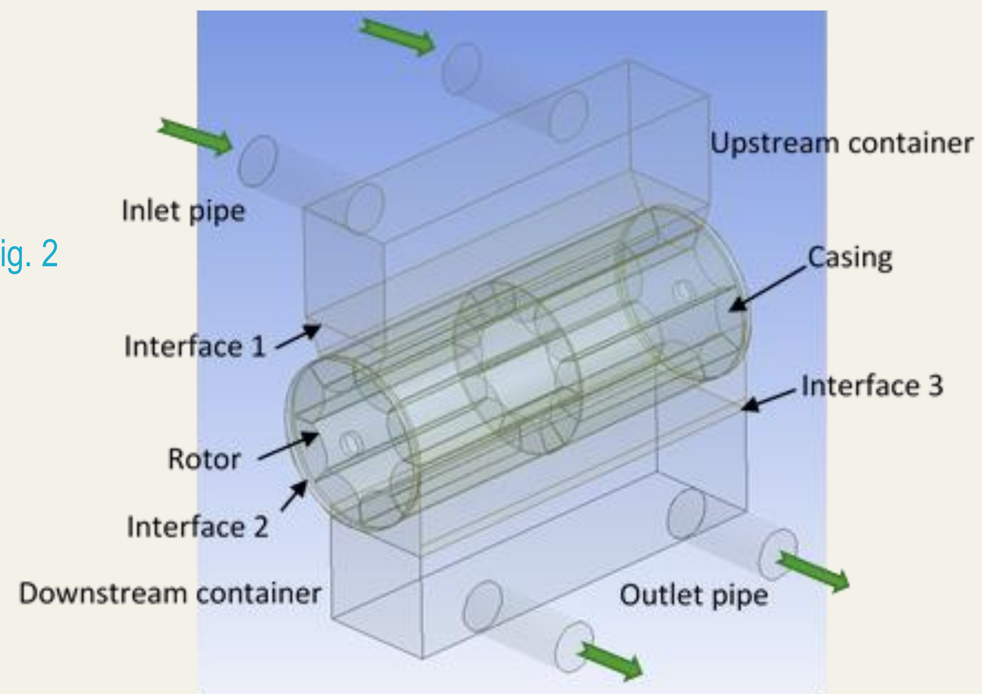
Overshot waterwheels can be traced back to the 18th century. They are more efficient in comparison with undershot and breastshot waterwheels for micro-hydro power generation with 2-10m head in hilly areas [1, 2]. Fig. 1 illustrates a typical wooden overshot waterwheel on which the work is done by the weight of water on the wheel.

Fig. 1



Scotstream Limited has developed a new design called the Hydrobox to improve performance of the overshot waterwheel and to make it compact. This design is shown in Fig. 2 and specified in the table below. We intend to study its hydraulic performance and hydrodynamics by using a CFD (computational fluid dynamics) method which is an advanced technique for solving and simulating fluid flow on a computer.

Fig. 2



Diameter(mm)		Axial length (mm)	
Rotor	Casing	Rotor	Casing
2225	2245	5648	6000

Method and Flow Model

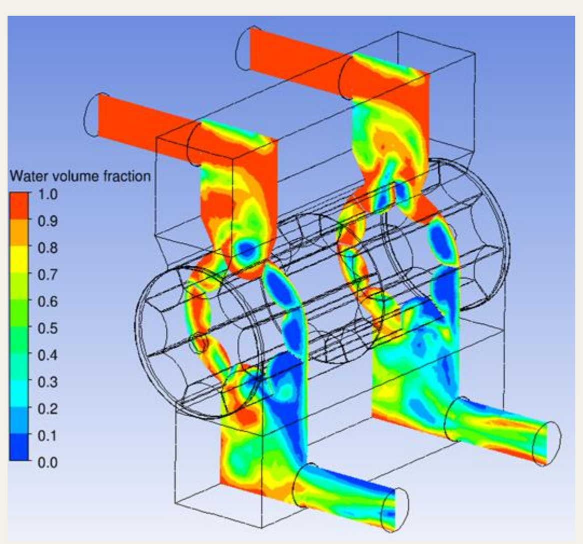
The waterwheel has four fluid domains : upstream container, stator/casing, rotor and downstream container. The latter three domains comprise unsteady, three-dimensional, water-air two-phase flows. These are solved for with the Reynolds time-averaged Navier-Stokes equations, along with the mixture multiphase flow model, and standard k-epsilon turbulence model. The domains are divided into 1.0 million mesh cells in ANSYS CFX 18.1 software. The finite volume method is used to numerically solve the equations.

