Pelton turbine bucket flow analysis and visualization for evaluation of area-wise contribution

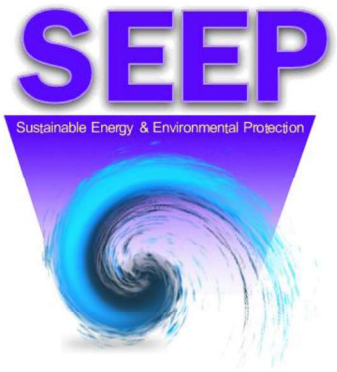
Bhattarai, S.; Vichare, P.; Dahal, K.

Published in:
Renewable and Sustainable Energy Developments Beyond 2030

Published: 08/05/2018

Document Version
Publisher's PDF, also known as Version of record


SEEP
Sustainable Energy & Environmental Protection
UWS UNIVERSITY OF THE WEST of SCOTLAND
Institute of Engineering and Energy Technologies
RENEWABLE AND SUSTAINABLE ENERGY DEVELOPMENTS BEYOND 2030
VOLUME 3
SEEP CONFERENCE 2018
University of the West of Scotland, Paisley Campus
Tuesday 8-Friday 11 May 2018



Renewable and Sustainable Energy Developments Beyond 2030

VOLUME 3

Proceedings of the 8th International Conference on Sustainable Energy & Environmental Protection

EDITED BY:

Prof Abdul Ghani Olabi

University of the West of Scotland

School of Engineering & Computing

Institute of Engineering & Energy Technologies

"Renewable and Sustainable Energy Developments Beyond 2030

VOLUME 3"

First Published in 2018
by
University of the West of Scotland
High Street, Paisley, PA1 2BE
UK

ISBN: 978-1-903978-62-7

Table of Contents

Air quality

Battery and Energy Storage

Energy Development and Storage

PELTON TURBINE BUCKET FLOW ANALYSIS AND VISUALIZATION FOR EVALUATION OF AREA-WISE CONTRIBUTION

S. Bhattarai, P. Vichare and K. Dahal
School of Computing and Engineering, University of the West of Scotland, Paisley, UK; email: suyesh.bhattarai@uws.ac.uk, parag.vichare@uws.ac.uk, keshav.dahal@uws.ac.uk.

ABSTRACT

The objective of this paper is to visualize, evaluate and analyse the flow in a Pelton turbine bucket. The numerical study of the buckets in rotating condition allows for detailed study and analysis of the jet-bucket interaction from the moment of impingement to the moment of exit. Unsteady numerical simulations, based on two phase homogenous model, was performed for wall pressure measurement and flow visualization. Realizable K-epsilon viscous model with near wall treatment was used for turbulence modelling. Similar studies have been conducted for the inner bucket surface and are limited to verifying the experimental and numerical results. This study extends the research to the chronologic performance of specific segments inside the bucket; on the reverse side of the bucket and the edges. It attempts to expose the most and the least efficient parts of the bucket so that the designers can contemplate better designs to enhance the performance. The research suggests that the splitter area and the central base region of the bucket are the most productive whereas the portion at the root; and at the outer periphery is the least productive.

Keywords: Pelton turbine, Computational fluid dynamics, Flow analysis, Design analysis.

1 INTRODUCTION

Pelton turbine is a technology that has been used in simple form as water wheels for centuries. But due to its complex water jet interactions, detailed study on energy transfer has not been possible. The basic working principle of the Pelton turbine makes it susceptible to bucket reverse side splashing, atomization of the jet, interference of water layers, secondary flow within the jet and gravity deviation of the water etc. which play a major role in energy loss in the turbine [1]. In such conditions, obtaining accurate measures of energy that can be generated from the turbine based on analytical methods alone is a humungous task. The CFD methods utilize the processing capabilities of the computers that can help study the water-bucket interaction in detail. This detailed evaluation and visualization allows for inspection of chronological contribution of the different areas of the Pelton bucket.

This paper presents the CFD results of Pelton bucket flow analysis with respect to time for each of the contribution areas identified in the bucket. The second section presents related work in this area, and the third section details the methodology used for conducting the simulation which is rarely available in other literature. The fourth section provides the results and critical discussion based on the results while the fifth section provides the conclusion for the entire study.

