



**PERFORMANCE AND OPTIMIZATION OF SAVONIUS, DARRIEUS, AND
COMBINED SAVONIUS-DARRIEUS HYDROKINETIC TURBINES IN
LOW-SPEED FLUID FLOW**

By

KOMSAN TANTICHUKIAD

**Thesis Submitted to the School of Graduate Studies, Universiti Putra Malaysia,
in Fulfilment of the Requirements for the Degree of Doctor of Philosophy**

February 2024

FK 2024 29

All material contained within the thesis, including without limitation text, logos, icons, photographs and all other artwork, is copyright material of Universiti Putra Malaysia unless otherwise stated. Use may be made of any material contained within the thesis for non-commercial purposes from the copyright holder. Commercial use of material may only be made with the express, prior, written permission of Universiti Putra Malaysia.

Copyright © Universiti Putra Malaysia



Abstract of thesis presented to the Senate of Universiti Putra Malaysia in fulfilment
of the requirement for the degree of Doctor of Philosophy

**PERFORMANCE AND OPTIMIZATION OF SAVONIUS, DARRIEUS,
AND COMBINED SAVONIUS-DARRIEUS HYDROKINETIC TURBINES IN
LOW-SPEED FLUID FLOW**

By

KOMSAN TANTICHUKIAD

February 2024

Chairman : Professor Ir. Azmi bin Dato Haji Yahya, PhD

Faculty : Engineering

Most wetland rice cultivation schemes in Malaysia and Thailand were designed and built with good irrigation and drainage canal networks for the benefit farming activities. These canals have ample volume of running water all year round and could be harnessed for hydropower. Energy generation from small hydropower plants has been increased. Hydropower could be generated by converting the kinetic energy of flowing water into mechanical power through the use of turbines. Vertical hydrokinetic turbines, either Savonius or Darrieus, have their own specific operational ranges and their pros and cons. Thus, the combination of Savonius-Darrieus turbines could overcome the weakness and complement the strength of them. This study was conducted to optimize separately the design of Savonius hydrokinetic turbine (SHKT) and the Darrieus hydrokinetic turbine (DHKT) with the ultimate objective of getting an optimum combined Savonius-Darrieus hydrokinetic turbine (SDHKT) design.

ANSYS Fluent was used to optimum design configurations for the individual SHKT- and DHKT-models based on the typical flow conditions in canals. Validation tests on the developed computational approach against the published work of Saad et al., 2020 showed only 1.85% differences of $c_{p,max}$. The c_p of a two-stage model with full-shaft and a constant-stage aspect ratio was higher than a single-stage-model and a two-stage-model with a different stage-aspect ratio. To evaluate and optimize the adopted new designs of the delta-shaped blade DHKT with NACA0012 hydrofoils. Simulations were done for 2-blade models with four designs (*MD1-MD4*) of varying blade configurations (in-plane- and twisted-blades) and cross-sectional areas (reduced and constant areas). Validation tests on this developed computational method against the experimental work of Doan et al., 2020 on the power coefficient curve showed good agreement with deviations of less than 1.25%. The optimum designs were models with twisted blades and reduced and constant cross-sectional areas (*MD3* and *MD4*). The 3-bladed models with similar blade characteristics were tested and compared with the 2-bladed models. The 2-bladed models performed better during the higher range of tip speed ratio (λ), whereas the 3-bladed models were outstanding at the lower range. Overall, the *MD4*-model was the most appropriate design to operate under the specified conditions.

To optimize the design configurations of the combined SDHKT-model, the optimum designs for SHKT and DHKT-models were taken as the fundamental basis. A 2-stage-SHKT model was placed at the center and inside the *MD3* and *MD4*. The attachment angles (φ) and ratios of swept area (R_{sa}) were investigated using similar computational approaches employed in the earlier DHKT simulations. Validation tests on the developed computational approach against the laboratory work of Sahim et al., 2013 showed good agreement with a maximum deviation value of about 1.5%. The $c_{p,max}$ of

$\varphi = 0^\circ$ was the highest when λ equaled 1 and higher by 0.02% and 0.88% when $c_{p,max}$ of $\varphi = 20^\circ$ and 30° . Besides, the combined SDHKT-models with $\varphi = 0^\circ$ and three ratios of $R_{sa1} = 0.4$, $R_{sa2} = 0.3$, and $R_{sa3} = 0.2$ were simulated. The R_{sa1} performed better when λ was lower than 0.9. Beyond the value of 0.9, the performances of the R_{sa3} were higher than the rests.

Keywords: Attachment angle, Combined Savonius-Darrieus hydrokinetic turbine, Delta-shaped-blade Darrieus Hydrokinetic turbine, Ratio of swept area, Savonius Hydrokinetic turbine

SDG: GOAL 4: Quality Education, GOAL 7: Affordable and Clean Energy

Abstrak tesis yang dikemukakan kepada Senat Universiti Putra Malaysia sebagai memenuhi keperluan untuk ijazah Doktor Falsafah

**PRESTASI DAN PENGOPTIMUM TURBIN HIDROKINETIK SAVONIUS,
DARRIEUS DAN GABUNGAN SAVONIUS-DARRIEUS DALAM ALIRAN
BENDALIR HALAJU RENDAH**

Oleh

KOMSAN TANTICHUKIAD

Februari 2024

Pengerusi : Professor Ir. Azmi bin Dato Haji Yahya, PhD

Fakulti : Kejuruteraan

Kebanyakan skim tanaman padi basah di Malaysia dan Thailand direkabentuk dan dibina dengan rangkaian taliair pengairan dan penyaliran yang terbaik untuk memberi faedah kepada aktiviti pertanian pesawah. Taliair ini mempunyai isipadu aliran air yang mencukupi disepanjang tahun yang membolehkan penjanaan kuasa hidro. Kuasa hidro dijana dari turbin yang dipasang di taliair untuk menukar tenaga kinetik arus aliran air kepada kuasa mekanikal. Penjanaan tenaga dari kuasa hidro telah meningkat secara mendadak di peringkat global disebabkan. Turbin hidrokinetik menegak, samada Savonius atau Darrieus, masing-masing mempunyai julat pengendalian, kebaikan serta keburukan tertentu. Kombinasi antara turbin Savonius dengan Darrieus mampu mengatasi kelemahan disamping melengkapkan kekuatan kedua turbin ini. Kajian ini dilaksanakan untuk mengoptimumkan secara berasingan rekabentuk turbin Savonius (SHKT) dan rekabentuk turbin hidrokinetik Darrieus (DHKT) dengan

objektif muktamad untuk mendapat rekabentuk turbin hidrokinetik tergabung Savonius-Darrieus (SDHKT) yang optimum.

ANSYS Fluent telah digunakan bagi mendapat konfigurasi rekabentuk yang optimum untuk model SHKT dan DHKT berdasarkan kepada keadaan aliran taliair lazim. Ujian validasi terhadap kaedah pengkomputaran yang telah dibangunkan ini dengan kajian berangka oleh Saad et al., 2020 menunjukkan hanya 1.85% perbezaan bagi $c_{p,max}$. Nilai c_p bagi model beraci-penuh dan dua-peringkat dengan nisbah aspek peringkat malar adalah lebih tinggi dari model satu-peringkat dan model dua-peringkat dengan nisbah aspek peringkat yang berbeza.

ANSYS Fluent digunakan lagi untuk menilai dan mengoptimum reka bentuk terbaru DHKT dengan NACA0012-hidrofoil. Simulasi dilaksanakan untuk Model 2 bilah dengan empat rekabentuk (*MD1-MD4*) dengan konfigurasi bilah (bilah satu satah dan terherot) dan luas rentangan yang berbeza (luas terkurang dan malar). Ujian validasi keatas kaedah berkomputer yang telah dibangunkan ini dengan kajian uji kaji oleh Doan et al., 2020 mengenai lengkung pekali kuasa menunjukkan persamaan yang baik dengan sisihan kurang dari 1.25%. Reka bentuk optimum adalah dengan bilah terherot dan luas rentangan malar. Model 3-bilah dengan ciri-ciri bilah yang sama telah diuji and dibandingkan dengan model 2 bilah. Model 2-bilah berfungsi lebih baik pada nisbah laju hujung (λ) pada julat tinggi, manakala model 3 bilah adalah lebih baik pada julat bawah. Secara keseluruhannya, model-*MD4* telah menunjukkan rekabentuk yang paling sesuai untuk beroperasi di bawah keadaan yang telah dinyatakan.

Bagi tujuan untuk mengoptimumkan konfigurasi rekabentuk model turbin tergabung SDHKT, rekabentuk optimum bagi kedua2 model optimum SHKT dan DHKT diambil sebagai asas pertimbangan. Model SHKT 2-peringkat diletakkan di bahagian pusat di

dalam MD3 and MD4 bagi model DHKT. Sudut cantuman (φ) dan nisbah luas sapuan (R_{sa}) dikaji dengan menggunakan sepertimana pendekatan pengkomputeran yang digunakan dalam simulasi sebelum dengan DHKT. Ujian validasi keatas pendekatan pengkomputeran yang dibangunan dengan kajian makmal oleh Sahim et al., 2013 menunjukkan persamaan yang terbaik dengan nilai sisihan maksimum sebanyak 1.5% sahaja. $c_{p,max}$ bagi $\varphi = 0^\circ$ adalah tertinggi apabila λ bersamaan 1 dan tinggi sebanyak 0.02% dan 0.88% apabila $c_{p,max}$ bagi $\varphi = 20^\circ$ dan 30° . Selain itu, model tergabung SDHKT dengan $\varphi = 0^\circ$ dan tiga nisbah $R_{sa1} = 0.4$, $R_{sa2} = 0.3$, dan $R_{sa3} = 0.2$ telah disimulasikan. R_{sa1} berprestasi lebih baik apabila λ adalah rendah daripada 0.9. Melebihi nilai 0.9, prestasi model dengan R_{sa3} adalah lebih tinggi daripada yang lain.

Kata Kunci: Sudut cantuman, Turbin hidrokinetik tergabung Savonius-Darrieus, Bilah berbentuk delta turbin hidrokinetik Darrieus, Nisbah luas sapuan, Turbin hidrokinetik Savonius

SDG: MATLAMAT 4: Pendidikan Berkualiti, MATLAMAT 7: Tenaga Berpatutan dan Bersih

ACKNOWLEDGEMENTS

First and foremost, I would like to give thanks to my supervisor and all members of my supervisory committee for their continuous support. I am grateful for their insightful suggestion, and recommendations on my research.

To my friends in Universiti Putra Malaysia especially PJ and Serdang Angle Rugby friends who always accompany me, help me, support me, and make my studying life abroad in Malaysia easier and more wonderful.

Many thanks go to Encik Mohd Hafizul Hashim, a technical staff of the CAD/CAM Laboratory, Faculty of Engineering, Universiti Putra Malaysia for helping me to utilize equipment there.

This research was supported by the Southeast Asian Regional Center for Graduate Study and Research in Agriculture (SEARCA). As a SEARCA scholar, I would also like to extend my sincere thanks for your kind support.

This thesis was submitted to the Senate of Universiti Putra Malaysia and has been accepted as fulfilment of the requirement for the degree of Doctor of Philosophy. The members of the Supervisory Committee were as follows:

Azmi bin Dato Haji Yahya, PhD

Professor Ir
Faculty of Engineering
Universiti Putra Malaysia
(Chairman)

Azmin Shakrine bin Mohd Rafie, PhD

Associate Professor
Faculty of Engineering
Universiti Putra Malaysia
(Member)

Anas bin Mohd Mustafah, PhD

Senior Lecturer
Faculty of Engineering
Universiti Putra Malaysia
(Member)

Ahmad Suhaizi bin Mat Su, PhD

Senior Lecturer
Faculty of Agriculture
Universiti Putra Malaysia
(Member)

ZALILAH MOHD SHARIFF, PhD

Professor and Dean
School of Graduate Studies
Universiti Putra Malaysia

Date: 10 October 2024

TABLE OF CONTENTS

	Page
ABSTRACT	i
ABSTRAK	iv
ACKNOWLEDGEMENTS	vii
APPROVAL	viii
DECLARATION	x
LIST OF TABLES	xv
LIST OF FIGURES	xvii
LIST OF ABBREVIATIONS	xxiv
 CHAPTER	
 1 INTRODUCTION	 1
1.1 Research Background	1
1.2 Problem Statement	4
1.3 Research Aims and Objectives	5
1.4 Scope of Research Work	6
1.5 Thesis overview	7
 2 LITERATURE REVIEW	 9
2.1 Introduction	9
2.2 Classification of Hydrokinetic Turbines	12
2.3 Turbine Selections	15
2.4 Principal of Hydrokinetic Energy	18
2.4.1 Turbine Electrification	18
2.4.2 Coefficient of Performances	19
2.5 Fundamental of Computational Fluid Dynamics (CFD)	22
2.5.1 Governing Equations	23
2.5.2 Turbulence Model	24
2.5.3 Domain Independent Study	29
2.5.4 Mesh Sensitivity Analyses	34
2.5.5 Time Step Independency Study	38
2.6 Savonius Hydrokinetic Turbine	39
2.6.1 Effect of Bucket Number	41
2.6.2 Effect of Shape of Bucket Profile	43
2.6.3 Effect of End Plates	46
2.6.4 Effect of Aspect Ratio	49
2.6.5 Effect of Overlap Ratio	51
2.6.6 Effect of the Number of Stages	56
2.6.7 Influence of Reynolds Number and Turbulence Intensity	63

2.7	Darrieus Hydrokinetic Turbine	66
2.7.1	Effect of Airfoil Profiles	69
2.7.2	Effect of Bladed Shapes	72
2.7.3	Effect of Blade Number	73
2.7.4	Effects of Solidity	77
2.7.5	Effects of Aspect Ratio	79
2.7.6	Effect of Pitch Angle	80
2.8	Combination of Hydrokinetic Turbine	86
2.8.1	Effect of the Compact Design	87
2.8.2	Effects of the Radius Ratio	90
2.8.3	Effect of the Attachment Angle	93
2.8.4	Effect of the Rotational Velocity	96
2.9	Summary and Research Gap	97
3	METHODOLOGY	99
3.1	Introduction	99
3.2	Turbine Design and Optimization Processes	100
3.3	CAD Design	103
3.4	CFD Optimization Procedure	104
3.5	Computational Methodology	108
3.5.1	Governing Equations	108
3.5.2	Time Step Size	109
3.5.3	Convergence Criteria	109
3.6	CFD Procedure of the Savonius Hydrokinetic Turbine	110
3.6.1	Two-Dimensional Simulations of the SHKT Models	110
3.6.1.1	Design of 2D SHKT Models	110
3.6.1.2	Computational Methodology	112
3.6.1.3	Computational Domain	113
3.6.1.4	Mesh Configurations and Boundary Conditions	114
3.6.1.5	Mesh Independency Study	115
3.6.2	Three-Dimensional Simulations of the SHKT Models	116
3.6.2.1	Physical Design of 3D SHKT Models	117
3.6.2.2	Computational Turbulence Model	122
3.6.2.3	Computational Domain	123
3.6.2.4	Boundary and Initial Conditions	125
3.6.2.5	Mesh Configurations	128
3.7	CFD Approaches of the Darrieus Hydrokinetic Turbine	130
3.7.1	Designs of the 3D Delta-Shaped Blade DHKT Models	131
3.7.2	Computational Methodology	137
3.7.2.1	Turbulence Model	137
3.7.2.2	Computational Domain	138
3.7.2.3	Boundary and Initial Conditions	139
3.7.3	Mesh Configurations and Independency Study	141

3.8	CFD Method of the Combined SDHKT Models	146
3.8.1	Model Designs and Configurations of a Combined SDHKT	146
3.8.2	Computational Methodology	151
3.8.2.1	Turbulence Model	151
3.8.2.2	Computational Domain	152
3.8.2.3	Boundary and Initial Conditions	154
3.8.2.4	Meshing Configuration	156
4	RESULTS AND DISCUSSION	159
4.1	Predicted Simulation Results of SHKT Models	159
4.1.1	2D Simulations	159
4.1.1.1	Effects of the Central Shaft	160
4.1.1.2	Effects of a Guiding Wall	164
4.1.2	3D Simulations of the SHKT Models	168
4.1.2.1	Convergence of Simulation Results	168
4.1.2.2	Results of Case 1 and Validation with the Study of Saad et al., 2020	169
4.1.2.3	Effect of Stage Numbers	171
4.2	Simulation Results of DHKT Models	174
4.2.1	Validation of Simulation Results	174
4.2.2	Effect of the Model Designs	176
4.2.3	Effect of the Number of Blades	181
4.3	Result of the Combined SDHKT Models	184
4.3.1	On-Scale Model Validation	185
4.3.2	Comparison of Simulation Results	186
4.3.3	Effect of Attachment Angles (φ)	189
4.3.4	Effect of Ratios of Swept Area (R_{sa})	193
5	SUMMARY, CONCLUSION AND RECOMMENDATIONS FOR FUTURE RESEARCH	197
5.1	Summary and Conclusion	197
5.2	Recommendations for Future Research	201
	REFERENCES	203
	BIODATA OF STUDENT	217
	LIST OF PUBLICATIONS	218

LIST OF TABLES

Table	Page
2.1 Classification of hydro plants by capacity from previous works	11
2.2 Comparisons between horizontal and vertical axis turbines	16
2.3 Summary of the CFD turbulence models [41]	27
2.4 Domain sizes of 2D simulations used from previous studies	30
2.5 Domain sizes of 3D simulations used from published literature	30
2.6 Domain independency study from the study of Arab et al., 2017 [51]	31
2.7 Summary of previous works on the effect of aspect ratios	50
2.8 Summary of previous research works with various overlap ratios of Savonius rotors	55
2.9 Summary of published works on the effect of the number of stages	57
2.10 Summary of previous studies on the effect of pitch angles	81
2.11 Summary of published literature on the effect of attachment angles	94
3.1 Detail specifications of the designed 2D SHKT models	112
3.2 Design parameters for the SHKT model of case 1	118
3.3 Design parameters of the SHKT models of the case 2 – 4	121
3.4 The boundary and initial conditions for 3D simulations of combined SDHKT models	126
3.5 Comparison of solver setting between the case 1 and the study of Saad et al., 2020	127
3.6 Mesh details of case 1 compared with the study of Saad et al., 2020	129

3.7	Geometric features of DHKT models	135
3.8	The boundary and initial conditions for the 3D DHKT model simulations	140
3.9	The solver setting for simulations of SDHKT models	141
3.10	Setting of the mesh independency study	142
3.11	Main features of a combined SDHKT model	149
3.12	Design parameters for the models with various ratios of swept area	151
3.13	The boundary and initial conditions for 3D simulations of combined SDHKT models	155
3.14	The solver setting for simulations of SDHKT models	155
3.15	Mesh detail and configurations	158
4.1	Detail specifications of the verification DHKT models using NACA0012 profile	174
4.2	Details of combined Savonius-Darrieus turbines used for the validation in this study	187
5.1	Main features of an optimum combined SDHKT model	200

LIST OF FIGURES

Figure		Page
2.1	(a) World market energy consumption 1990–2035, (b) World net electricity generation by fuel from 2007 and project to 2035, (c) World renewable energy generation by source, and (d) hydrokinetic capacity forecast	10
2.2	Classification of hydro plants [11]	11
2.3	Components of small hydropower plant [12]	12
2.4	Classification of hydrokinetic turbines	13
2.5	Horizontal axis flow turbines adapted from Vermaak et al. [28]	13
2.6	Vertical axis flow turbines adapted from Vermaak et al. [28]	14
2.7	The power coefficient and operating range for different turbine types modified from F. Behrouzi et al [20]	17
2.8	Electrification system of hydro-turbine	18
2.9	Comparison between simulated results employed by several turbulence models and experimental results [36],[51]	28
2.10	2D Computational domain and boundary conditions from the study of Arab et al., 2017 [51]	32
2.11	(a) 2D-computational domain size, and (b) Computational domain analysis from the study of Liang et al., 2017 [43]	32
2.12	Computational domain sensitivity analysis [76]	33
2.13	Mesh independency studies [77],[51],[78]	36
2.14	Mesh independency results by using the refinement levels [42],[79]	37
2.15	Mesh sensitivity analysis by utilizing a variety of numbers of elements [43],[80]	37
2.16	Time dependency study [77],[78]	39
2.17	Schematic views of a conventional Savonius hydrokinetic turbine	40
2.18	Studies on some geometric parameters [17],[36],[82]	41
2.19	(a) The flow fields of two and three buckets Savonius turbines, (b)-(c) The performance curves of the rotors with two- and three-blades [38],[58]	42
2.20	Variation of mechanical power, power coefficient, and static torque coefficient with wind speed from different numbers of blades [83]	43
2.21	Experimental investigation on the effect of the shape of bucket profile from the studies of Talukdar et al., 2018 [58]	44