The rotor fluid is rotating at 15rpm, and the flow parameters between fluid domains are coupled at interface 1, 2 and 3 in Fig. 2. At the inlets of the upstream container, flow rates of 7.93, 4 and $2m^3/s$ are specified, respectively. The time-step size is 0.02s.

Initial Results

Water volume fraction contour at the 250th time-step is illustrated in Fig. 3. It is shown that the upstream and downstream containers are too small in size to separate the water and air. The air is unable to escape from the rotor to the outside, resulting in low efficiency of the waterwheel : 0.11%, 2% and 47% at the flow rates 7.93, 4 and $2m^3/s$.

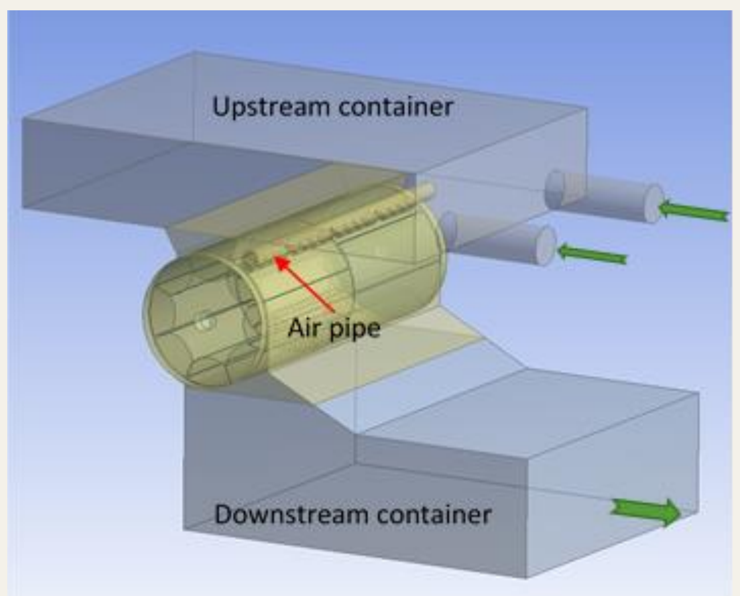
Fig. 3



First Update of the Design

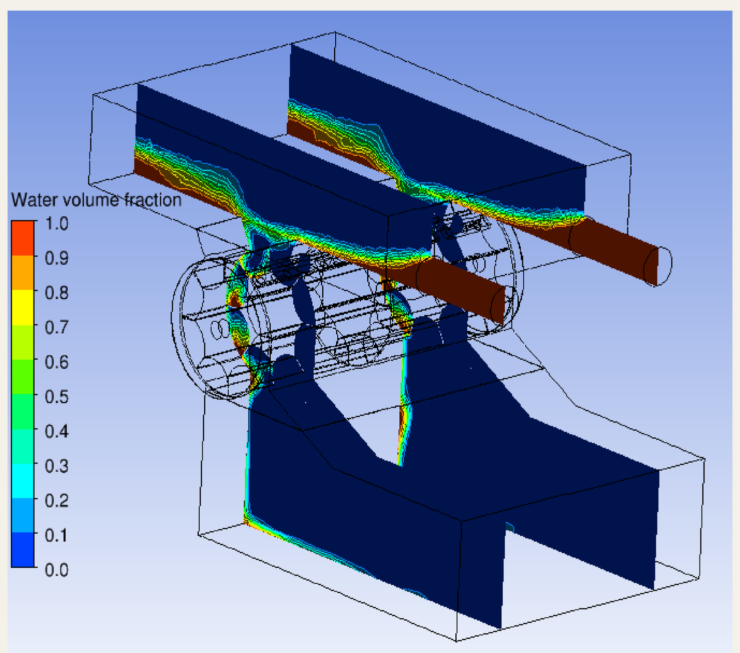
To make air separate quickly from the water, the upstream and downstream containers are updated as shown in Fig. 4. The top surface of the upstream container is opened to the air. The remaining boundary conditions remain unchanged. The flow rates are 4 and $2.5m^3/s$, respectively. Water volume fraction is shown Fig. 5 at a flow rate of $4m^3/s$.