2 RELATED WORK

The literature available on Pelton turbine performances ranges from investigating the effect of a single bucket feature such as the number of buckets [1] and the angle of attack on the bucket [2] or the length, width, depth, height of the splitter, angle of the splitter, angle of exit, etc. [3] to the effect of the bucket as a whole. There has been a rise in research after the CFD capabilities were proven for free surface flows by the end of the twentieth century. Avellan et. al. in [4] verified the performance of the Volume of Fluid approach used in some of the commercial purpose CFD codes to predict the flow in injectors and bucket geometry of a Pelton turbine.

The environment definition with rotating buckets is more accurate and realistic representation of the problem. But it requires excessive computational capacity and time. In most studies, this computational resource requirement problem has been eased by modelling only three successive buckets [5] or using alternate less accurate methods [6]. The bucket at the middle undergoes the complete cycle of jet entering the bucket and exiting it along with the splashing effects from the previous bucket; hence, it encompasses the all-inclusive cycle of any bucket.

Perrig et al. [7] measured the pressure in the inner surfaces of the buckets using two phase homogenous model. The resultant patterns had a qualitative agreement and a variation of 7-26% was observed in experimental and numerical results. From in-depth analysis, it was concluded that the area near the cut-out

contributed most for the bucket power and the region at the root of the bucket was least productive. This study did not study the edges and the reverse surface of the bucket.

3 METHODOLOGY

Computational fluid dynamics analysis has various aspects for a successful outcome. The different stages and requirements have been highlighted in the subsequent sub-sections.

3.1 Geometric Modelling

Existing design of the Pelton turbine which has been used by many researches as the benchmark was modelled with commercial CAD software and imported into Ansys for pre-processing. The basic details of the turbine assembly are as listed in table 1. Turbine with same features has been used in the studies [3, 8].

Table 1. Basic features of the Turbine [8]

Conditions	Values
Runner Pitch Diameter	400 mm
Jet Diameter	31 mm
Rotation Speed	1150 rpm
Flowrate (nominal)	270.6 m3/h
Net head	129.6 mWG
Buckets	22
Injectors	1
Power Output (2x nominal flowrate)	80.14 Kw
Torque (2x nominal flowrate)	761.94 Nm

The turbine used by Stamatelos [8] was recreated to be analysed as it provides the detailed geometry and also the experimental results hence, we could replicate the same experiment. These results provide benchmark for validation of the simulation results obtained.

3.2 Environment and Mesh

For further processing only a sector with 4 consecutive buckets were taken with the casing and the injector manifold. This allows reduction of the computational domain and ensures faster runtimes for the simulation and smaller data size.

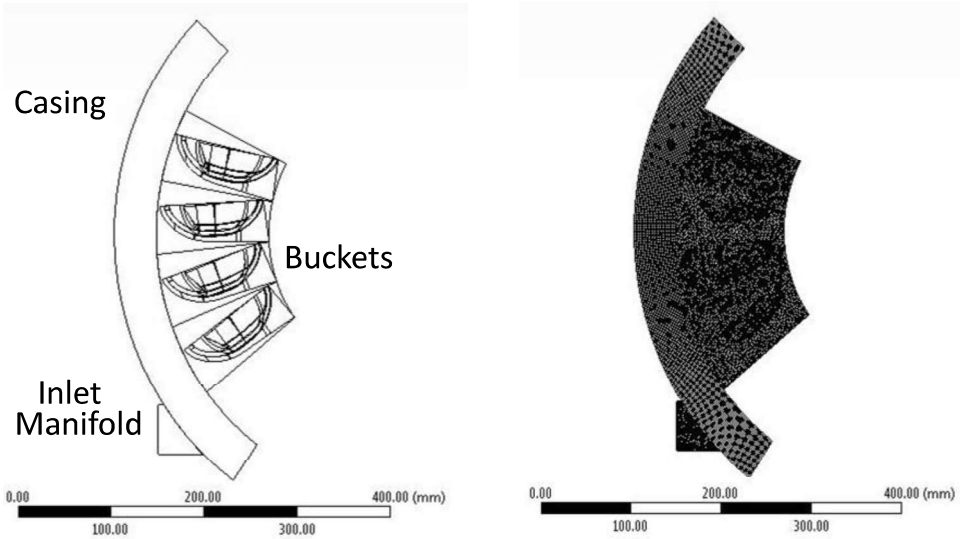


Figure 1. Computational Domain and Mesh

Mesh for the computational domain was generated using Ansys Meshing and best efforts were made to ensure the quality of the mesh. Basic settings used for meshing was of CFD Physics preference and Fluent Solver preference with Fine relevance centre and Element size of 2mm. Patch Independent Tetrahedral Mesh was used for meshing the buckets as it was demanding to generate a Hexahedral Mesh for an irregular surface such as a bucket profile. Table 2 shows the element count for the mesh and the various attributes of the mesh that determines its quality and in turn its accuracy.