2.22	Vorticity contours for semi-circular and elliptical-bladed turbines from the studies of Talukdar et al., 2018 [58]	44
2.23	Velocity vector plots of semicircular and elliptical bucket shapes from the studies of Talukdar et al., 2018 [58]	45
2.24	(a), and (b) effect of the presence of end plates [17],[83], (c) c_p and c_t curves from the study of Saad et al., 2020 [36]	47
2.25	Helical Savonius designs with various shape designs and sizes of end plates from the study of Jeon et al., 2015 [82]	48
2.26	c_p and c_t curves from various shape designs and sizes of end plates from the study of Jeon et al., 2015 [82]	48
2.27	c_p and c_t curves from various end plate configurations from the study of Jeon et al., 2015 [82]	49
2.28	(a) Computational and (b) experimental results on the effect of aspect ratios [38],[83]	50
2.29	Experimental results using the modified Savonius rotors with different aspect ratios [84],[85]	51
2.30	Sketches of different overlap configurations	52
2.31	The maximum turbine performances occurred when β was equal to zero [36],[83]	53
2.32	The maximum turbine performances occurred when β was not equal to zero [84]	53
2.33	(a) – (c) Numerical results show the maximum turbine performance occurred when β was not equal to zero [38],[55],[88], and (d) contour of the pressure of a turbine with a shaft and spaces [38]	54
2.34	Multiple stage configurations of Savonius turbine design	56
2.35	Specific power and static torque coefficient of Savonius turbines with single and double stages [83]	58
2.36	Optimum number of stages [93]	59
2.37	(a) single- and three-stage configurations, (b) static torque coefficient (c_{ts}), (c) torque coefficient (c_t), and (d) power coefficient (c_p) from the study of Hayashi et al., 2005 [94]	60
2.38	Peak-to-peak values of dynamic torque coefficients at different tip speed ratios between a single-stage rotor and a three-stage rotor [94]	61
2.39	Dynamic torque coefficient versus bucket rotation angles between a single-stage rotor and a three-stage rotor at different tip speed ratios [94]	61
2.40	Turbine configurations with different and constant stage aspect ratios [95]	62

2.41	Variation of power, torque, and static torque coefficient for single, two, and three-stage rotors of (a) varied stage aspect ratio and (b) constant stage aspect ratio [95]	63
2.42	Influence of the Reynolds numbers and wind speed [36]	64
2.43	The Influence of the Reynolds number on the turbine performances [85],[86]	65
2.44	Influence of (a) the Reynolds number and (b) turbulence intensity of medium flow [17]	66
2.45	Historical line of Darrieus VAWT development [96]	67
2.46	Symmetric and non-symmetric airfoil shapes from the study of Mohamed, 2012 [45]	69
2.47	(a) $c_{p,max}$ of each 20 airfoil profile [132], and torque and power coefficient of (b) symmetric (NACA00XX), and (c) non-symmetric (NACA63xx) airfoils of NACA-series [45]	70
2.48	Variation of c_p with λ of (a) 2-blade, and (b) 3-blade Darrieus turbines with four different airfoil profiles [100]	71
2.49	(a) Variation of power coefficient with tip speed ratios of selected 25 airfoil profiles, (b) top five of power coefficient, and (c) shapes of LS(1)-0413 and NACA63-415 profiles [52]	72
2.50	Four different values of helical angles [103]	73
2.51	Variations of power coefficient with TSR from different helical angles [103]	73
2.52	Two and three-bladed H-rotor Darrieus turbines [45]	74
2.53	Computational results on the effect of the number of blades from the previous studies [22],[43],[100]	74
2.54	Power coefficient for different numbers of blades according to the optimum pitch angle [134]	75
2.55	Mid-span vorticity contours of (a) 2-bladed, and (b) 3-bladed NACA 0030 profiles [100]	76
2.56	Power coefficient versus tip speed ratio with various solidity values of (a) NACA0018, (b) NACA0015 [22], and (c) S-1046 airfoil profiles [45]	77
2.57	Instantaneous torque (c_m) and power (c_p) coefficients of S-1046 H-Darrieus turbines with four different solidities [45]	78
2.58	Experimental results on the effect of aspect ratios of Darrieus wind turbines [139],[104]	79
2.59	(a) Variation of pitch angle [61], and (b) Schematic of pitch angle [145],[146]	81

2.60	The optimum power coefficient occurred when pitch angles were negative (outward) [61],[103],[102],[142]	82
2.61	The optimum power coefficient occurred when pitch angle was zero ($\beta = 0$) [52],[144]	83
2.62	Power coefficient curves from different pitch angles [134]	85
2.63	Power coefficient of different wind turbines as functions of the tip speed ratio [21]	87
2.64	Turbine configurations of the combined systems; (a) nested coupling [21],[23],[148], and (b) stage coupling [39],[47],[87]	88
2.65	Effect of compact designs [150]	89
2.66	Turbine performances versus radius ratios from the studies of Saini and Saini, 2020 and Wakui et al., 2005 [46],[150]	91
2.67	Effects of radius ratios on the c_p and c_t of the Darrieus turbines from previous studies [42],[43],[154]	92
2.68	Numerical (a) [43], (b) [42], and (c) [46] and experimental (d) [149], and (e) [154] results of the effect of attachment angles on the combined turbine performances from previous studies	95
2.69	Three different operational configurations of hybrid turbines on the effect of the rotational velocities [48]	96
3.1	The flow of whole methodological approaches in these studies	100
3.2	(a) concept of turbine installation in an irrigation canal, and typical configuration of an irrigation canal with (b) straight wall and (c) inclined wall	101
3.3	Flow chart of turbine design and optimization processes	102
3.4	CFD simulation processes	104
3.5	A flow diagram of the optimization processes in the current studies	105
3.6	A flow chart of the simulation procedure of SHKT models	106
3.7	DHKT investigation processes	107
3.8	Investigation procedure of the combined SDHKT models	107
3.9	(a) a schematic diagram of a 2D SHKT model, and (b) Designs of a central shaft	111
3.10	The design of a guiding wall configuration	111
3.11	A symmetric computational domain for 2D simulations	114
3.12	The 2D mesh configuration	115

3.13	Mesh independency study of a 2D SHKT model	116
3.14	A 3D conventional SHKT model	117
3.15	Isometric, front, and top views of the SHKT model of case 1	118
3.16	Schematic configurations of the SHKT model of the case 2 - 4	120
3.17	The SHKT models of the case 1 – 4	121
3.18	Comparison between the predicted power coefficient versus tip speed ratio and experimental results of (a) Damak et al. 2018 [157] and experimental and numerical results of (b) Kothe et al. 2020 [37]	122
3.19	Schematic representation of SHKT computational domain	124
3.20	Mesh configuration of a 3D SHKT model using in case 1	129
3.21	Mesh configuration of 3D SHKT models used in the case 2-4	130
3.22	(a) NACA0012 profile, (b) Normal and bent NACA0012 profiles, (c) In-plane, and (d) Twisted delta-shaped blade designs	132
3.23	Schematic sketches of the isometric, front, and top views of the 2-blade (a) In-plane, and (b) Twisted delta-shaped blade with reduced cross-sectional areas models	133
3.24	The 2-blade DHKT model designs (MD1-MD4)	134
3.25	Designs of 3-blade DHKT models	134
3.26	Optimization procedures of the Darrieus hydrokinetic turbine	136
3.27	Different views and geometrical parameters of (a) a rectangular computational domain, (b) a top view of a domain, (c) an isometric view of the rotating zone, and (d) a top view of the rotating zone	139
3.28	(a) Variation of instantaneous c_p with flow time of several mesh configurations (M1-M7), and (b) The average c_p of the 8th revolution of each mesh configuration and its percentage differences (δ)	144
3.29	Mesh configuration, M4 (MI = 30 mm., MO = 40 mm.) in simulations of the DHKT models	145
3.30	Design of the combined SDHKT model	147
3.31	Combined SDHKT models with several attachment angles (φ)	148
3.32	Combined SDHKT models with three different values of R_{sa}	150
3.33	Different views and geometric parameters of (a) the computational domain, (b) the top view, (c) the isometric view of the rotating zone, and (d) the top view of the rotating zone	153
3.34	Mesh configuration, M4 (MI = 30 mm., MO = 40 mm.) using in the simulations of SDHKT models	157

4.1	Comparison of the results from this study with published literature	160
4.2	Instantaneous torque with different shaft designs and upstream velocities U_∞	161
4.3	Variation of $P_{m,max}$ (watts) as a function of U_∞ for all cases	162
4.4	Velocity and pressure contours across 2D SHKT models	163
4.5	The variation of c_p and instantaneous torque with θ , and the velocity contours of a full-shaft SHKT model with and without a guiding wall at fixed $U_\infty = 0.5$ m/s	165
4.6	Pressure contours of a full-shaft SHKT model in cases of (a) with and (b) without a guiding wall at $\theta = 135^\circ$, 180° and 225° for fixed $U_\infty = 0.5$ m/s	167
4.7	Instantaneous power coefficient (c_p) and cycle-averaged power coefficient ($c_{p,avg}$) of each revolution from the simulation of a model in Case 4	169
4.8	Power and torque coefficient of a conventional SHKT model verified with the CFD results of Saad et al., 2020	170
4.9	Contour of pressure distribution along with streamlines of the case 1 from this study and the study of Saad et al., 2020	171
4.10	Power and torque coefficient of SHKT models of the case 1 – 4	172
4.11	c_p versus θ of the case 2 – 4 when $\omega = 30$ rad/s and $\lambda = 0.8$	173
4.12	Validation of the current CFD work with experimental data of Doan et al., 2020 [124]	175
4.13	Comparison of simulation results	176
4.14	Variation of the average c_p and c_t versus λ of (a) <i>MD1</i> and <i>MD2</i> , and (b) <i>MD3</i> and <i>MD4</i> designs	177
4.15	Isometric views of velocity streamlines inside the inner domain of the model designs, <i>MD1</i> - <i>MD4</i> when $\theta = 360^\circ$, $\omega_b = 10$ rad/s, and $U_\infty = 1.5$ m/s	178
4.16	Sectional total pressure contours and velocity vector around the blades of the DHKT model, <i>MD1</i> – <i>MD4</i> at $\theta = 0^\circ$, and 120° , $\omega_b = 10$ rad/s and $U_\infty = 1.5$ m/s	180
4.17	Variation of the average (a) c_p and (b) c_t with λ of 2-blade (<i>MD3</i> , <i>MD4</i>), and 3-blade (<i>MD7</i> , <i>MD8</i>) models	181
4.18	Isometric views inside the inner domain surrounded by velocity streamlines of <i>MD3</i> , <i>MD4</i> , <i>MD7</i> , and <i>MD8</i> at the maximum operational condition	183
4.19	Sectional velocity and total pressure contours of <i>MD3</i> , <i>MD4</i> , <i>MD7</i> , and <i>MD8</i> at the maximum operational condition	184

4.20	CFD validation of (a) the CAD model in this study and (b) the model from the experiment of Sahim et al., 2013 [154]	185
4.21	Validation of the current CFD work with experimental data of Sahim et al., 2013 [154]	186
4.22	Comparison of the current CFD results and previous published experimental and numerical results	188
4.23	Variation of the average (a) c_p and (b) c_t with tip speed ratios for various attachment angles (φ)	190
4.24	Average c_p and c_t with tip speed ratios for each attachment angle (φ)	191
4.25	Maximum average power coefficient of SDHKT combined with MD3 and MD4 at different attachment angle	192
4.26	Variation of average c_p and c_t with tip speed ratios at different R_{sa}	194
4.27	Velocity streamlines and vectors of a combined SDHKT model (R_{sa3})	195
4.28	Velocity contours, streamlines, and vectors around the combined SDHKT and the MD3 of DHKT models compared with results of Saini & Saini, 2020 [153]	196
5.1	Characteristics of the proposed combined Savonius-Darrieus hydrokinetic turbine	201

LIST OF ABBREVIATIONS

ABS	Acrylonitrile Butadiene Styrene
SHKT	Savonius hydrokinetic turbine
DHKT	Darrieus hydrokinetic turbine
SDHKT	Savonius- Darrieus hydrokinetic turbine
2D	Two-dimensional
3D	Three-dimensional
URANS	The time-dependent Unsteady Reynolds-Averaged Navier Stokes
SIMPLE	Semi-Implicit Method for Pressure Linked Equation
A_s	Swept area ; [m^2]
A_{sS}	Swept area of a SHKT model ; [m^2]
A_{sD}	Swept area of a DHKT model ; [m^2]
A	Projected, sectional area of rotor ; [m^2]
c	Airfoil chord length ; [m]
C_p	Power coefficient ; [-]
$C_{p,avg}$	Cycle-averaged power coefficient ; [-]
$C_{p,max}$	Maximum power coefficient ; [-]
C_t, C_m	Torque coefficient ; [-]
$C_{t,avg}$	Cycle-averaged torque coefficient ; [-]
$C_{t,max}$	Maximum torque coefficient ; [-]
t	Thickness ; [m]
r	Rear distance ; [m]
R	Radius of a turbine ; [m]
R_r	Radius of a rotor ; [m]
R_b	Radius of a bucket ; [m]
R_s	Radius of a SHKT model ; [m]
R_D	Radius of a DHKT model ; [m]
R_{sa}	Ratio of swept areas ; [-]
D	Diameter of turbine ; [m]
D_r	Diameter of rotor ($2R_r$) ; [m]

D_b	Diameter of a bucket ; [m]
D_S	Diameter of a SHKT model; [m]
D_D	Diameter of a DHKT model; [m]
D_C	Diameter of a combined SDHKT model; [m]
D_o	Diameter of endplates ; [m]
D_s	Diameter of a shaft ; [m]
F	Frontal distance ; [m]
H_r	Height of rotor ; [m]
H_S	Height of a SHKT model ; [m]
H_D	Height of a DHKT model ; [m]
H_C	Height of a SDHKT model ; [m]
n	Number of blade ; [-]
S	Tunnel sectional area ; [m ²]
L	Length ; [m]
L_G	Length of a guiding wall ; [m]
S_G	Location of a guiding wall ; [m]
W	Width ; [m]
Re	Reynolds number ; [-]
ρ	Density of fluid ; [kg/m ³]
μ	Molecular viscosity of fluid ; [Pa s]
μ_t	Turbulent viscosity ; [Pa s]
k	Turbulent kinetic energy ; [m ² /s ²]
ε	Turbulent dissipation rate [m ² /s ³]
ω	Specific dissipation rate ; [s ⁻¹]
ω_o	Rotation speed, angular velocity ; [rad/s]
U_∞	Upstream velocity of fluid ; [m/s]
λ	Tip speed ratio ; [-]
σ	Solidity ; [-]
s/D	Gap ratio (GP) ; [-]
AR	Aspect Ratio ; [-]
β	Overlap ratio ; [-]

e	Overlap distance ; [m]
ψ	Twist angle ($^{\circ}$)
ϕ	Turbines diameter ; (m)
θ	Azimuthal angle ; ($^{\circ}$)
φ	Attachment angles ; ($^{\circ}$)
α	Angle of attack ; [$^{\circ}$]
δ, G	Pitch angle ; [$^{\circ}$]
τ	Torque ; [$N\cdot m$]
τ_{ins}	Instantaneous torque ; [$N\cdot m$]
P_{Theory}	Theoretical hydrokinetic power ; [W]
Δt	Time step ; [s]

CHAPTER 1

INTRODUCTION

1.1 Research Background

For more than half a century, massive industries and city civilization have been rapidly expanding however, agriculture providing food and nutrients still has been important for worldwide populations. Both industrial and agricultural activities require energy especially electricity for their productivity. It was reported that the demand for fossil fuel usage has been progressively increasing corresponding with the significant increase in electricity demand, or amount of energy consumption [1],[2]. Therefore, greenhouse gas emissions mainly caused by fossil fuel combustion have also increased. Developing and developed countries are concerned about global warming situations caused by greenhouse gases (GHGs), especially carbon dioxide (CO₂) and Methane (CH₄). Most countries have been endeavoring to reduce GHG emissions by avoiding the usage of fossil fuel combustion. Renewable energy used instead of fossil fuel is a promising solution to solve global warming issues. There are many kinds of renewable energy sources. However, hydropower is more popular than other resources [2],[3], especially in countries with ample irrigation networks. Global energy generation from hydropower has rapidly increased yearly [2]. Hydropower generation is more sustainable and reliable [4] because it has competitive costs, accepted efficiency [5], and minimal environmental impacts [6],[7]. Besides, water levels and flow rate to use in hydropower generation systems are controllable [8], predictable, and measurable [9],[10].

Most wetland rice production schemes in Malaysia and Thailand have good networks of irrigation and drainage canals that have been built for the benefit of the farmers. Those canals which have ample volume of running water all year round could be harnessed for hydropower. Moreover, both countries have plenty of massive hydro resources and a high average annual rate of rainfall in highland areas. As a consequence, hydropower has much more potential for harnessing hydro energy to convert into electricity rather than other types of renewable resources.

Based on the generated capacity hydropower plants are classified as pico, micro, mini, small, or large plants [11],[12]. The potential energy from the different water elevations can be converted to electricity by using a mechanical device called a turbine. Generally, electricity generation from a dam which is classified as a large plant (> 10 MW [13]) can generate massive quantities of electricity. However, a huge dam site requires enormous construction which not only disturbs the natural environment, but also incurs a lot of construction costs, materials, and equipment [14]. A dam construction requires a small forebay, penstock, powerhouse, and turbine [12],[14]. Those requirements are not suitable for areas that have only a small shallow river or a human-made irrigation canal with low flow rates and with or without small elevations. Consequently, hydrokinetic technologies known as hydrokinetic turbines which are a promising solution in the current century can be an alternative solution to replace fossil fuel usage and dam construction thanks to its ability to operate under low head, low flow rate, and only require a small hydro resource area. It also has a competitive cost, accepted efficiency, and minimal environmental impacts.

Each type of turbine including vertical, or horizontal axis hydrokinetic turbines has a specific operational range as well as pros and cons. Vertical axis turbines have been widely utilized in many applications due to many advantages for example the ability to operate under a variety of flow directions, simple fabrication and installation, low generator cost, manufacturing cost, and maintenance cost, and proper for shallow rivers or channels [9]. A vertical axis turbine is a mechanical device used to generate electricity from running water in rivers flowing at between 0.5 and 2.5 m/s [15]. In practice, the Savonius hydrokinetic turbine (SHKT) and Darrieus hydrokinetic turbine (DHKT) categorized as vertical axis turbines have also been used broadly to generate electricity as a means of a renewable source over using fossil fuel.

Principally, the performances of these SHKT and DHKT depend on geometric parameters or turbine shapes, operational conditions, and fluid flow conditions [16]–[18] whether laminar or turbulence flows. Moreover, each turbine type has its specific range of operation. The Savonius turbines operate efficiently at a low tip speed ratio (λ) range with optimum efficiencies between 0.55 to 1.2 [19]. On the other hand, Darrieus turbines perform well during a high λ range ($\lambda > 1.2$) of operation [20]–[22]. In general, self-starting capability and efficiency are the main concern. A conventional Savonius turbine is good at starting ability at lower flow speeds [8],[19]. However, it suffers from low performance at a high rate of flow or high tip speed ratio [8],[19]. On the other hand, a conventional Darrieus turbine provides excellent dynamic efficiency at higher operational ranges but suffers from the problems of self-starting [19],[23]. To overcome the aforementioned problems and complement the advantages of each other, a combination of the Savonius and Darrieus

hydrokinetic turbines known as a combined Savonius-Darrieus hydrokinetic turbine (SDHKT) has been proposed in this research study.