Fig. 4



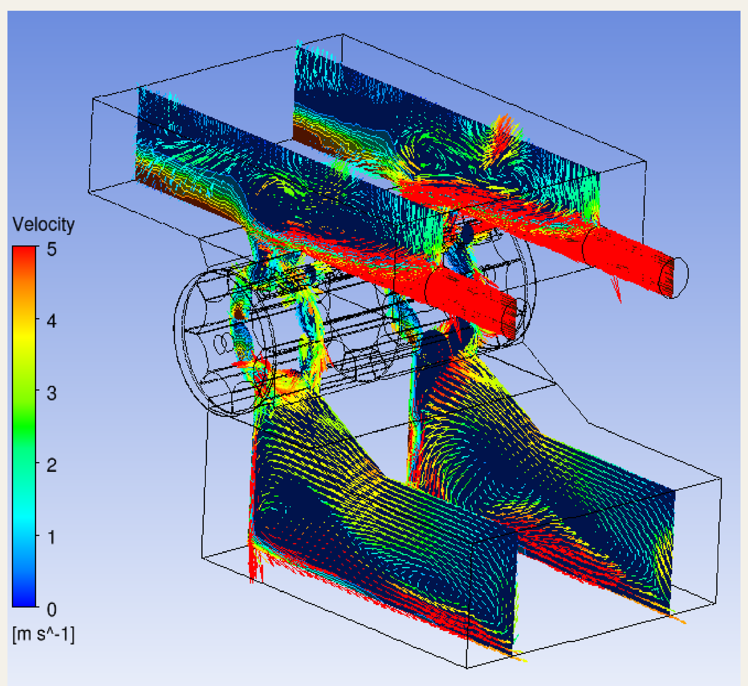
It is seen that the air has been separated completely from the water in this design, and a clear free surface is formed in both containers. As a result, the efficiencies of the waterwheel are 34% and 57% at flow rates 4 and $2.5m^3/s$..

Fig. 5



However, the two inlet pipes generate jet flows and induce turbulence in the upstream container, see Fig .6., which will affect the flow rate into the rotor of the waterwheel .

Fig. 6



Second Update of the Design

The second update of the model is shown in Fig. 7, in which the two inlet pipes are simply removed. The wall housing the pipes become the inlet, where flow rates of 6, 4 and $2.5m^3/s$ are applied, respectively. The remaining boundary conditions remain the same as for the first model update.