Table 2. Mesh Attributes

Parameters	**Values**		
	Min	Max	Average
Nodes	409306		
Elements	934034		
Aspect Ratio	1.0041	46.678	1.6851
Element Quality	0.0296	0.9999	0.8691
Orthogonal Quality	0.0398	0.9997	0.8861
Skewness	0.00001	0.96636	0.1850

The number of nodes was limited as Academic Researcher Licence of Ansys Fulent was being used. The mesh attributes have been kept well in check with great difficulty. The Aspect ratio and Skewness although acceptable was very difficult to handle as the environment had a lot of curved surfaces.

3.3 Solver

After the pre-processing, the generated mesh was loaded in the Fluent solver environment. The Pressure Based Transient solver was chosen with the Multiphase Volume of Fluid model and Realizable K-epsilon viscous model as basic settings for Fluent environment. The details of the solver scheme are listed below in table 3.

Table 3. Fluent Settings

Description	**Mode**
Solver Type	Pressure Based
Time	Transient
Model	Multiphase Volume of Fluid
Viscous Model	Realizable K-epsilon
Scheme	Simplec
Spatial Discretization Gradient	Least Squares Cell Based
Pressure	PRESTO!
Momentum	Second Order Upwind
Time Stepping Method	Fixed
Time Step Size	2e-5 s
Number of Time Steps	400

Fixed time stepping with the time step size of 2e-5s was used to keep the Cournat Number in check. The boundary conditions were defined in such a way that it resembled the actual working conditions. The net head was used as the gauge pressure for the inlet and the inlet flow rate for primary phase Air, was set at zero as only water flows through the inlet.

3.4 Boundary Conditions

Boundary conditions as listed in table 4 were applied to replicate the realistic working environment.

Table 4. Boundary Conditions

Parameters	**Details**
Net head	129.6 mWG
Inlet Water Flow Rate	37.9491 kg/s
Inlet Air Flow Rate	0 kg/s
Turbine RPM	120.427 rad/s
Rotation Axis	Z
Outlet	Atmospheric Pressure Outlet
Walls	Movement relative to adjacent cell zone

4 RESULTS AND DISCUSSION

To study the impact of different parts of the bucket surface to the overall power output, the bucket surface was divided into 27 different zones. There were 9 zones on the inside of the bucket, 7 zones on the edges, 10

zones on the back and the cut-out. This division made it easier to obtain localized data for the zones and evaluate their overall performance. The division of zones is illustrated in figure 2.