1.2 Problem Statement

Hydropower, one of several renewable resources is a promising solution to solve the global warming issue and generate clean electricity. Hydrokinetic turbines that can capture and transform hydropower into mechanical power are suited for remote rural areas that have only running water and have not enough area to construct huge infrastructures for a dam. Vertical axis turbines including Savonius and Darrieus turbines suited for shallow rivers or drainage channels have a specific range of operation. The performances of these turbines depend on the turbine's geometric parameters, operational conditions, and flow conditions. In general, self-starting capability and efficiency are the main concerns. A conventional Savonius turbine has good starting ability at lower flow speeds. However, it suffers from low performance. On the other hand, a conventional Darrieus turbine provides excellent dynamic efficiency at higher operational ranges but suffers from the problems of self-starting. As a consequence, the combination of Savonius and Darrieus turbines known as a combined Savonius-Darrieus hydrokinetic turbine (SDHKT) is proposed aimed to overcome the aforementioned problems and complement the advantages of each other.

Optimized designs of the Savonius, Darrieus, and combined Savonius-Darrieus hydrokinetic turbines were investigated computationally. For the Savonius hydrokinetic turbine, several parameters have been investigated so far. However, it is not clear about the information of a turbine central shaft and a guiding wall. Therefore,

the effect of a turbine central shaft and the presence of a guiding wall were considered. For the Darrieus hydrokinetic turbine, many geometric parameters have been studied. However, there is a lack of study on the delta-shaped blade design. Therefore, the effect of delta-shaped-blade designs and their blade numbers are considered in this study. The in-plane and twisted-bladed turbines with reduced and constant cross-sectional areas of the blades are investigated. Besides, the number of blades e.g. two and three blades are considered. Finally, the optimum designs of the Savonius and Darrieus hydrokinetic turbines are combined. The combined turbines with different configurations of attachment angles and ratios of swept areas are extendedly considered by using CFD simulations.

1.3 Research Aims and Objectives

This study aims to evaluate through computational fluid dynamics (CFD) the model designs and configurations of the combined Savonius-Darrieus hydrokinetic turbine (SDHKT).

The specific objectives of this research are:

1. To study the principal design parameters of the Savonius hydrokinetic turbine by using CFD simulation.
2. To study the principal design parameters of Darrieus hydrokinetic turbine using three-dimensional CFD simulations.
3. To synthesize the advantages of Savonius and Darrieus turbines into a combined design and optimized for application in irrigation canals by employing three-dimensional CFD simulations.

1.4 Thesis Scope and Limitation

This research has been conducted with three main parts which were involved with the investigation of the optimum designs of Savonius, Darrieus, and combined Savonius-Darrieus hydrokinetic turbines by using the method of computational fluid dynamics.

1. Determine the effect of three different designs of a central shaft: without shaft, with a shaft and space between a shaft and buckets, and with full shaft, and evaluate with and without the presence of a guiding wall of the Savonius hydrokinetic turbine (SHKT) models by using two-dimensional (2D) CFD simulations.
2. Investigate the effect of stage numbers including single stage, 2 stages with different and constant stage aspect ratios of the 3D Savonius hydrokinetic turbine models by using three-dimensional CFD simulations. Validation tests on the developed computational approach against the published work of Saad et al., 2020 will be carried out.
3. Evaluation of new designs of the delta-shaped blade Darrieus hydrokinetic turbine. Four different designs of turbine blades which are the blades that are in-plane and twisted with reduced and constant delta-shaped blades. Besides, the effect of the number of blades is also determined. Validation tests on this developed computational method against the published experimental work of Doan et al., 2020 will be conducted.

4. Investigate and optimize the combined configurations indicated by the five different values of the attachment angles, e.g. 0° , 20° , 30° , 90° , 120° , and three different ratios of swept areas, for example, 0.4, 0.3, 0.2 of the combined Savonius-Darrieus hydrokinetic turbine (SDHKT) models by employing 3D CFD simulations. Validation tests on the developed computational approach against the laboratory work of Sahim et al., 2013 will also be carried out.

The purpose of this study was to evaluate the individual model designs of the Savonius and Darrieus hydrokinetic turbines, and the model configurations of the combined Savonius-Darrieus hydrokinetic turbines by using CFD simulations. To evaluate several design parameters of those turbines, many simulations are employed rather than investigations by experiments in order to reduce fabrication costs. Moreover, to reduce the computational cost, domain, and turbulence model independency studies were imposed based on the published literature. The upstream water velocity used in this study was assumed to be in the range of 0.5-2.5 m/s which was mentioned by A. Janon, 2020 [15].

1.5 Thesis Overview

This thesis is divided into 5 chapters. The opening chapter introduces the background and motivation of the study, the problem statement, the research objectives as well as the scope of the research work. In Chapter 2, fundamental knowledge of hydrokinetic turbines, literature reviews of the Savonius, Darrieus, and combined Savonius-Darrieus turbines as well as the principle of computational fluid dynamics (CFD) are introduced. Chapter 3 presents the methodology of whole processes including turbine design and optimization processes, and CAD designs. CFD approaches including

computational domain, time step size, turbulence model, mesh configurations, boundary, and initial conditions used in the simulations of the Savonius, Darrieus, and combined Savonius-Darrieus hydrokinetic turbines were also explained systematically in this chapter. Chapter 4 presents the CFD results from the investigation of the Savonius, Darrieus, and combined Savonius-Darrieus hydrokinetic turbines. The results are also discussed throughout this chapter. Chapter 5 concludes the overall results of the study. The recommendations for the possible future works are also stated.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

Currently, the emission of greenhouse gases has progressively increased due to the significant increase in electricity needs and energy consumption [1] as illustrated in Figure 2.1(a) [2]. Global electricity generation generated mainly from fossil fuels including natural gas, and coal has continuously increased from past to future forecasts as depicted in Figure 2.1(b) [2]. To reduce amounts of fossil fuel usage, renewable energy is a promising solution to solve global warming issues to help reduce dependency on fossil fuels. There are many kinds of renewable energy sources including hydroelectricity, wind, geothermal, and others. Hydropower however is more popular than others [1],[2] as seen in Figure 2.1(c) [2] thanks to the many advantages, especially in countries where there are ample irrigation networks.

Hydropower generation is reliable because it has competitive costs, accepted efficiency [2],[3], and minimal environmental impacts [6]. Besides, water levels and flow rate to use in hydropower generation systems are controllable [8], predictable, and measurable [6],[7]. Among those sources, global energy generation from hydropower has rapidly increased yearly as depicted in Figure 2.1(d) [2] due to many advantages including competitive cost, reliability, sustainability, efficiency, and environmental impacts [3]. It was accepted that hydropower is controllable, predictable, measurable, reliable, and less cost per unit.

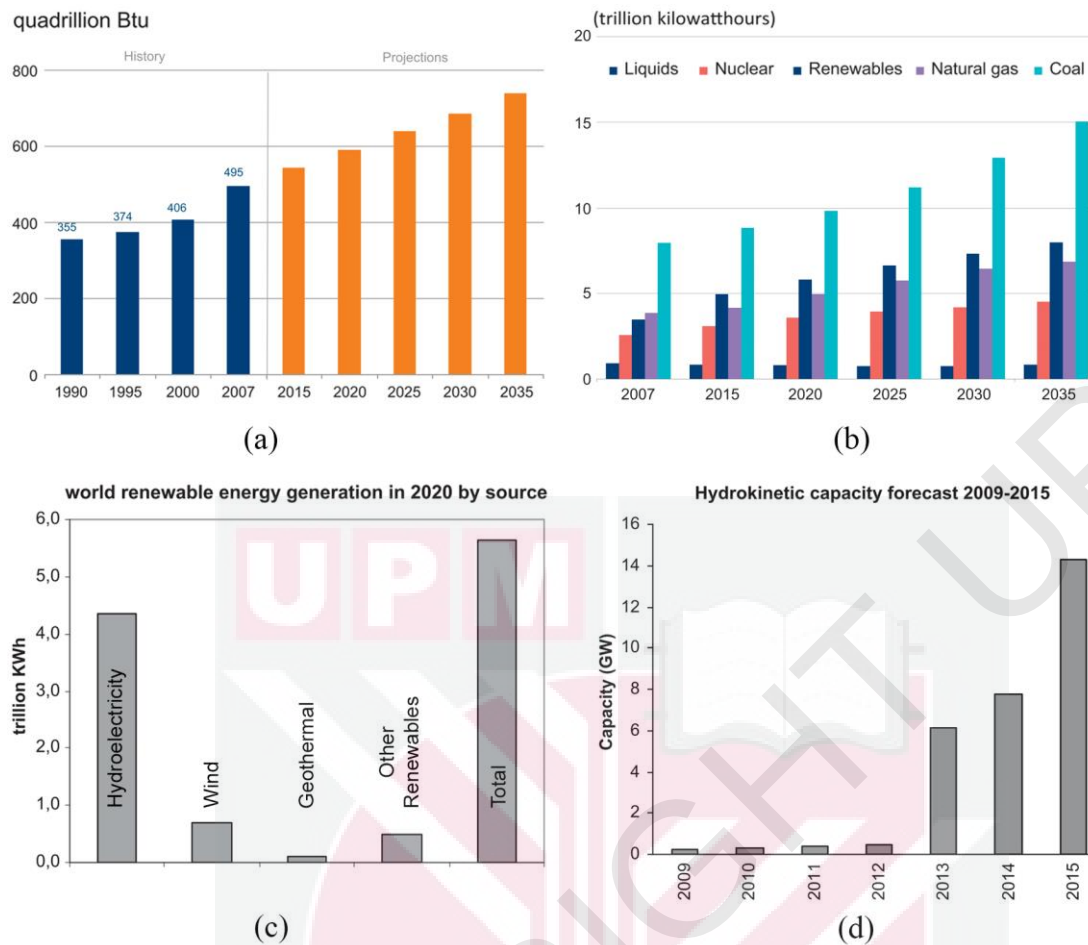


Figure 2.1 : (a) World market energy consumption 1990–2035, (b) World net electricity generation by fuel from 2007 and project to 2035, (c) World renewable energy generation by source, and (d) hydrokinetic capacity forecast

Based on the generated capacity as defined in Table 2.1 and Figure 2.2 [11], hydropower plants are categorized into pico, micro, mini, small, and large plants [11],[12]. The potential energy from the different water elevations can be converted to electricity by using a mechanical device known as a turbine. Generally, electricity generation from a dam which is classified as a large plant (> 10 MW [13]) can generate massive quantities of electricity. Nevertheless, a huge dam site requires enormous construction which either disturbs the natural environment or incurs a lot of construction costs, materials, and equipment [14].

A dam construction requires a small forebay, penstock, powerhouse, and turbine [12],[14] as shown in Figure 2.3 [12]. Those mentioned requirements are not appropriate for the areas that have only a small shallow river or a human-made irrigation canal with low flow rates and with or without small elevations.

Table 2.1 : Classification of hydro plants by capacity from previous works

Categories	Quantity of electricity production
Pico hydro plant	< 5 kW [13],[24]–[26] < 10 kW [6]
Micro hydro plant	< 20 kW [27] > 5 kW but < 100 kW > 6 kW but < 100 kW [13] ≤ 100 kW [28] 10 kW - 200 kW < 500 kW [6]
Mini hydro plant	> 500 kW but < 2 MW [6] 100 – 1,000 kW [28] < 1 MW [13]
Small hydro plant	< 10 MW 100 kW - 10 MW 2.5 MW - 25 MW [6] Up to 25 MW [13]
Large hydro plant	> 100 MW [13]

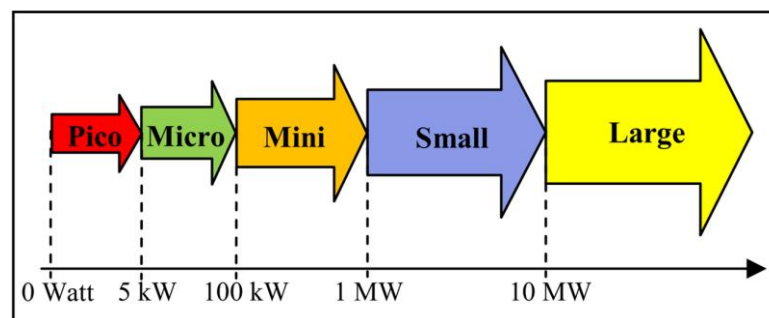


Figure 2.2 : Classification of hydro plants [11]

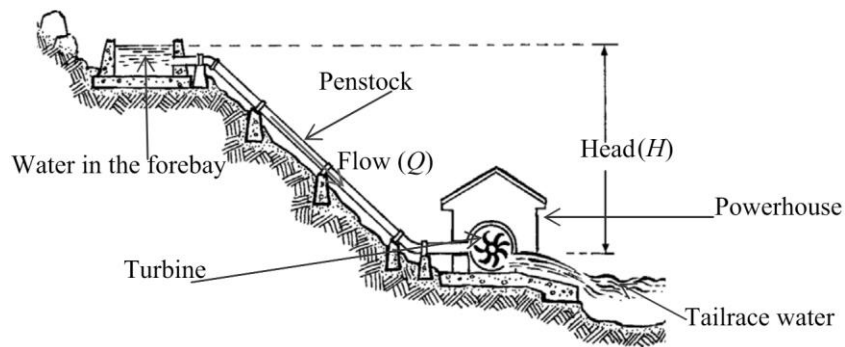


Figure 2.3 : Components of small hydropower plant [12]

Therefore, hydrokinetic technologies known as the hydrokinetic turbines which are promising solutions nowadays can be alternative solutions replacing fossil fuel usage and dam construction thanks to their ability to operate under low head, low flow rate, and only require a small hydro resource area. It also has a competitive cost, accepted efficiency, and minimal environmental impacts.

2.2 Classification of Hydrokinetic Turbines

In general, hydrokinetic devices utilized to generate electricity are called hydrokinetic turbines. Based on the axis alignment of a turbine rotor, hydrokinetic turbines can be classified into horizontal and vertical axis turbines [22],[28]. As depicted in Figure 2.4, in a group of horizontal axis turbines including axial flow, incline axis, and cross-flow turbines, a turbine axis is principally parallel to the water surface [9],[14]. On the other hand, the rotational axis of vertical axis turbines is vertical and orthogonal to the water surface and flow direction [9],[14]. The horizontal axis turbines illustrated in Figure 2.5 [28] are categorized into an inclined axis turbine and an axial flow turbine when its axis is parallel to the fluid flow direction [28], and a cross-flow turbine or an in-plane axis turbine if and only if the rotor axis is perpendicular to the flow direction but parallel to the water surface [14],[28].

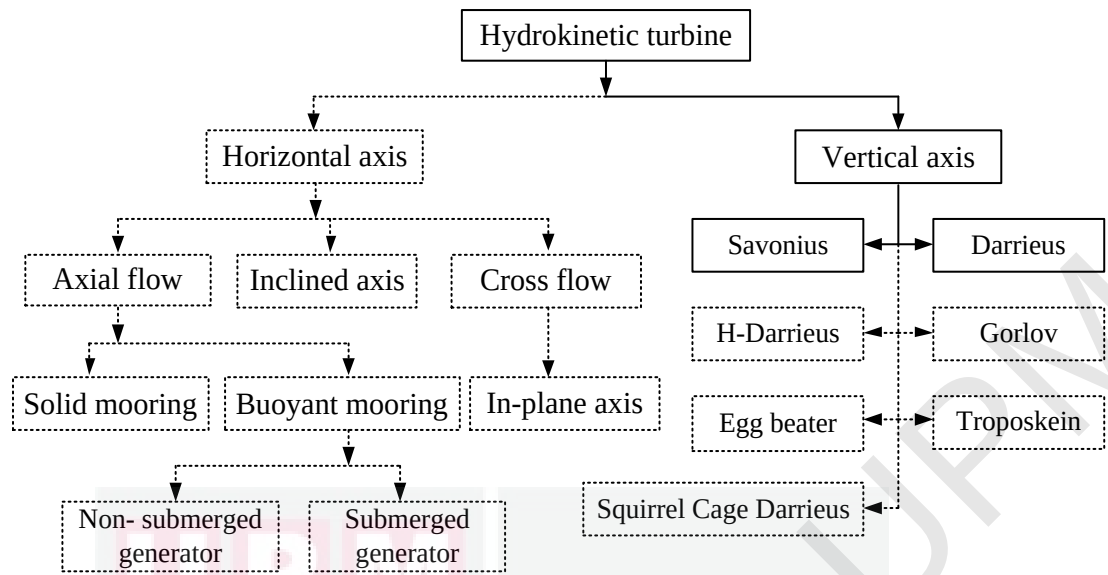


Figure 2.4 : Classification of hydrokinetic turbines

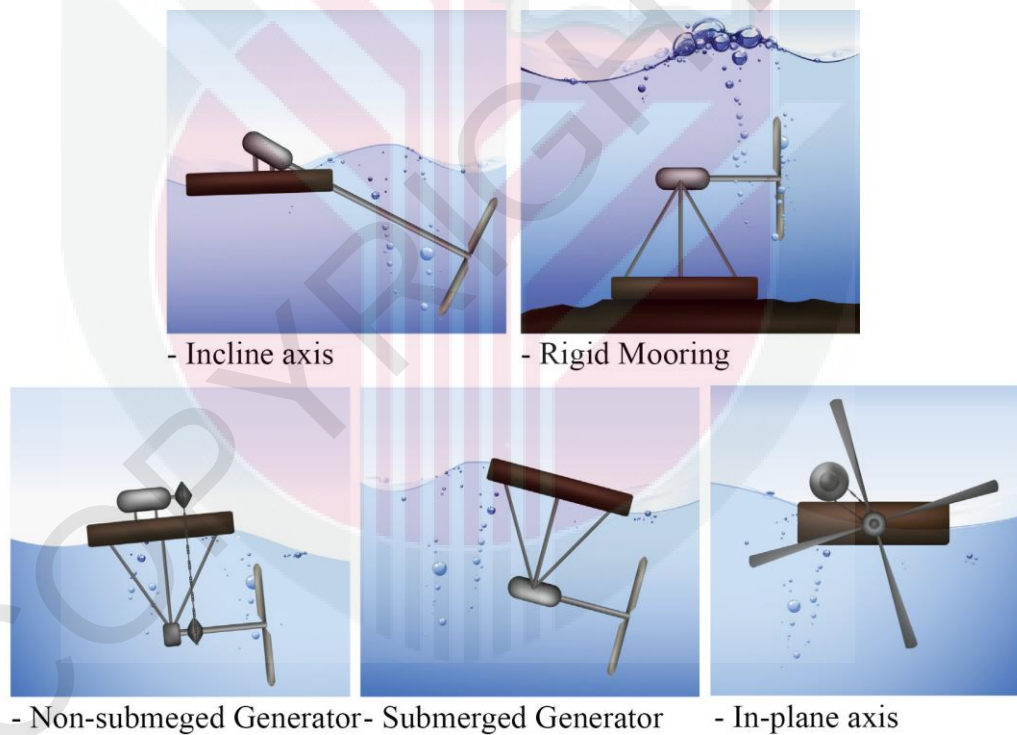


Figure 2.5 : Horizontal axis flow turbines adapted from Vermaak et al. [28]

There are several types of vertical axis turbines, for example, the Savonius, Darrieus, and Gorlov turbines which are named after their inventor. Figure 2.6 [28] presents some examples of the Savonius, Darrieus, and Gorlov turbines. Furthermore, vertical axis turbines can be distinguished according to fluid force characteristics i.e., lift-based turbines including all Darrieus rotor designs, and drag-based turbines which are all Savonius rotor designs.