Fig. 7

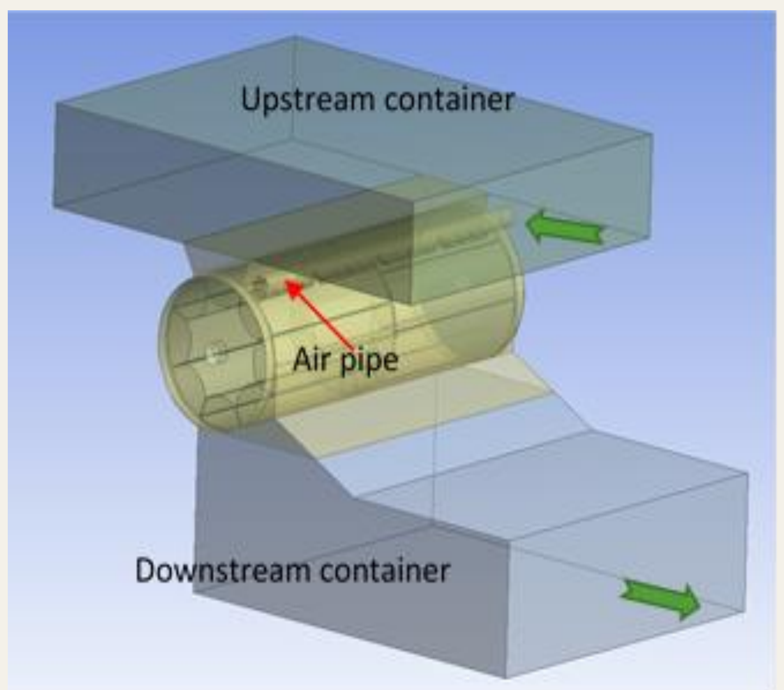
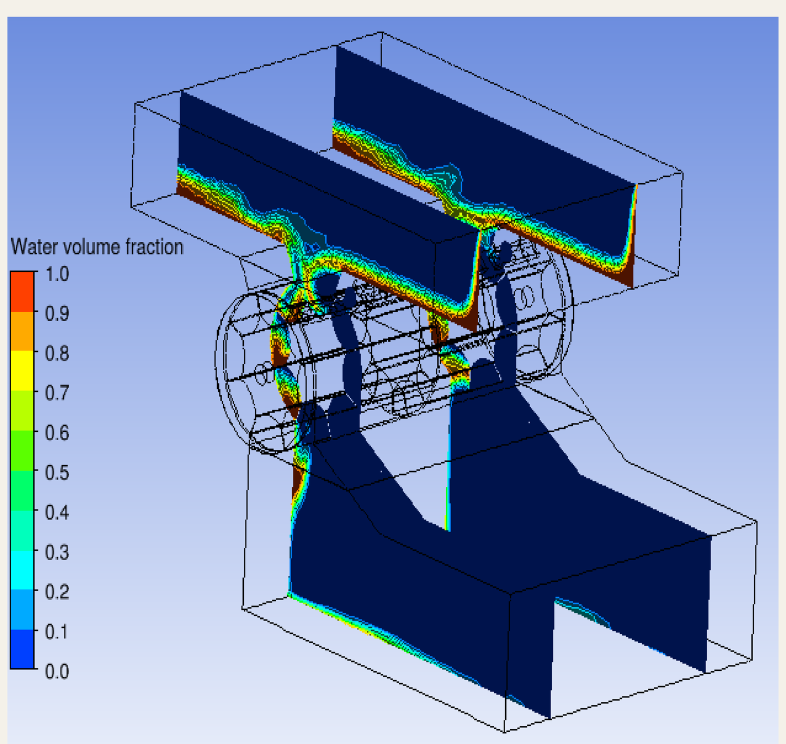
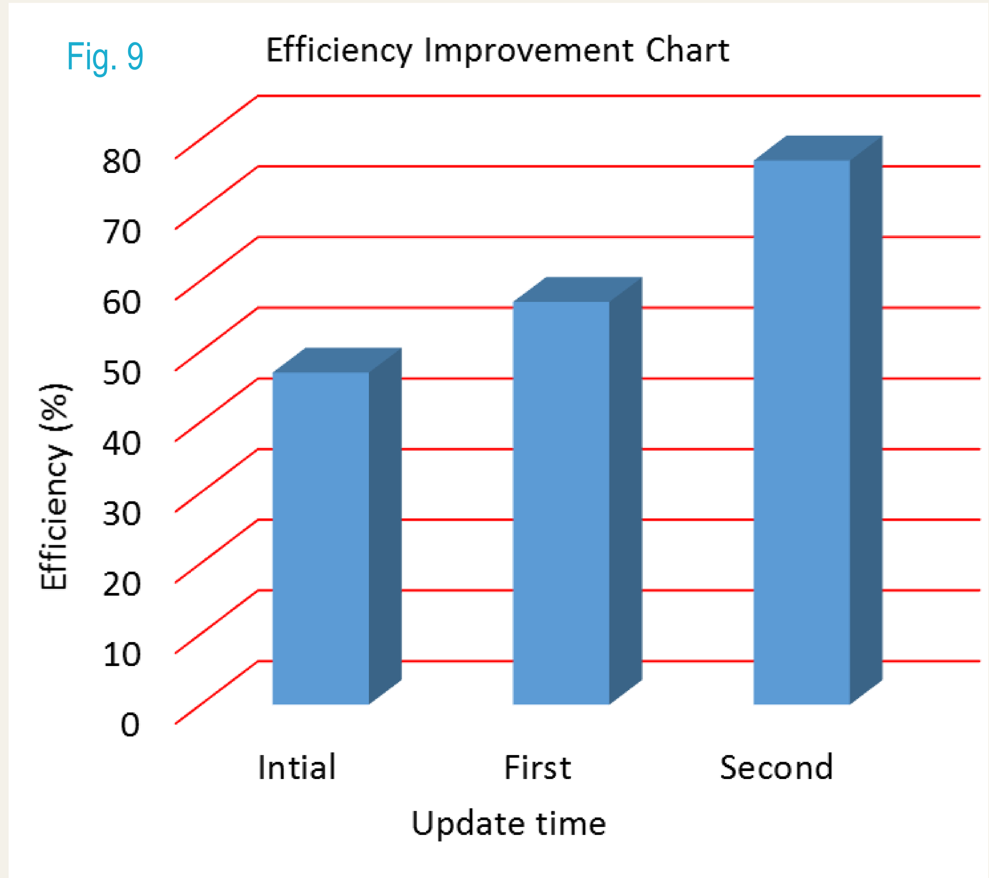


Fig. 8



Water volume fraction at a flow rate of $4m^3/s$ is presented in Fig. 8 at the 200th time-step. The jet flows in the inlet are seen to disappear and water streams smoothly into the rotor. Accordingly, the efficiencies of the waterwheel are 36%, 53% and 77%, at flow rates of 6, 4 and $2.5m^3/s$. The maximum improvement in the efficiency is 30% at $2.5m^3/s$ in comparison with the initial design. The efficiency improvement chart is demonstrated in Fig. 9 for this case.