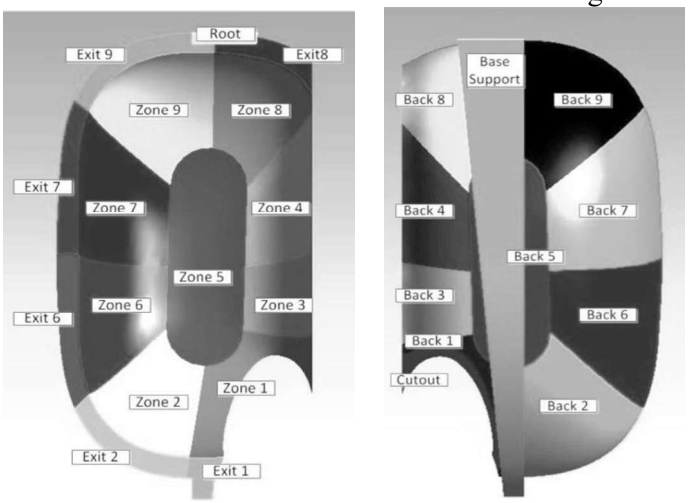


Figure 2. Zone division on bucket surface

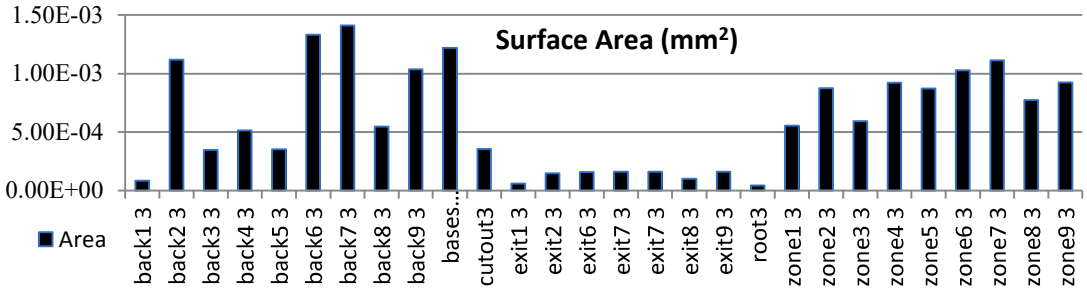


Figure 3. Surface area comparison of different zones

All the zones do not represent equal surface area hence, the surface area of each zones was measured to evaluate the total force experienced by each zone. The area represented by each zone is illustrated in figure 3. The naming convention followed for each zone is the zone name followed by the bucket number. Back1 3 means it is the zone back1 of bucket number 3. Calculations have been made only for this bucket as it undergoes the entire cycle of back splashing to the jet leaving the bucket.

4.1 Inner Surface

The pressure readings and the respective forces generated with respect to time are represented in the graphs below. The first set of graph represents the pressure respective forces on the internal zones of the bucket profile. The readings show that Zone 5 and Zone 3 are the most productive zones on the bucket surface. Zone 2 and Zone 8 are the least productive regions of the bucket as they experience the least pressure from the jet and hence exert the least force and torque to the overall system. Other internal zones are moderately productive.

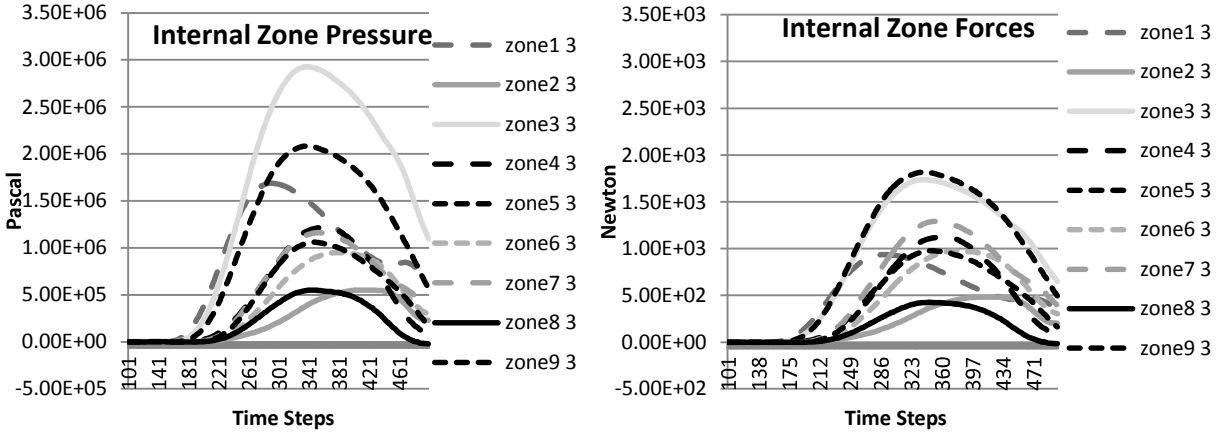


Figure 4. Pressure and Forces on Internal Zones

4.2 Reverse Surface

The back surfaces collectively share more surface area than the internal zones. These zones are expected to experience minimal force on them. But Back 7 and Back 9 experience some considerable amount of force on them which acts the counter force to the system. This force should be kept to a minimum to ensure maximum productivity of the system. This is because of splashing effect from the jet that was reflected from the preceding bucket.