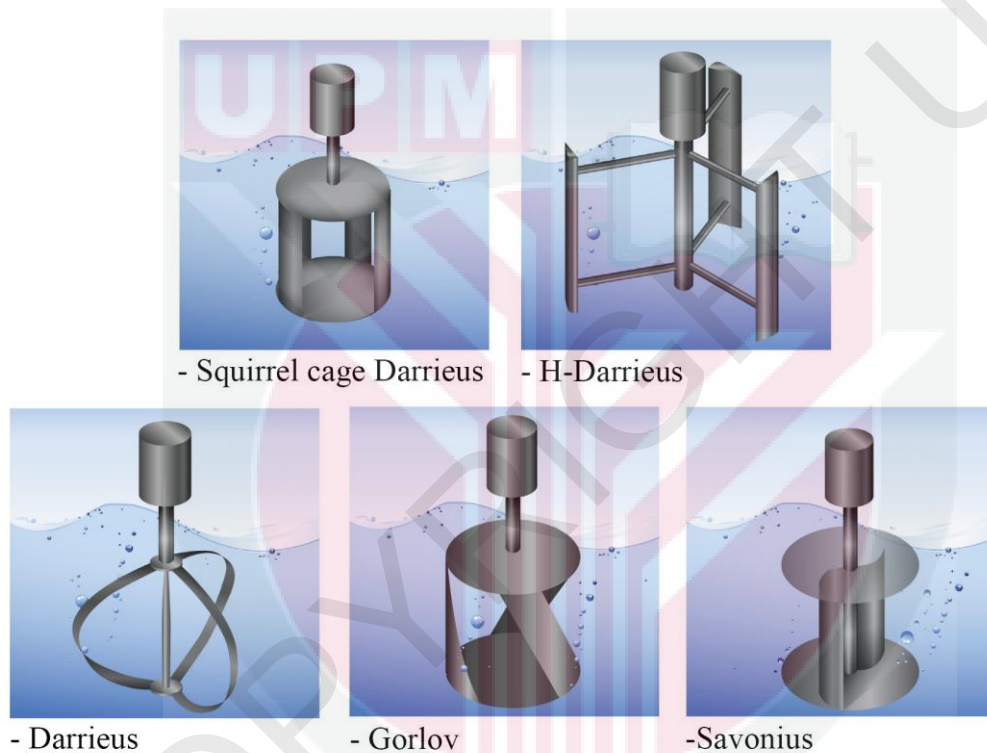


Figure 2.6 : Vertical axis flow turbines adapted from Vermaak et al. [28]

2.3 Turbine Selections

The selection of the most optimum turbine in any applicable hydro-plant site is followed crudely by the information on site and flow characteristics [14], head and flow available [6], and velocity of water current [20]. However, the available head is not so important for small hydrokinetic plant which depends upon only flow velocity of medium fluids.

As mentioned before, based on its rotor axis alignment, there are two main types of turbines including the horizontal and vertical axis turbines. To get the optimized turbine, a comparison of the technical specifications between both these turbine types including advantages and disadvantages presented in Table 2.2 is necessary. It can be noticed clearly that, horizontal axis turbines are better in self-starting capability and have higher efficiency compared to vertical ones, but operations of those turbines are limited and clogged by direction of flow and debris. Even though the efficiency of vertical axis turbines is lower than the horizontal ones, the vertical axis turbines are suitable for low flow and shallow rivers or channels. Besides, the operation of these turbines is available in any flow direction. It has been recommended by many researchers that vertical axis turbines are more suitable to utilize in irrigation systems, compared with the other types due to the pros such as shallow canals, flow directions, and costs [3],[9],[14].

Table 2.2 : Comparisons between horizontal and vertical axis turbines

Horizontal axis turbines	
Advantages	Disadvantages
- Self-starting capability [14] - Higher efficiency [14]	-The operation is limited by the direction of flow [14] - High generator cost, manufacturing cost, maintenance cost [14] - Improper for small, narrow rivers - Clogged with debris easily
Vertical axis turbines	
Advantages	Disadvantages
- The operation occurs in any variety of flow directions [14] - Low generator cost, manufacturing cost, maintenance cost [14] - Proper for shallow rivers or channels [9],[14] - Deflect incoming debris	- Low exciting torque at the beginning of working [14],[28] - Starting mechanisms are required [14],[28] - Lower efficiency [14],[28]

To do a turbine selection, efficiency, and self-starting capability of the turbines are the main concerns. The performance of turbines is represented by power coefficient (c_p), and torque coefficient (c_t) which depend mainly upon the operational characteristics, and operational range [16]–[18] represented usually by a tip speed ratio (λ) which is a ratio of upstream water flow velocity (U_∞) and tip speed of a turbine blade (v). As depicted in Figure 2.7 [20], the power coefficient and operating range for a sampling of horizontal and vertical axis turbines are plotted by using data from previous studies. It can be seen that each type of turbine has its own specific range of operation. The Savonius turbines operate efficiently at a low range of tip speed ratio (λ) with

optimum efficiencies between 0.55 to 1.2 [19]. On the other hand, the Darrieus turbines operate well during a high λ range ($\lambda > 1.2$) [20]–[22] which is wider than the Savonius turbines. Apart from a vertical axis turbine, it notes that the horizontal turbine types such as high-speed propeller, American multi-blade, and Dutch multiblade turbine show higher power performance than the vertical ones. The maximum power coefficient (c_p) of the Savonius and Darrieus turbines are around 0.2 and 0.4, respectively. However, this performance still depends on a turbine scale. The power coefficient of a small-scale river turbine is reduced to around 0.10–0.25 due to its own losses [2]. There is a fact that, turbine power performance of all types is limited by the Betz limit which is a theoretical upper limit for power generation of isolated turbines without ducting. It was derived that a maximum power coefficient is around 16/27 or 59.3% [2],[20],[28].

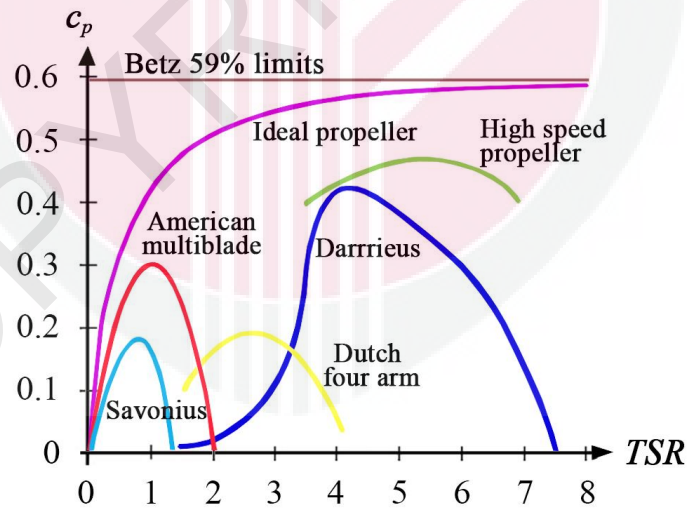


Figure 2.7 : The power coefficient and operating range for different turbine types modified from F. Behrouzi et al [20]

2.4 Principal of Hydrokinetic Energy

2.4.1 Turbine Electrification

Electricity generation from a small hydropower plant such as typical irrigation canals or rivers consists of 3 main components which are intake, turbine, and power generation as shown in Figure 2.8. The processes start from water flowing through a turbine called a hydrokinetic turbine where kinetic energy from running water is transmitted to mechanical power to rotate a turbine. The turbine rotor and shaft connected directly to the rotor are rotated corresponding to the entrance velocity of water and turn a generator. Finally, electricity is transmitted to households by transmission lines.

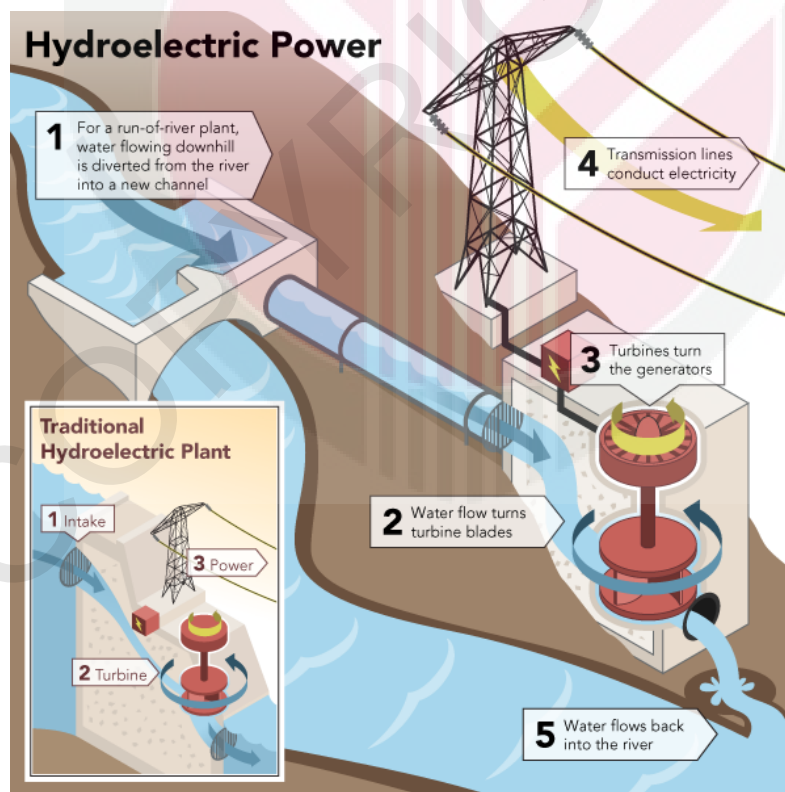


Figure 2.8 : Electrification system of hydro-turbine

Theoretically, the rate of kinetic energy change of a water stream with velocity U_∞ flowing through such an area A with mass flow rate ρAU_∞ can be represented in terms of the hydrokinetic power (P_{Hydro} ; watts) as written in an equation (Eq. 2.1). P_{Hydro} is converted by a turbine rotor into the mechanical power (P_m) or shaft power (P_s) derived in an equation (Eq. 2.2) which is a product of torque at a turbine shaft and a rotational speed of a turbine rotor. In general, hydrokinetic power cannot convert 100% into mechanical power. The available energy that a turbine can extract depends on turbine efficiency or turbine performance. Besides, the maximum power extracted by any wind or hydrokinetic turbines limited by the Betz limit is 59.3% of the power available by flowing wind or water.

$$P_{Hydro} = \frac{1}{2} \rho U_\infty^3 A = \rho U_\infty^3 H_s R_s \quad (\text{Eq. 2.1})$$

$$P_m = \tau \omega \quad (\text{Eq. 2.2})$$

2.4.2 Coefficient of Performances

Theoretically, the turbine performance can be identified by two dimensionless parameters which are the power coefficient (c_p) and torque coefficient (c_t). The c_p is defined as a ratio of the actual mechanical power (P_m) generated by a turbine rotor to the available hydropower (P_{Hydro}) as expressed in an equation (Eq. 2.3). The c_t expressed as an equation (Eq. 2.4) represented the net dynamic torque produced from all blades of a turbine rotor at a particular rotor angle. Besides, the relationship between c_p and c_t is derived in an equation (Eq. 2.5) which λ is called a tip speed ratio (TSR). As stated in an equation (Eq. 2.5), a tip speed ratio (λ) or TSR defined as a

ratio of the tangential velocity (v), or tip speed of a turbine blade to the upcoming water velocity (U_∞) is a dimensionless parameter used to distinguish the performance of the turbine. Mainly, the c_p and c_t of general hydrokinetic turbines rely on TSR. In fact, for constant upstream flow, lower TSR means that a turbine rotor rotates considerably slowly and allows medium fluids to flow through the open spaces between the turbine blades with a few extracted powers. On the other hand, when a turbine rotor rotates too fast, the rotating blades behave as a solid obstruction blocking the flow and accordingly decreasing the extracted power.

$$c_p = \frac{P_m}{P_{Hydro}} = \frac{\tau\omega}{0.5\rho U_\infty^3 H_D D_D} \quad (\text{Eq. 2.3})$$

$$c_t = \frac{4\tau}{\rho H_D D_D^2 U_\infty^2} \quad (\text{Eq. 2.4})$$

$$\lambda = \frac{c_p}{c_t} = \frac{v}{U_\infty} = \frac{\omega D_D}{2U_\infty} \quad (\text{Eq. 2.5})$$

Furthermore, some dimensionless parameters, for instance, the Reynolds number (Re), solidity ratio (σ), and Froude number (Fr) also influence turbine performances. Principally, the Reynolds number (Re) used to determine whether the flow is laminar or turbulent is typically defined as the ratio of inertial force to viscous force as shown in an equation (Eq. 2.6), where X refers to the general length scale. In the flow regime of a hydrokinetic turbine, the Reynolds number based on diameter (D), or the foil chord (c) can be expressed in equations (Eq. 2.7) [17] and (Eq. 2.8) [16],[29],[30] respectively.

$$Re = \frac{\rho X v}{\mu} \quad (\text{Eq. 2.6})$$

$$Re = \frac{\rho D v}{\mu} \quad (\text{Eq. 2.7})$$

$$Re = \frac{\rho c v}{\mu} = \lambda \frac{\rho c U_{\infty}}{\mu} \quad (\text{Eq. 2.8})$$

The solidity ratio expressed in an equation (Eq. 2.9) [31],[32] is defined as the ratio of total blade chord length to the turbine circumference.

$$\sigma = \frac{Nc}{\pi D} \quad (\text{Eq. 2.9})$$

A non-dimensional Froude number (Fr) is defined as a ratio of inertial force to gravitational forces of liquid flow in an open channel. It is used to describe the different flow regimes or flow characteristics in an open channel, that is similar to the flow in typical irrigation canals or rivers. For a horizontal rectangular channel, the Froude number is usually defined as an equation (Eq. 2.10) where W is the top width. It can be interpreted that, the flow is called to be critical if $Fr = 1.0$, subcritical if $Fr < 1.0$, and supercritical if $Fr > 1.0$. The hydraulic behavior of flow in an open channel relies on whether the flow is subcritical or supercritical.

$$Fr = \frac{U}{\sqrt{g \frac{A_c}{h}}} = \frac{U_{\infty}}{\sqrt{gW}} \quad (\text{Eq. 2.10})$$

2.5 Fundamentals of Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics known as CFD is a tool used to solve and analyze complex fluid problems. It has been employed for difficult, complicated, expensive, or impossible solved problems [19] which cannot be solved by traditional methods or experiments. Besides that, it is also able to investigate, predict, or visualize the fluid flow behaviours before taking the place of the manufacturing processes. CFD simulations including two-dimensional (2D) and three-dimensional (3D) simulations are an affordable approach to simulating the flow field around a turbine model compared to the experimental approaches which are high cost, especially during the design optimization process.

Generally, 2D simulation is simpler than 3D simulation. it requires shorter times, however, it is less accurate for flows with complexities [33] with 6-8 % overestimation of results [19],[33],[34], and limited to the single plane case [33]. On the contrary, 3D simulation is necessary when a turbine model is not symmetrical along the rotational axis. Besides that, it can capture the 3D effects, for example, tip vortices and secondary flow [33]. It provides more accuracy of predicted results, but it also needs longer computational times and larger computer storage.

2.5.1 Governing Equations

To obtain realistic simulation results, the simulation models are crucial to take into consideration. Typically, the Reynold-Averaged Navier–Stokes (RANS) method is the fastest CFD approach widely utilized to solve the complex problems of flow across turbines. A set of basic fluid mechanic equations, for example, conservation of mass or continuity equation, momentum and energy transport equations known as Navier–stokes equations has been employed. The flow around the turbine models was assumed to be three-dimensional, turbulent, and transient incompressible fluid flow. Therefore, the continuity equation and conservation of momentum in the Cartesian coordinate system can be written in a compact form using tensor notation as equations (Eq. 2.11) and (Eq. 2.12) [35]–[37], respectively.

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \quad (\text{Eq. 2.11})$$

$$\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{P}}{\partial x_i} + \frac{\mu}{\rho} \frac{\partial^2 \bar{u}_i}{\partial x_j^2} - \frac{\partial}{\partial x_j} (\bar{u}'_i \bar{u}'_j) \quad (\text{Eq. 2.12})$$

The ANSYS Fluent available commercial software based on the finite volume method was utilized to carry out all three-dimensional simulations in this study. The time-dependent Unsteady Reynolds-Averaged Navier Stokes (URANS) equations with the pressure-based formulation were solved with the pressure-velocity coupling solution scheme using the Semi-Implicit Method for Pressure Linked Equation (SIMPLE) algorithm. Besides that, the spatial discretization of the pressure, momentum, and turbulence equations was used with the second-order upwind scheme.

2.5.2 Turbulence Model

There is no agreement on the most suitable model adopted in the CFD simulation. Hence, several turbulence models have been applied for 2D or 3D simulations of wind or hydro turbines. The most popular and simplest models of the two-equation series are the k - ε and k - ω turbulence models. The k - ε family computed the eddy viscosity in the RANS equations can solve two transport equations for turbulent kinetic energy (k) and turbulent dissipation rate (ε) as expressed in equation (Eq. 2.13) and (Eq. 2.14) [38]–[40], respectively.

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} + G_k - \rho \varepsilon \quad (\text{Eq. 2.13})$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j} \left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} \quad (\text{Eq. 2.14})$$

Where μ_t is the turbulent viscosity and can be expressed as an equation (Eq. 2.15) [39],[41]. G_k represents the generation of turbulence kinetic energy because of the mean velocity gradients [39]. Besides, C_μ , $C_{1\varepsilon}$, and $C_{2\varepsilon}$ are the model constants [41]. σ_k and σ_ε represent the turbulent Prandtl numbers for turbulent kinetic energy (k) and energy dissipation rate (ε) [39], respectively.

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \quad (\text{Eq. 2.15})$$

Basically, the $k-\varepsilon$ family consists of the standard $k-\varepsilon$ model suited for fully turbulent flow [39], the realizable $k-\varepsilon$ model [42]–[44] which is more advanced than the standard $k-\varepsilon$ model and recommended for rotating bodies, for swirling flows, and flows involving with separation [45], and the Renormalization Group (RNG) $k-\varepsilon$ model [46] modified by adding the effects of rotation on the eddy viscosity, employing an analytical formula for turbulent Prandtl numbers instead of constant values in the standard model and an additional term in ε for fast strained flows [22].

The other group is $k-\omega$ models incorporated the effect of compressibility and shear spreading. The omega term (ω) represents the specific dissipation rate of the turbulence kinetic energy (k) into internal energy (thermal) is a ratio of ε over k [41]. Principally, the $k-\omega$ model gives better results compared to the $k-\varepsilon$ model in which adverse pressure flow conditions are controlled. It provides good performances for the flow near boundary layers and low Re flows. For these models, the turbulent viscosity (μ_t) can be obtained by an equation (Eq. 2.16).

$$\mu_t = \alpha \rho \frac{k}{\omega} \quad (\text{Eq. 2.16})$$

Where, α is the damping factor provided dumping for flow regions with low Re numbers [41].

To combine the benefits of both the k - ε and k - ω models, the hybrid two-equation model known as the Shear Stress Transport (SST) k - ω has been developed with a new transport equation for ω . The SST k - ω model is a two-equation eddy viscosity model that combines the advantages of k - ω formulations in the near walls, or boundary layer of a turbine, with a standard k - ε model for free stream flows that are far away from the boundary layer zones. It is widely used [47],[48] because it shows better in terms of reliability and stability, and the best agreement with experiments [49] This model is appropriate for flows with strong adverse pressure gradients and flow separation [50]. The transport equation for the SST k - ω model can be written as equations (Eq. 2.17) and (Eq. 2.18) [35],[36].

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] + G_k - Y_k \quad (\text{Eq. 2.17})$$

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_i}(\rho \omega u_i) = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_i} \right] + G_\omega - Y_\omega \quad (\text{Eq. 2.18})$$

Where, G_ω represents the generation of ω . The Y_k and Y_ω represent the dissipation of turbulence kinetic energy and the dissipation of ω , respectively. A detailed summary of each model is explained in Table 2.3.