Fig. 9



Effect of Rotor Rotational Speed

Additional simulations were conducted at 20, 25 rpm rotational speeds for the flow rate of $2.5m^3/s$ on the second update model. The effect of rotational speed is shown in Fig. 10. It is observed that an increase in speed of 67% can reduce the efficiency by 8%. 15rpm may be the optimal rotational speed.

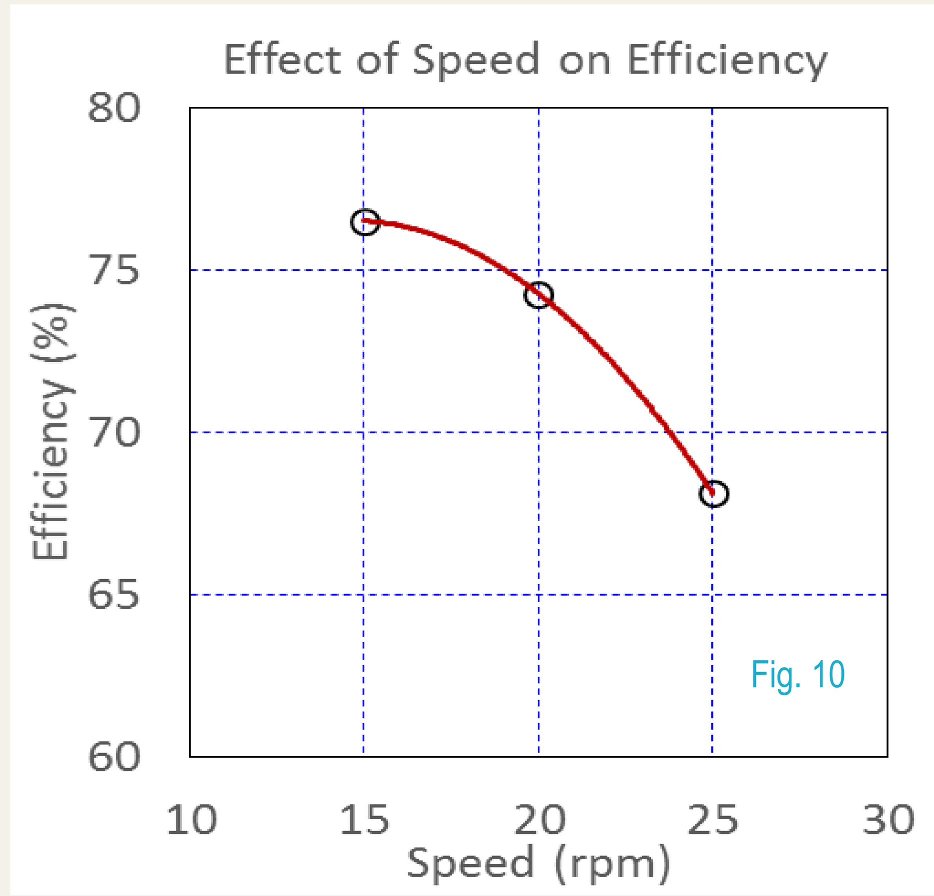


Fig. 10

Conclusions

1) Based on ANSYS CFX 18.1 CFD software, a method for modelling the water-air two phase flow with free surface and buoyancy effect in a waterwheel is established.
2) The designed configurations, i.e. upstream and downstream containers, were updated twice. As a result, the peak efficiency of the waterwheel reached 77% (from 47% in the initial design) at $2.5m^3/s$ flow rate.
3) The waterwheel performance is sensitive to rotational speed, and the performance degrades significantly when the speed is higher than 25rpm at $2.5m^3/s$ flow rate. 15rpm maybe the optimal rotational speed from hydrodynamics point of view.

Further Work

Further work includes:

(1) studies with a finer mesh,
(2) study on new design configurations, i.e. without two side covers and the appropriate casing,
(3) study on effects of the number, size and shape of buckets on the performance,
(4) study on effect of rotational speed on the performance at other flow rates.

References

[1] Capecchi D., Over and undershot waterwheels in the 18th century; science-technology controversy, Advances in Historical Studies, 2013, 2(3): 131-139.

[2] Denny M., The efficiency of overshot and undershot waterwheels, European Journal of Physics, 2004, 25: 193-202.

* Kiran.Ramesh@glasgow.ac.uk