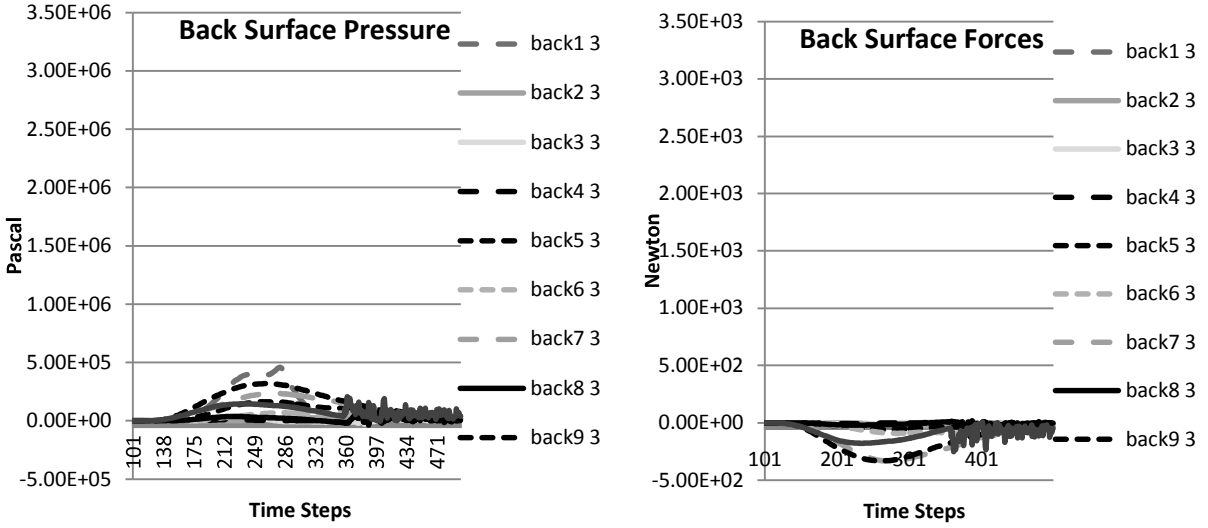


Figure 5. Pressure and Forces on Back Surfaces

4.3 Edges and Cut-out

The next set of graphs represents the readings for the cut-out and the exits. These are the borderline regions that do not cover a wide area. The cut-out is the portion where the jet first interacts with the bucket and although it represents a small area, it experiences considerable pressure and adds to the total force generated. It is almost as productive as the zones 2 and 8 although it has much lower surface area. But the force generated is for a smaller time period as the jet does not interact with this surface for a longer time. Also, effort must be made to reduce this surface area as more interaction with this area leads to less interaction with the internal zones which are more productive than the cut-out.

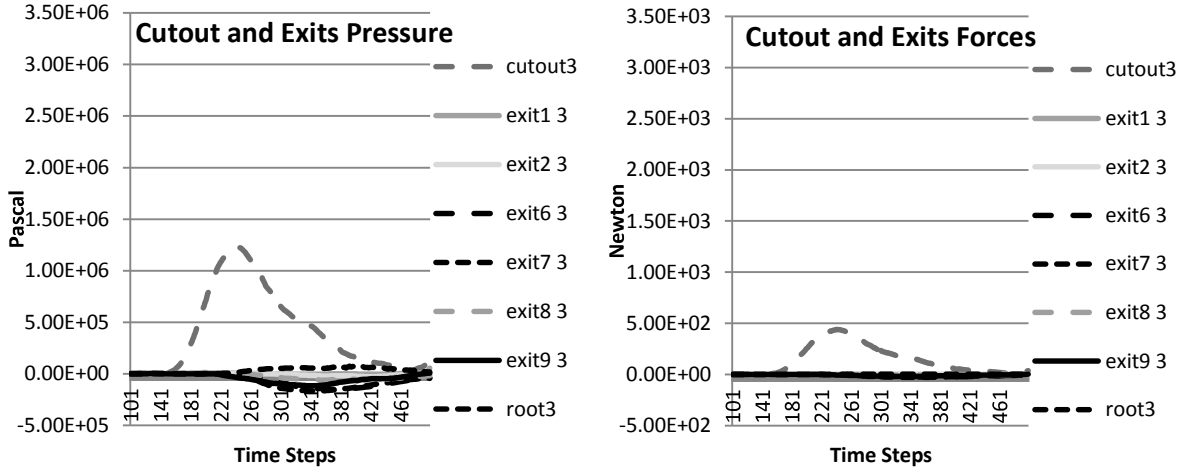


Figure 6. Pressure and Forces on Cut-out and Exits

The above analysis reveals that there are some zones on the surface of the bucket such as Zone 2, Zone 8, the Cut-out, Back 7 and Back 9 that need special attention while considering changes in design for optimization. Also, since the Exit areas do not seem to be adding much to the productivity, using much narrower exits can be considered to reduce the material required for fabrication. This would help reduction of the weight of the wheel and its material cost.