Table 2.3 : Summary of the CFD turbulence models [41]

Model	Number of equations	Turbulence variables	Characteristics
$k-\varepsilon$	2	Kinetic energy (k) dissipation rate (ε)	Empirical constants, over-predicts turbulence generation at locations with strong local accelerations
Realizable $k-\varepsilon$	2	Kinetic energy (k) dissipation rate (ε)	Removal of small-scale eddies, performs well with curved streamlines
RNG $k-\varepsilon$	2	Kinetic energy (k) dissipation rate (ε)	More sufficient to simulate flow in the boundary layer with an adverse pressure gradient
$k-\omega$	2	Kinetic energy (k) specific dissipation rate (ω)	Amplifier for boundary layer flow, sensitive for flow in free stream regions
SST $k-\omega$	2	Kinetic energy (k) specific dissipation rate (ω)	Combine the benefits of the $k-\varepsilon$ and $k-\omega$ models, applicable in the near w all and free stream flow regions

The optimized turbulence models have been investigated to consider the most accurate, reliable, and suitable model. Figure 2.9 shows the comparison between CFD results employed by several turbulence models and experimental results. As shown in Figure 2.9(a), Saad et al., 2020 [36] conducted 3D simulations with 4 different turbulence models. It was found that the predicted results using the SST $k-\omega$ turbulence model were more accurate and correlated with the experimental results of Fujisawa compared to other models.

As shown in Figure 2.9(b), 2D simulations on the Darrieus wind turbine with three different turbulence models including the $k-\omega$ SST, the transition SST, and the $k-kl-\omega$ turbulence models were carried out by Arab et al., 2017 [51]. The results can be concluded that using the $k-\omega$ SST or transition SST turbulence model showed better agreement with the experimental results.

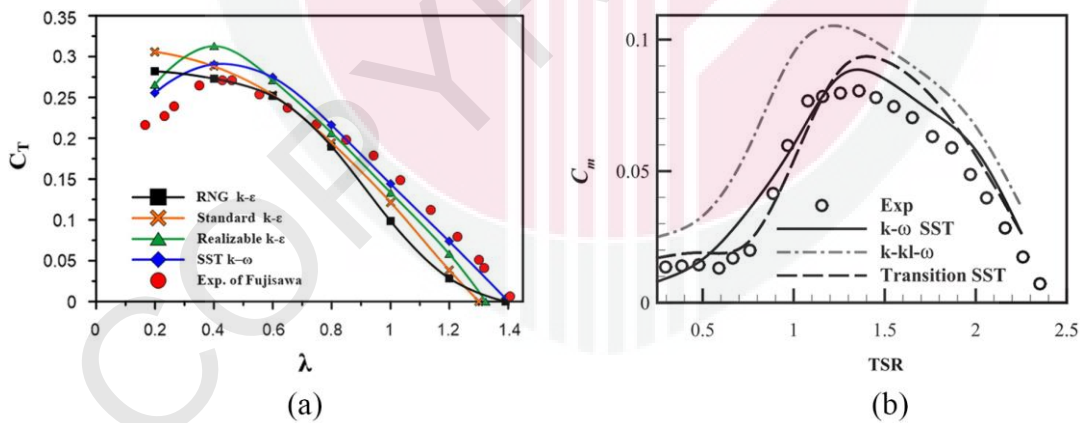


Figure 2.9 : Comparison between simulated results employed by several turbulence models and experimental results [36],[51]

2.5.3 Domain Independent Study

The computational domain representing a free flow region around a turbine model is crucial and influences the accuracy of CFD results. The dimensions of the domain are given in multiples of a turbine diameter (D_r). Because there is no agreement on the most suitable domain size. Hence, investigation of the optimal domain size is necessary to make sure that the lateral boundaries are located at sufficient distances and to avoid any interference in the determination of the flow field around the turbine model [52]. Typically, domain dependency studies can be conducted by two main methods. First, the domain size is tested by increasing its size step by step and observing the convergence of some criteria namely, instantaneous torque (τ), average torque coefficient ($c_{t,avg}$), or average power coefficient ($c_{p,avg}$). The second method, domain size can be determined by the information from former CFD studies.

Based on the published literature, the computational domain was divided into two main regions [49],[53], the first one was called a stationary region and the second was a rotating region that surrounded the turbine model. A rotating region could be a circle for two-dimensional (2D), or a cylinder for three-dimensional (3D) simulations. Both regions were separated by an interface to ensure the continuity of the flow field and the rotating region was allowed to rotate with several rotational speeds while the rest was kept fixed [23],[49],[52]–[54]. Plenty of publications extensively discussed domain size for 2D or 3D CFD simulations of the Savonius, Darrieus, and combined Savonius-Darrieus turbines. The domain sizes used from the previous studies are summarized in Table 2.4 for 2D and Table 2.5 for 3D simulations.

Table 2.4 : Domain sizes of 2D simulations used from previous studies

Region	Savonius	Darrieus	Combined
Rotating region	1.1D [55], 2D [56][57], 3D [58]	1D [51], 1.2D [59], 1.5D [53][60][61]	1.5D [42][62][63]
Upstream	3D [57], 6D [55][56]	3D [51], 5D [59][61] 10D [60]	5D [43][62][63], 10D [64]
Downstream	9D [56], 11D [57], 20D [55]	15D [51][59], 25D [60][61]	14D [64], 15D [62], 25D [63], 30D [43]
Width	6D [57][58], 12D [55][56]	8D [51], 10D [59], 20D [60][61], 25D [52][65][66]	4D [42], 8D [62], 15D [43], 20D [63]

Table 2.5 : Domain sizes of 3D simulations used from published literature

Region	Savonius	Darrieus	Combined
Rotating region	1.1D [67], 1.5D [54], 1.55D [68]	0.7D [69], 1.5D [33][70], 2D [71], 3D [72]	1.3D [46]
Upstream	5D [36], 20D [67]	4D [31][69][70][73], 5D [34][71], 10D [33]	5D [74]
Downstream	20D [36], 40D [67]	8D [69], 11D [34], 12D [31][70][73][75], 15D [33][71]	10D [74]
Width	10D [36], 40D [67]	3D [31][70] 5.3D [69], 6D [73], 10D [33][34][71][72]	4D[46] 8D [74]
Height	10D [36], 20D [67]	2.5D [72], 3D [71], 6D [73], 8D [34], 10D [33]	1.7D[46] 8D [74]

The optimum domain size is also important in CFD simulations. An oversized domain requires too much computer storage and simulation time. Whereas, the simulation results are interrupted by domain boundaries for a too small size of a domain. To get a suitable domain size, a domain independence study is required.

Arab et al., 2017 [51] conducted domain independency studies with six different cases of rotor distances of a 3-blade Darrieus wind turbine. The cases consisted of three different distances from the inlet boundary (1R, 6R, and 20R) and outlet boundary (10R, 30R, and 90R) as shown in Table 2.6 [51]. The results could be concluded that case B (6R and 30R) showed the convergence of the torque coefficient since the further increase of the distances did not significantly influence the simulation results. Therefore, 6R of the inlet distance and 30R of the outlet distance shown in Figure 2.10 [51] were the most optimum distances for the computational domain.

Table 2.6 : Domain independency study from the study of Arab et al., 2017 [51]

Parameter / Case	A	B	C	D	E	G
Distance from the inlet boundary	1R	6R	20R	6R	6R	20R
Distance from the outlet boundary	30R	30R	30R	10R	90R	90R
Torque coefficient	0.0728	0.054	0.0532	0.0894	0.0564	0.0562

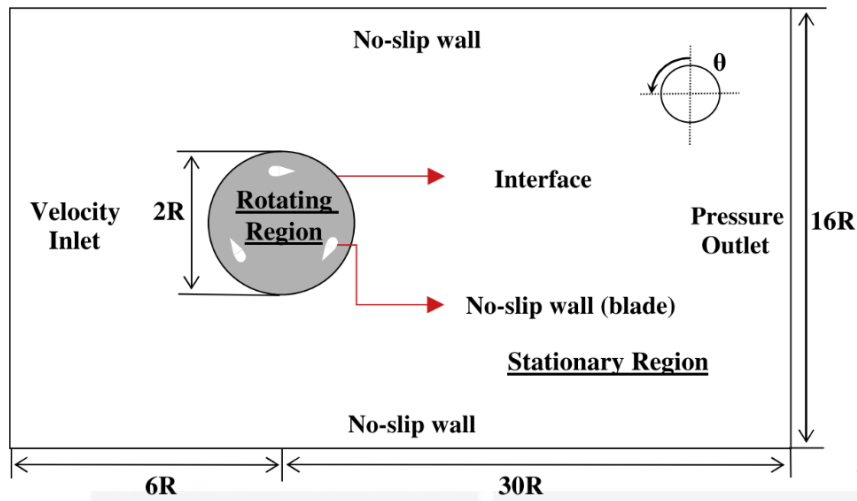


Figure 2.10 : 2D Computational domain and boundary conditions from the study of Arab et al., 2017 [51]

Liang et al., 2017 [43] investigated the optimal dimension for a 2D domain as depicted in Figure 2.11(a) [43] of the Darrieus-Savonius combined wind turbine. The distances of frontal (F), rear (r), and width (W) were tested. To consider the convergence of simulation results, moment (N-m) as a function of length ratio which was the normalized distance ($L/2R_D$) where R_D was the radius of the Darrieus rotor was plotted as presented in Figure 2.11(b) [43]. It could be concluded that the optimized domain dimensions were $5D$ of front distance, $15D$ of width, and $30D$ of rear distance.

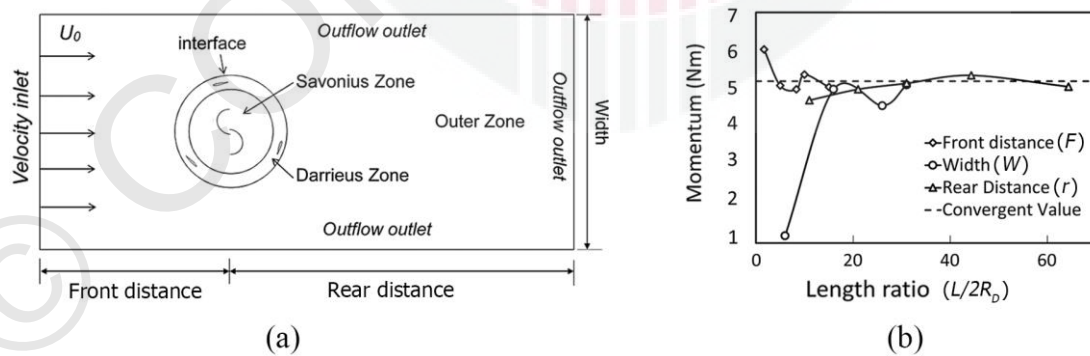


Figure 2.11 : (a) 2D-computational domain size, and (b) Computational domain analysis from the study of Liang et al., 2017 [43]

Furthermore, Bianchini et al., 2017 [76] mentioned the dimensions of the computational domain employed in their study. There were three different strategies as depicted in Figure 2.12(a) [76] that were analyzed and compared. The domain was defined and presented in terms of flow channel characteristics or a proportion of flow area. The domain having dimensions of $3.7D$ of width and $36D$ of length was called a tunnel which has been used widely by several researchers although the simulated results were not always in good agreement with experiments. Besides, the domain was defined as the equivalent blockage when the domain reproduced the similar ratio as available in experiments between the sectional area of the rotor, A , and the tunnel area, S as illustrated in Figure 2.12(b) [76]. This method with approximately $9D$ of width was employed to transpose boundary conditions from a $3D$ domain to a $2D$ domain. Finally, a domain was called an open field when the lateral boundaries of the domain were placed far enough ($36D$ of width) to avoid some effects of flow on the turbine model. These solutions were used by many researchers to ensure the accuracy of simulation results compared with the experiments.

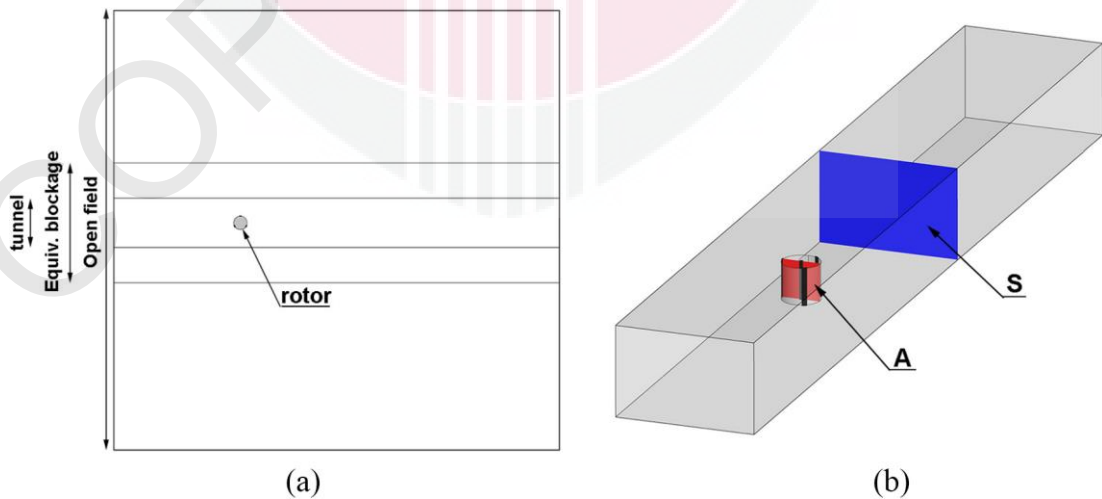


Figure 2.12 : Computational domain sensitivity analysis [76]

2.5.4 Mesh Sensitivity Analyses

In CFD simulations, the proper mesh resolution is extremely crucial to increase the reliability of predicted results. Practically, using high-quality or finer meshes can improve not only the rate of convergence and accuracy but also require more simulation times, larger computer storages, and computational costs and computational costs [22]. Low mesh quality however may contribute to meaningless results, or calculation errors [22].

Mesh generation can be carried out by several software. The ANSYS-Mesh module is utilized widely to generate mesh elements. Based on previous literature, there are three main methods in the ANSYS-Mesh module used to generate mesh elements whether for 2D or 3D simulations. The first one is using the mesh refinement categorized basically by coarse, medium, and fine [51],[77],[78]. The second one is using the refinement levels from level 1 to level 8 [39],[42],[44],[79]. As a consequence, the number of elements and nodes are generated automatically corresponding to those refinement categories, or levels. In the third method, the numbers of meshes are varied by mesh or element sizes manually [33],[43],[46],[80].

The mesh independency study can be considered by decreasing mesh size, or increasing mesh refinement levels leading to the increasing number of elements. The simulation processes have been repeated with different mesh sizes or number of elements until the simulated results have been converged. Usually, some parameters such as instantaneous torque (τ), average torque coefficient ($C_{t,avg}$), or average power coefficient ($C_{p,avg}$) are assessed and widely used to present the convergence of

simulation results. The optimum mesh size occurs when there is no significant change in the results after increasing of number of elements [33].

To consider whether the simulation results have already been converged or not, the repeated $c_{p,avg}$, or $c_{t,avg}$ were calculated and considered. In fact, when the convergent criterion (ε) expressed in equations (Eq. 2.19) and (Eq. 2.20) is less than 1% [33], or in equation (Eq. 2.21) is less than 0.02 [30] at least two consecutive revolutions, it means that the simulation results have already reached the convergence criterion.

$$\varepsilon = \frac{c_{p,avg(n+1)}(\theta) - c_{p,avg(n)}(\theta)}{c_{p,avg(n)}(\theta)} < 1\% \quad (\text{Eq. 2.19})$$

$$\varepsilon = \frac{c_{T,avg(n+1)}(\theta) - c_{T,avg(n)}(\theta)}{c_{T,avg(n)}(\theta)} < 1\% \quad (\text{Eq. 2.20})$$

$$\text{Convergence} = \frac{\text{Average torque at each time step}}{\text{Average torque per one revolution}} \leq 0.02 \quad (\text{Eq. 2.21})$$

By using the mesh refinement categories, mesh independency studies could be determined for example from the convergence of torque per unit blade length from the 2D study of Nobile et al. 2014 [77] as presented in Figure 2.13(a), instantaneous torque coefficient versus the rotor angular position (θ) from the study of Arab et al. 2017 [51] as depicted in Figure 2.13(b), or power coefficient as a function of θ from the study of Elkhoury et al. 2015 [78] as illustrated in Figure 2.13(c).

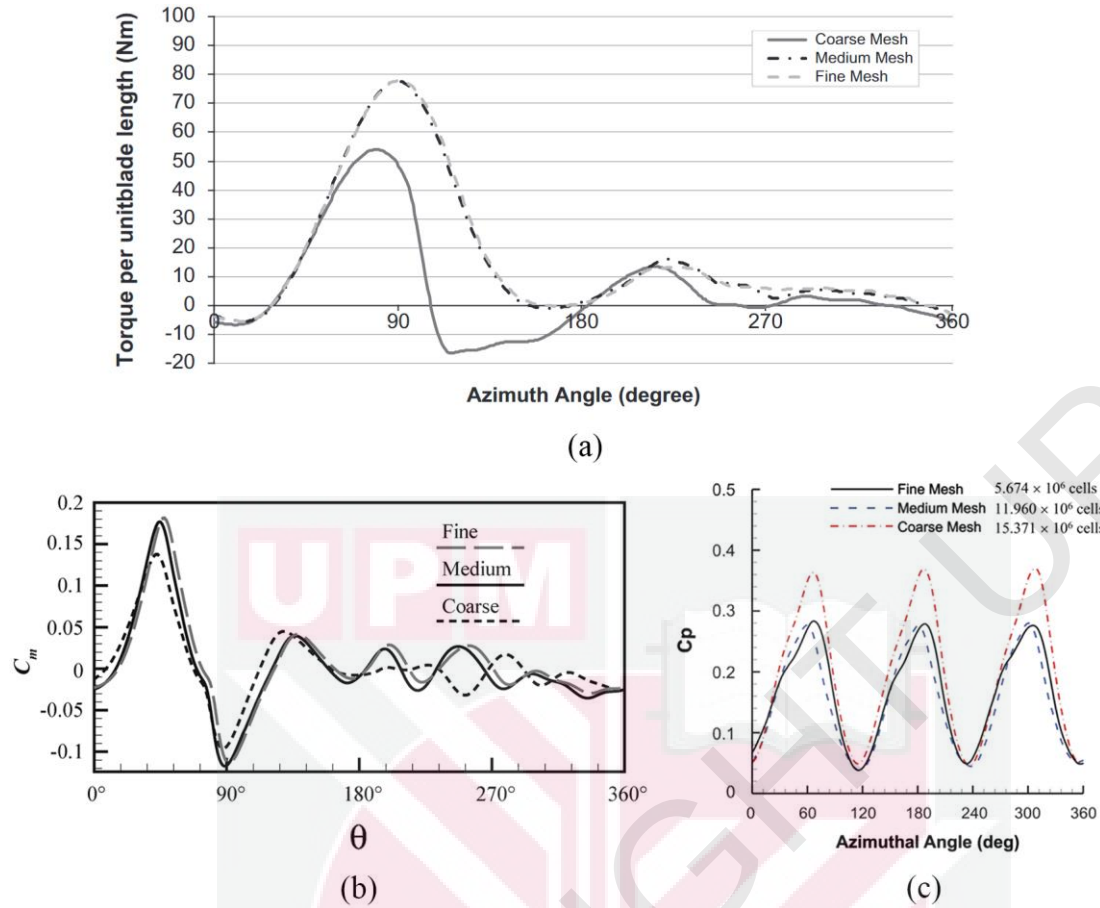


Figure 2.13 : Mesh independency studies [77],[51],[78]

Mesh independency results using the refinement levels were carried out by the 3D CFD simulations of N.K. Sarma et al., 2014 [79], and 2D simulations of Saini & Saini, 2018 [42] as depicted in Figure 2.14(a) [79], and (b) [42], respectively. Moreover, Figure 2.15(a) also represents mesh sensitivity analysis by utilizing a variety of numbers of elements from the CFD studies of A. Orlandi et al., 2015 [80] using the 2D and 3D model of 17-m. Darrieus H-Type VAWT. Figure 2.15(b) shows mesh sensitivity results from the 2D-CFD study of X. Liang et al., 2017 [43].

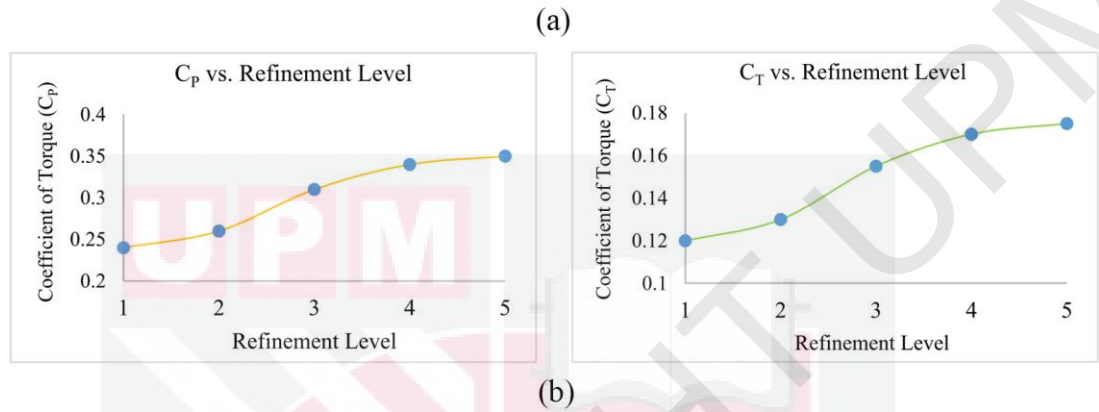
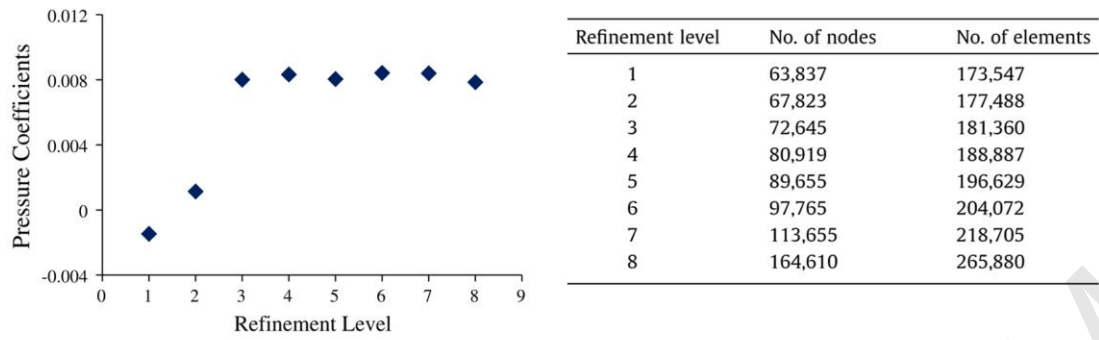


Figure 2.14 : Mesh independency results by using the refinement levels [42],[79]

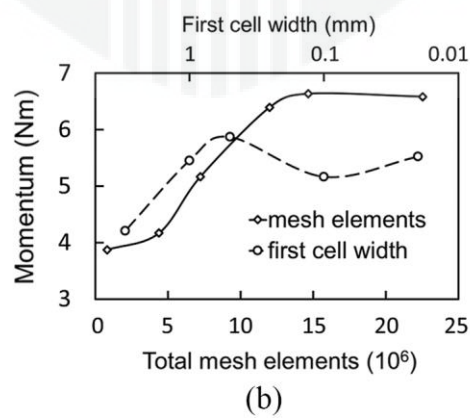
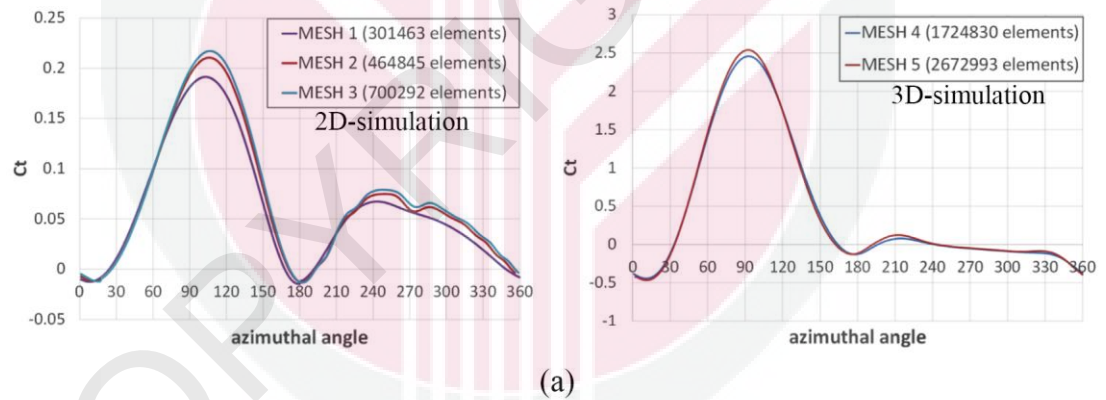


Figure 2.15 : Mesh sensitivity analysis by utilizing a variety of numbers of elements [43],[80]

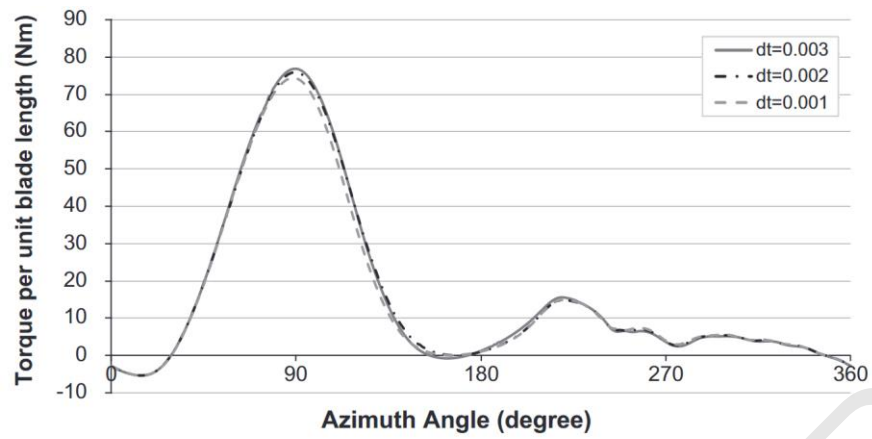
2.5.5 Time Step Independency Study

The time step is an important factor for the accuracy of simulated results. To optimize the value of time step size (Δt) expressed in an equation (Eq. 2.22) [35],[37], the simulation processes are repeated with different values of Δt to decrease computational time, but the result accuracy is still acceptable. Figure 2.16(a) [77] shows the variation of instantaneous torque per unit blade length on a single rotor blade along with azimuth angles for three different time steps.

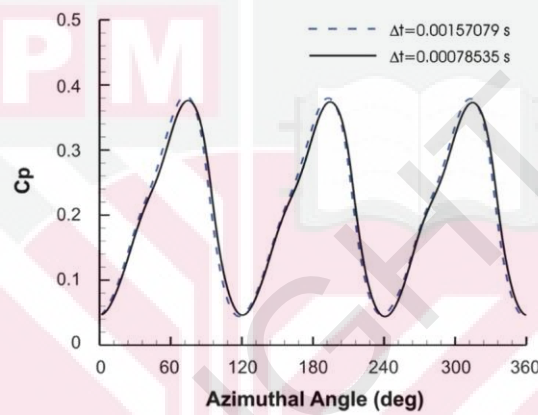
Figure 2.16(b) [78] depicts the turbine power coefficient (c_p) as a function of azimuth angle with two different time steps. It can be seen clearly that the results in each example are extremely close. Consequently, using the larger time step size (Δt) would not decrease the accuracy of the simulated results.

$$\Delta t(1^\circ) = \frac{\Delta \theta \pi}{180^\circ \omega_o} \quad (\text{Eq. 2.22})$$

Where, $\Delta \theta$ is the angular step ($^\circ$) and ω_o is the angular velocity.



(a)



(b)

Figure 2.16 : Time dependency study [77],[78]

2.6 Savonius Hydrokinetic Turbine

A conventional Savonius hydrokinetic turbine (SHKT) is a low-speed drag-based vertical axis turbine. It typically comprises two buckets. The first one calls an advancing bucket and another calls a returning bucket. Each bucket consists of a concave and a convex side. Schematic views including isometric, and top views of a SHKT are shown in Figure 2.17.

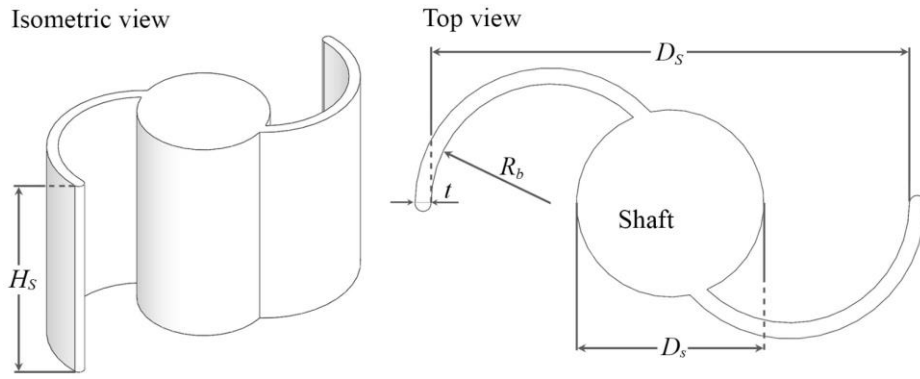


Figure 2.17 : Schematic views of a conventional Savonius hydrokinetic turbine

According to previous studies, it was found that the performance of the Savonius turbine depends on three main factors which are geometric parameters, operational conditions, and fluid flow parameters [16]–[18] and can be written as a functional form as shown in an equation (Eq. 2.23).

$$C_{p,avg} = f(\text{operational conditions; geometric parameters; flow parameters}) \quad (\text{Eq. 2.23})$$

In general, efficiency and self-starting capability are the main concerns. Several turbine parameters have been experimentally and numerically investigated, and optimized. Researchers found and concluded that a Savonius either wind or hydro turbine has a good starting ability at lower flow speeds [17],[23],[81]. However, it suffers with the low performance [8],[16],[19],[82]. In order to overcome the aforementioned problems and enhance the turbine performances, many parameters as illustrated in Figure 2.18 [17],[36],[82] of the Savonius turbines have been investigated. The details of those findings will be explained in the following section.

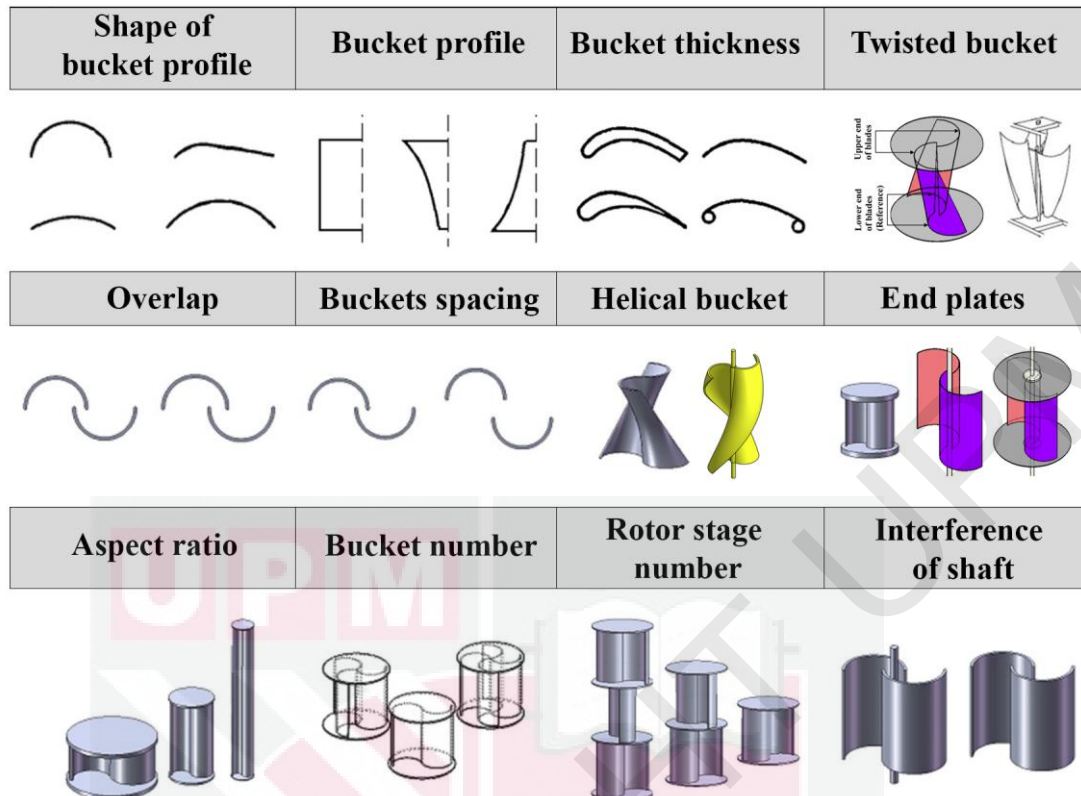


Figure 2.18 : Studies on some geometric parameters [17],[36],[82]

2.6.1 Effect of Bucket Number

The effect of bucket number was conducted numerically and experimentally by using either the Savonius wind or hydro turbines. Two and three semicircular-bucket Savonius wind turbines were simulated by Zhao et al., 2009 [38]. Those simulation results are shown in Figure 2.19(a) and (b). Besides that, hydro turbines with two and three semicircular buckets were carried out experimentally in field tests by Talukdar et al., 2018 [58]. Those finding results are shown in Figure 2.19(c). Furthermore, Mahmoud et al., 2012 [83] conducted experimental testing on two, three, and four semicircular-bucket wind turbines, and the performance characteristics are presented in Figure 2.20.

Based on all studied cases mentioned above, it can be concluded that the Savonius wind or hydro turbine with 2 buckets performs better than that one with 3, or 4 buckets. There are some reasons why a 2-bucket turbine shows better performance. Zhao et al. [38] explained that when the number of buckets is increased from two to three, or four, the medium fluid struck on the advancing bucket would be reflected onto the following bucket. The following bucket hence obtains the negative torque direction and tends to rotate in the opposite direction as compared to the previous bucket. This phenomenon can be presented by vector lines as illustrated in Figure 2.19(a) [38].

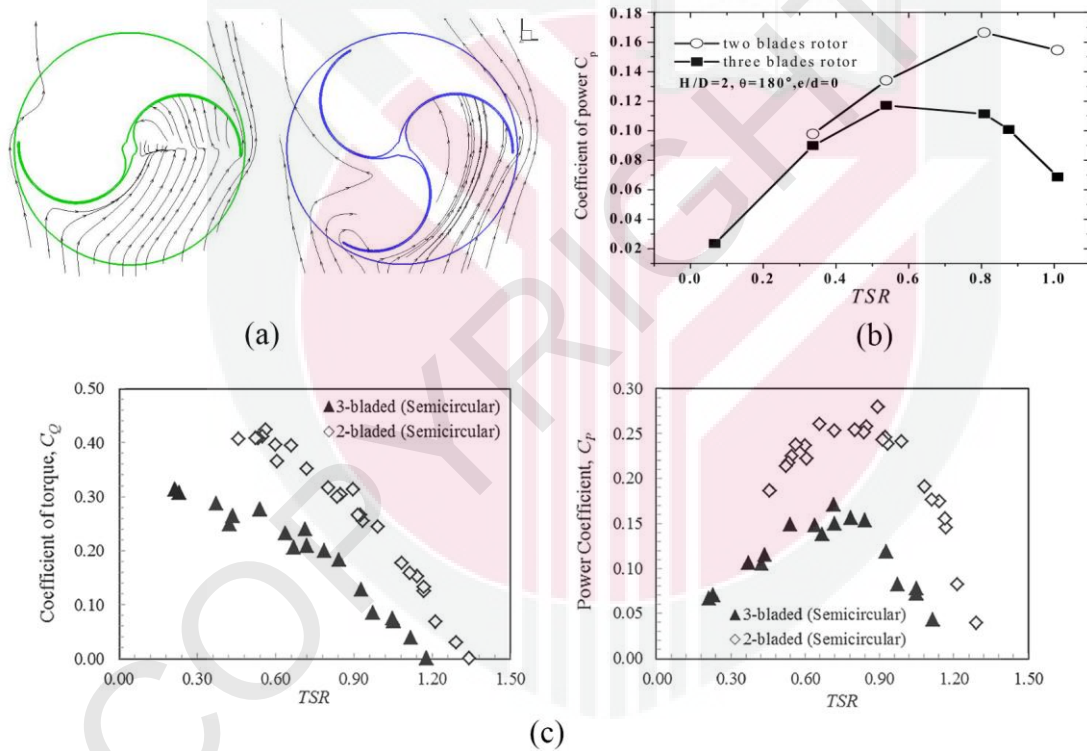


Figure 2.19 : (a) The flow fields of two and three buckets Savonius turbines, (b)-(c) The performance curves of the rotors with two- and three-blades [38],[58]

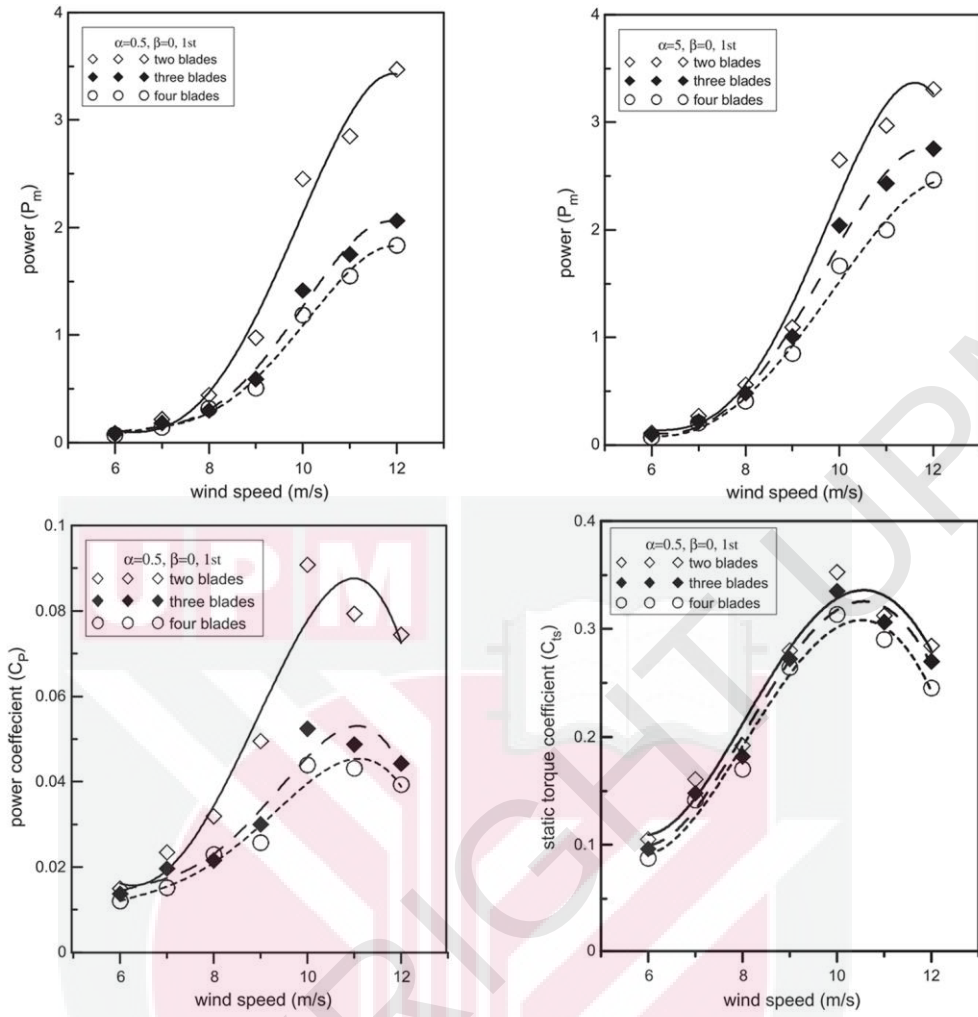


Figure 2.20 : Variation of mechanical power, power coefficient, and static torque coefficient with wind speed from different numbers of blades [83]

2.6.2 Effect of Shape of Bucket Profile

Shapes of bucket profiles have been determined so far to enhance the performances of a Savonius turbine. From the experimental studies of Talukdar et al., 2018 [58], the 2-bucket Savonius hydrokinetic turbines with the elliptical and semicircular shapes of bucket profiles (with $\beta = 0.15$, $\alpha = 1.0$, and $D_o = 1.1D$) were tested.