Fluent is an Eulerian method based solver for CFD problems. Although it has high accuracy, it requires long time and high computational power to obtain the final solution. This simulation took roughly 48 hours to

complete not accounting for the geometric modelling and pre-processing part. Following table shows the comparison of results obtained from experiments and the simulation conducted. The accuracy of 0.46% achieved is of a very high order although such simulations usually have an error range of 10-12%. This calculation validates the approach used in this study.

Table 5. Experimental and Simulated results

Parameters	**Values**
Input Power to turbine	91.93 kW
Power Output (Analytical)	80.14 kW
Torque (Fluent)	**766 Nm**
Torque (Analytical)	761.94 Nm
Error	**0.4699%**

5 CONCLUSION

This study was conducted to research the chronological contribution of different zones in the Pelton bucket with a view to identify the most productive and the least productive regions of the bucket. The simulation results show that the inner zones 3 and 5 are the most productive zones. The inner zones 2 and 8 are the least productive as they have similar force contribution as the cut-out which only has half their surface area. Hence, future studies could be concentrated on increasing the areas with profile similar to inner zones 3 and 5; and reducing areas with profile similar to inner zones 2 and 8. The reverse surface zones 7 and 9 experience reverse surface splashing which can be reduced by adjusting the exit angle at the bucket zones 7 and 9. Incorporating these changes to the design of the existing bucket could enhance its efficiency. The simulation conducted in this study has been validated with the experimental results and analytical calculations for same turbine in same flow conditions. The total torque measurement had an error of 0.4699% which confirms the authenticity of the simulations carried out. This study provides insight for the researchers on the contribution of the various zones in the Pelton bucket and provides suggestions for improvement of the efficiency of the bucket by making some design changes.

ACKNOWLEDGEMENTS

The first author likes to acknowledge the support provided by the EU Erasmus Mundus project SmartLink (552077-EM-1-2014-1-UK-ERA) to carry out this research at the University of the West of Scotland, UK.

REFERENCES

[1. Zidonis, A. and G. Aggidis, *Pelton turbine: identifying the optimum number of buckets using CFD.* Journal of Hydrodynamics, 2015.
2. Shogenji, K. and T. Inada, *On the Inclination of the Ridge of Buckets on a Pelton Wheel.* Memoirs of the College of Engineering 1927. **4**(7): p. 339-352.
3. Zidonis, A., et al., *Parametric Optimization of Two pelton Turbine Runner Designs Using CFD.* Journal of Hydrodynamics, 2014. **27**(3): p. 403-413.
4. Avellan, F., et al. *Flow calculations in Pelton Turbines, Part 2: Free Surface Flows*. in *Proceedings of the 19th IAHR Symposium*. 1998. Singapore.
5. Rossetti, A., et al., *Influence of the Bucket Geometry on the Pelton Performance.* Journal of Power and Energy - Proceedings of the institution of Mechanical Engineers, 2014. **228**: p. 33-46.
6. Bhattarai, S., P. Vichare, and K. Dahal, *Performance Comparison of Adapted Delaunay Triangulation Method over NURBS for Surface Optimization Problems*, in *The 31st Annual European Simulation and Modelling Conference*, P.J.S. Goncalves, Editor. 2017, EUROSIS-ETI: Lisbon. p. 76-80.
7. Perrig, A., et al., *Flow in a Pelton Turbine Bucket: Numerical and Experimental Investigations.* Journal of Fluids Engineering, 2006. **128**: p. 350-359.
8. Stamatelos, F.G., J.S. Anagnostopoulos, and D.E. Papantonis, *Performance Measurements on a Pelton Turbine Model* Journal of Power and Energy - Proceedings of the institution of Mechanical Engineers, 2010: p. 351-363.

SEEP Conference 2018

University of the West of Scotland, Paisley Campus
Tuesday 8 – Friday 11 May 2018

UWS UNIVERSITY OF THE WEST of SCOTLAND

Institute of Engineering and Energy Technologies

RENEWABLE AND SUSTAINABLE ENERGY DEVELOPMENTS BEYOND 2030

VOLUME 3

ISBN: 978-1-903978-62-7