The results depicted in Figure 2.21 [58] could be concluded that the semicircular bucket profile provided better performances compared to the one with the elliptical

bucket profile. Apart from the experiment, 2D CFD simulations were also carried out to demonstrate the mechanism of fluid flows passing through the turbine buckets and describe why a turbine with a semicircular bucket profile provides better performance in comparison to an elliptical one. Besides, the blade-wake interaction or vorticity distribution around turbine blades could be represented by the vorticity contours as illustrated in Figure 2.22 [58].

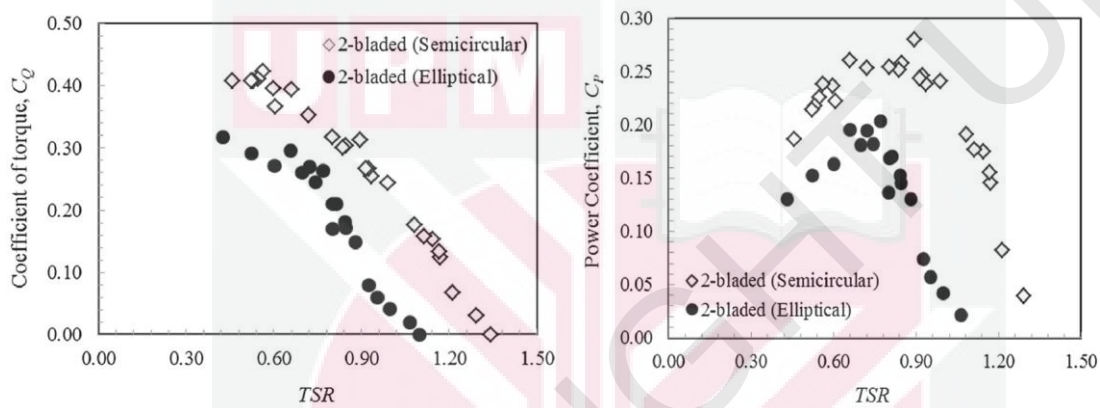


Figure 2.21 : Experimental investigation on the effect of the shape of bucket profile from the studies of Talukdar et al., 2018 [58]

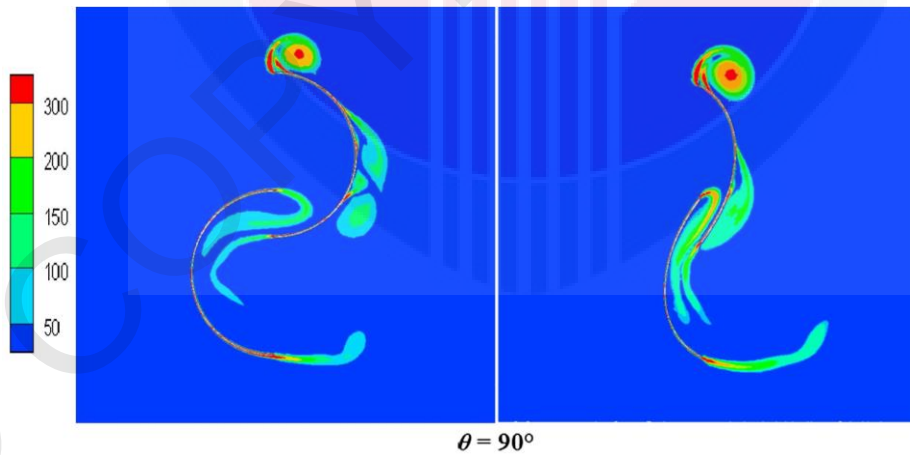


Figure 2.22 : Vorticity contours for semi-circular and elliptical-bladed turbines from the studies of Talukdar et al., 2018 [58]

The contour plots in Figure 2.23 [58] show the velocity vectors around the turbine buckets at different azimuthal angles (θ), and wake interactions or the distributions of vorticity generated from both cases either elliptical or semicircular at the tip of buckets, overlap areas, and at the downstream of turbine buckets.

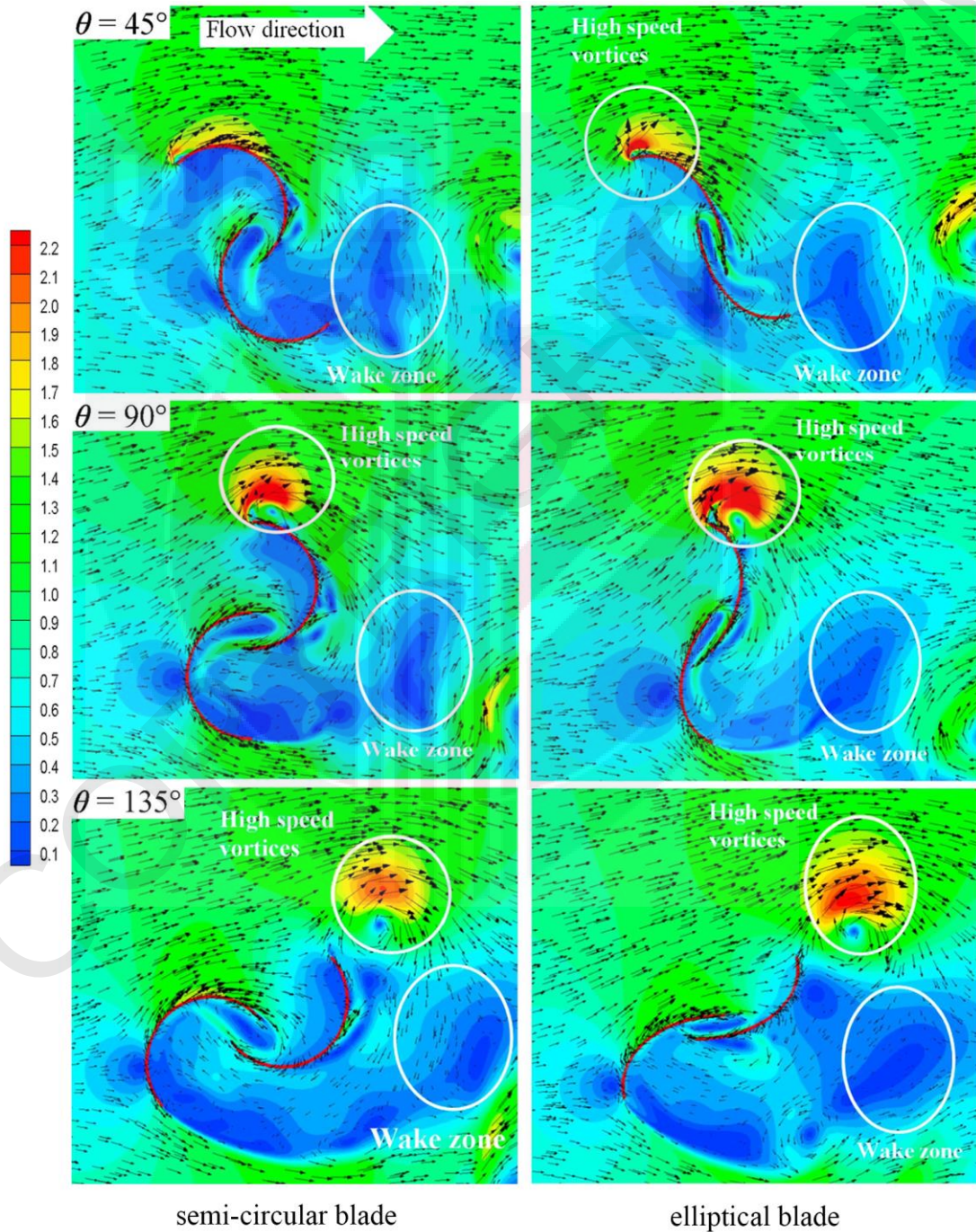


Figure 2.23 : Velocity vector plots of semicircular and elliptical bucket shapes from the studies of Talukdar et al., 2018 [58]

As mentioned above, it can be clarified the performance decrement of a Savonius turbine with the elliptical bucket profile compared to the semicircular bucket profile.

2.6.3 Effect of End Plates

End plates can be added to the turbine to prevent the leakage of fluid flowing from the concave side of the buckets out to the ambient flow. Then, the difference in pressure between the concave and convex sides of the turbine buckets is kept at satisfactory levels over the rotor height [17]. As a consequence, turbine performances can be improved and can be operated more efficiently with a wider range of tip speed ratio as depicted in Figure 2.24(a) [17].

Mahmoud et al., 2012 [83] conducted wind tunnel tests using the Savonius turbines with and without end plates and found that a turbine with end plates performed better than one without end plates as presented in Figure 2.24(b) [83]. Moreover, the effect of end plate diameters (D_o) was examined by Saad et al., 2020 [36] using turbines with five different endplate diameters. The results in Figure 2.24(c) [36] showed that the optimum end plate diameter (D_o) was 1.1 times of rotor diameter (D_r) [36].

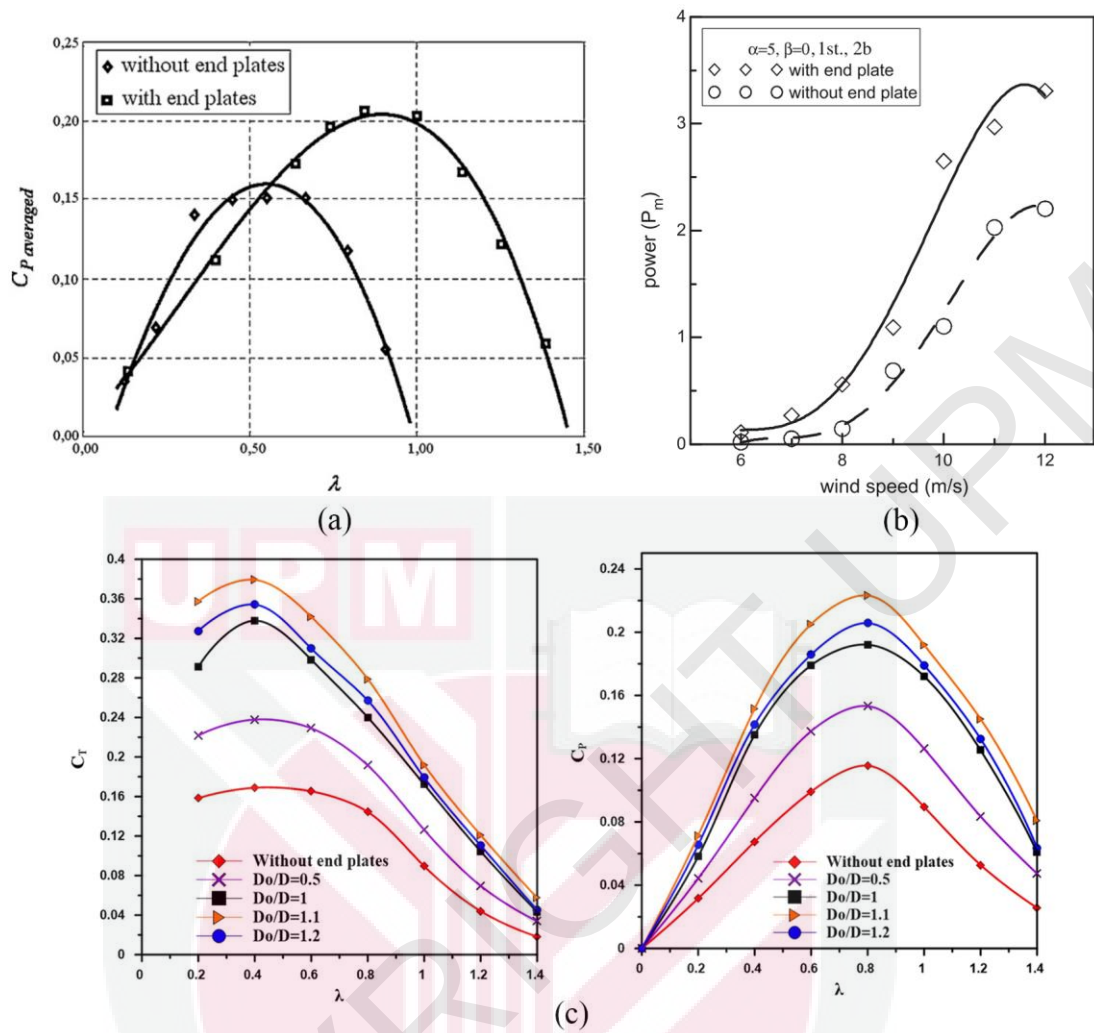


Figure 2.24 : (a), and (b) effect of the presence of end plates [17],[83], (c) c_p and c_t curves from the study of Saad et al., 2020 [36]

Jeon et al., 2015 [82] experimentally studied the shape designs and sizes of end plates using helical Savonius wind turbines with a diameter of 250 mm. and wind speed of 10 m/s. There were 5 different end plate designs as illustrated in Figure 2.25 [82] which were (a) the turbine with no end plate, and (b)-(e) with end plate styles 1 to 4, respectively. The results in Figure 2.26 [82] including the coefficient of power and torque were measured and plotted. It found that curves of power and torque coefficient changed significantly with variations of end plate shape designs. The end plates style 4 with the circular shape provided better performances than the rest designs.

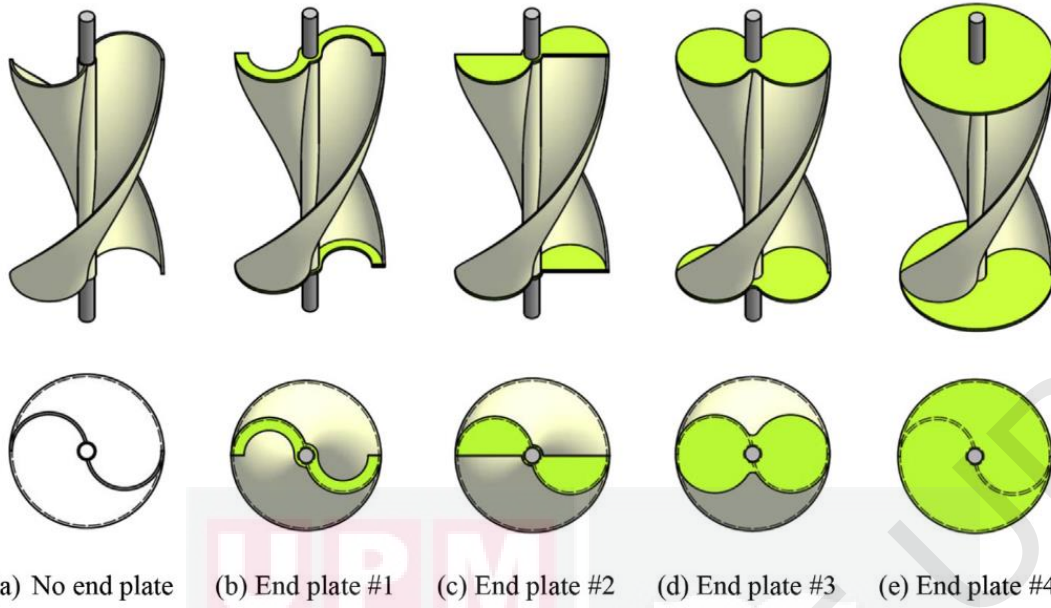


Figure 2.25 : Helical Savonius designs with various shape designs and sizes of end plates from the study of Jeon et al., 2015 [82]

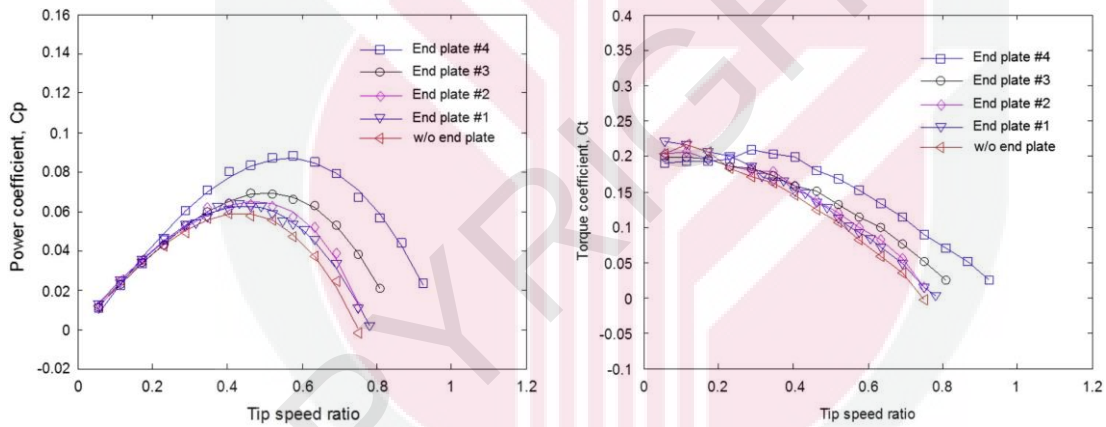


Figure 2.26 : c_p and c_t curves from various shape designs and sizes of end plates from the study of Jeon et al., 2015 [82]

In addition, to clarify the influence of end plates on turbine performances, the effects of different end plate configurations such as without end plates, with lower end plate only, and with both the upper and lower end plates were also considered in the studies of Jeon et al., 2015 [82] using the turbines with a diameter of 200 mm. and wind speed of 10 m/s.

The finding results shown in Figure 2.27 display that a turbine with both upper and lower end plates provides better performances than the rests, even if the performance curves of all various end plate conditions lie close together in the range of low tip speed ratio.

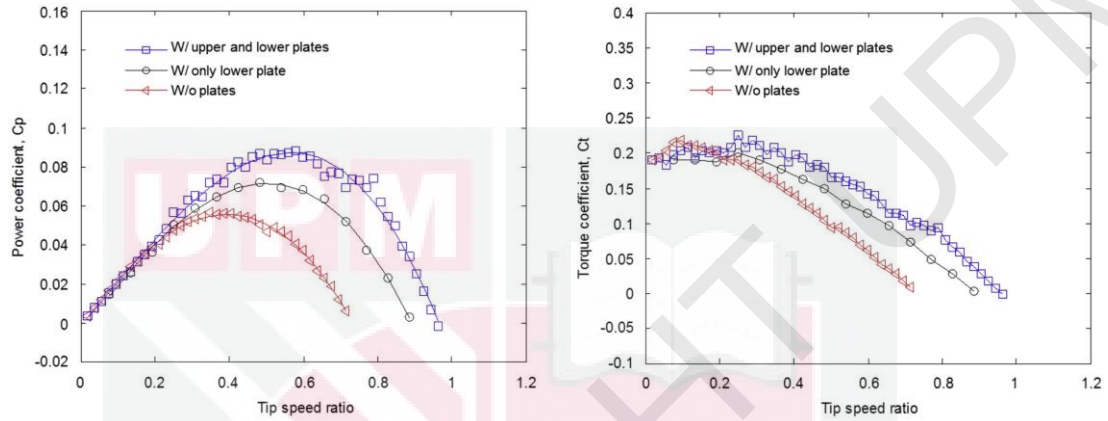


Figure 2.27 : c_p and c_t curves from various end plate configurations from the study of Jeon et al., 2015 [82]

2.6.4 Effect of Aspect Ratio

The aspect ratio (AR or α) of typical Savonius turbines is principally defined as a ratio of turbine height (H_s) and its diameter (D_s). Plenty of researchers have investigated experimentally and numerically so far on this parameter. Results from previous studies are summarized in Table 2.7.

The numerical and experimental results on the effect of aspect ratios are also illustrated in Figure 2.28(a) [38] and (b) [83], respectively. The finding showed that a turbine with higher AR performed better, the further increasing of an aspect ratio over the maximum value of AR might degraded the turbine performance.

This finding was consistent with experimental studies of Kamoji et al., 2009 [84],[85] as presented in Figure 2.29(a) and (b).

Table 2.7 : Summary of previous works on the effect of aspect ratios

Researchers	Methodology	Turbine	Range used	Optimum α , c_p , and c_t
Zhao et al., 2009 [38]	3D Numerical, Wind turbine	Two bladed 180° helical Savonius rotors	1 - 7	$\alpha = 6$ $c_p = 0.20$
Mahmoud et al., 2012 [83]	Experimental, Wind turbine	Two bladed semicircular	0.5 - 5.0	$\alpha = 5.0$ $c_p = 0.20$
Kamoji et al., 2009 [84]	Experimental, Wind turbine	A modified Savonius rotor, without a shaft	0.6, 0.7, 0.77, and 1.0	$\alpha = 0.7$ $c_p = 0.21$, $c_t = 0.30$ at $Re = 1.5e^5$
Kamoji et al., 2009 [85]	Experimental, Wind turbine	A 90° twisted helical Savonius rotor	0.88, 0.93 and 1.17	$\alpha = 0.88$ $c_p = 0.165$ at $Re = 1.2e^5$

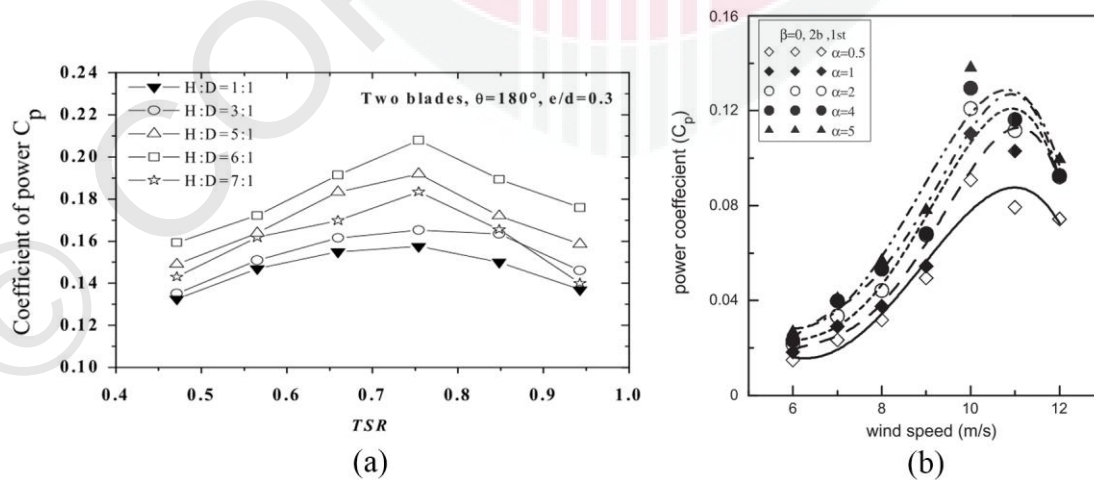


Figure 2.28 : (a) Computational and (b) experimental results on the effect of aspect ratios [38],[83]

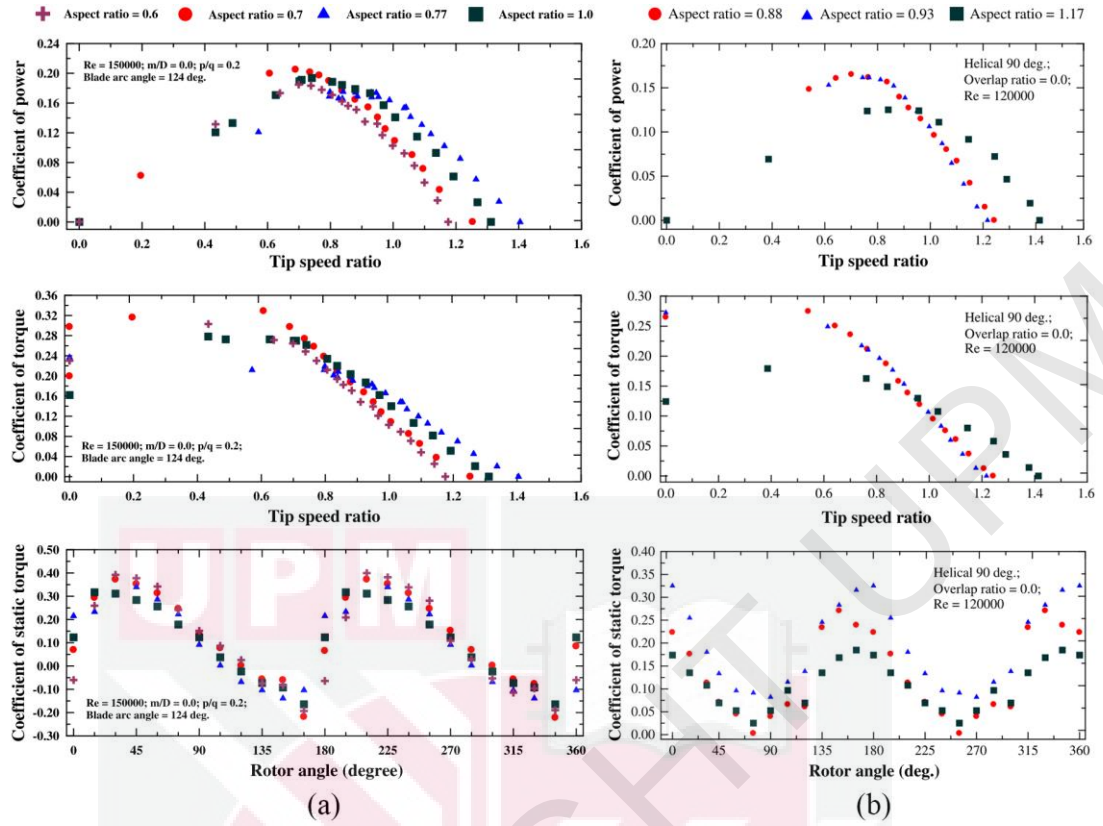


Figure 2.29 : Experimental results using the modified Savonius rotors with different aspect ratios [84],[85]

2.6.5 Effect of Overlap Ratio

For a conventional Savonius turbine, the overlap ratio (β) is defined as a ratio of the space or an overlap length (e) between two buckets to the bucket diameter (D_b) if that bucket is semicircular in shape. In case the turbine has no shaft as depicted in Figure 2.30(a) and (b), β can be expressed as in an equation (Eq. 2.24) [55],[86],[87]. In case the turbine has a shaft with a diameter (D_s) as illustrated in Figure 2.30(c) and (d), β can be determined by (Eq. 2.25) [83],[86],[87]. It means that β equals zero when (1) a turbine has no spacing between buckets ($e = 0$), or (2) in case of a turbine having a full center shaft ($D_s = e$), and β equals to e/D_b in case of a turbine has no shaft.

Many previous studies as displayed in Table 2.8 have been conducted on this parameter using experimental and numerical approaches. It found that the optimum turbine performances occurred when β was equaled to zero as shown in Figure 2.31, and was not equaled to zero as depicted in Figure 2.32 and Figure 2.33.

$$\beta = \frac{e}{D_b} \quad (\text{Eq. 2.24})$$

$$\beta = \frac{e - D_s}{D_b} \quad (\text{Eq. 2.25})$$

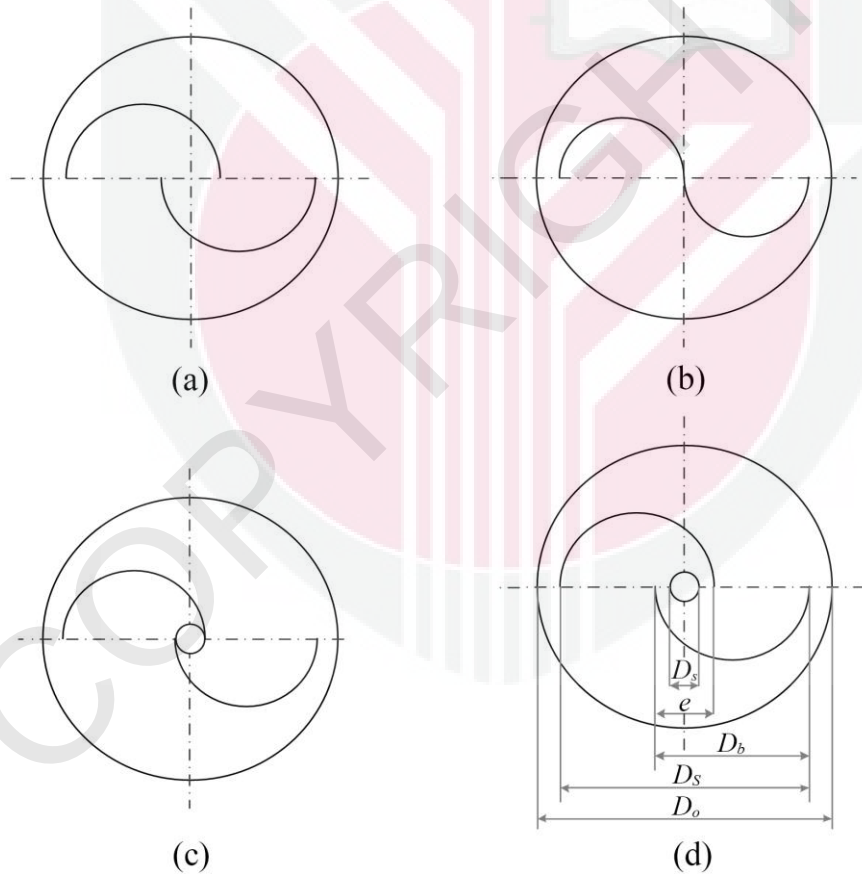


Figure 2.30 : Sketches of different overlap configurations

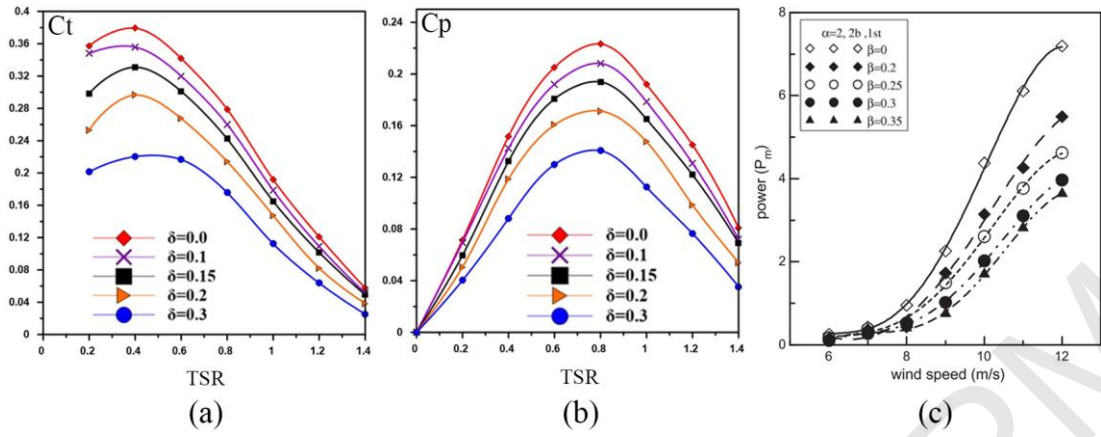


Figure 2.31 : The maximum turbine performances occurred when β was equal to zero [36],[83]

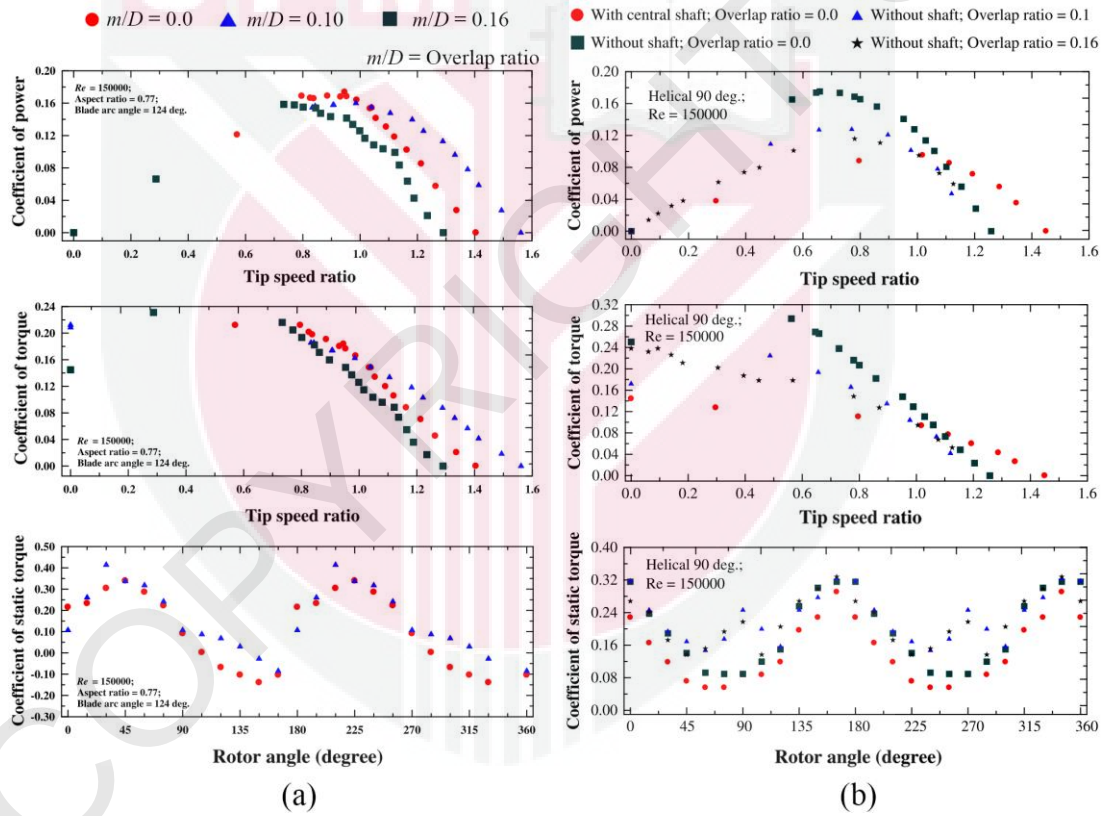


Figure 2.32 : The maximum turbine performances occurred when β was not equal to zero [84]

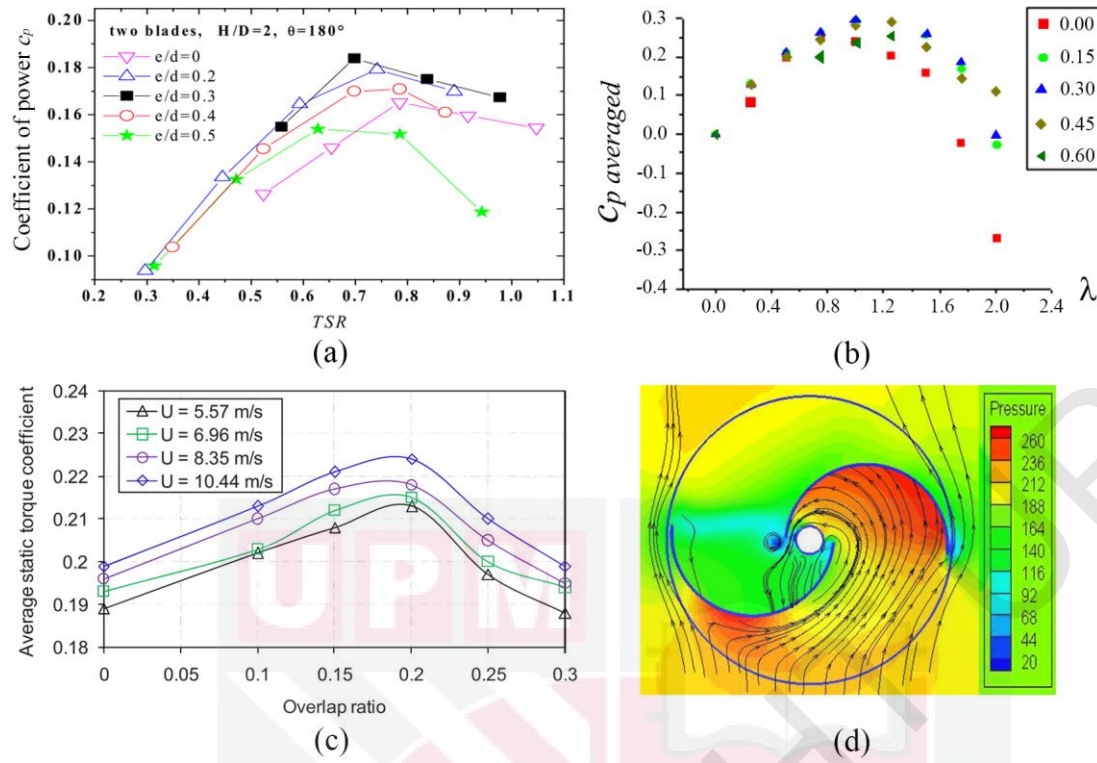


Figure 2.33 : (a) – (c) Numerical results show the maximum turbine performance occurred when β was not equal to zero [38],[55],[88], and (d) contour of the pressure of a turbine with a shaft and spaces [38]

Table 2.8 : Summary of previous research works with various overlap ratios of Savonius rotors

Researchers	Methodology	Turbine	Range used	Optimum β	Optimum c_p and c_t
Saad et al., 2020 [36]	3D Numerical, Wind turbine	A 45° twisted	0, 0.1, 0.15, 0.2, 0.3	0	$c_p = 0.223$, $c_t = 0.38$ at $U_\infty = 6$ m/s
Mabrouki et al., 2014 [89]	Experimental, Hydro turbine	Two bladed Savonius turbines	0, 0.2, 0.3	0.3	$c_p = 0.327$ at $\lambda = 2.51$ $Re = 588,300$
Roy et al., 2013 [88]	2D Numerical, Wind turbine	Two bladed Savonius turbine	0, 0.1, 0.15, 0.2, 0.25, 0.3	0.2	$c_{ts} = 0.224$ at $U_\infty = 10.44$ m/s
Akwa et al., 2012 [55]	2D Numerical, Wind turbine	A two-bladed semi-circular rotor	0, 0.15, 0.30, 0.45, 0.60	0.15	$c_p = 0.3161$ at $\lambda = 1.25$
Mahmoud et al., 2012 [83]	Experimental, Wind turbine	Two bladed semi-circular	0, 0.2, 0.25, 0.3, 0.35	0	-
Kianifar et al., 2011 [90]	Experimental & Numerical Wind turbine	Two bladed Savonius turbines	0, 0.20, 0.24, 0.40, 0.45	0.2	$c_p = 0.254$ at $Re = 1.5 \times 10^5$
Yaakob et al., 2010 [91]	3D Numerical, Hydro turbine	A turbine with double stages	0.1 - 0.6	0.21	$\tau_{avg} = 0.1362$ at $U_\infty = 0.169$ m/s
Zhao Z. et al., 2009 [38]	3D Numerical, Wind turbine	Two bladed 180° helical rotors	0.0, 0.2, 0.3, 0.4, 0.5	0.3	$c_p = 0.181$ at $\lambda = 0.73$
Kamoji et al., 2009 [84]	Experimental, Wind turbine	A modified rotor, without shaft	0, 0.1, 0.16	0	$c_p = 0.17$ at $Re = 1.5 \times 10^5$
Kamoji et al., 2009 [85]	Experimental, Wind turbine	A 90° twisted helical rotor	0, 0.1, 0.16	0	$c_p = 0.175$ at $\lambda = 0.9$, $Re = 1.5 \times 10^5$
Menet et al., 2004 [92]	2D Numerical, Wind turbine	2 buckets rotor	0.1-0.5	0.242	$c_t = 0.33$ at $Re = 1.56 \times 10^5$

2.6.6 Effect of the Number of Stages

Numbers of stages known as multi-staging also affect Savonius turbine performances. To optimize the number of stages, several configurations have been carried out by varying numbers of stages such as one, two, and three stages. According to previous research works, the stage designs of the Savonius turbine could be distinguished into two main designs based on the stage aspect ratio (AR) by comparing with a single-stage turbine. The first was a multi-stage turbine with varied stage aspect ratios and the second design was a turbine with a constant stage aspect ratio as illustrated in Figure 2.34. The summary of published research works on the number of stages is presented in Table 2.9 and details of the studies are described as follows.

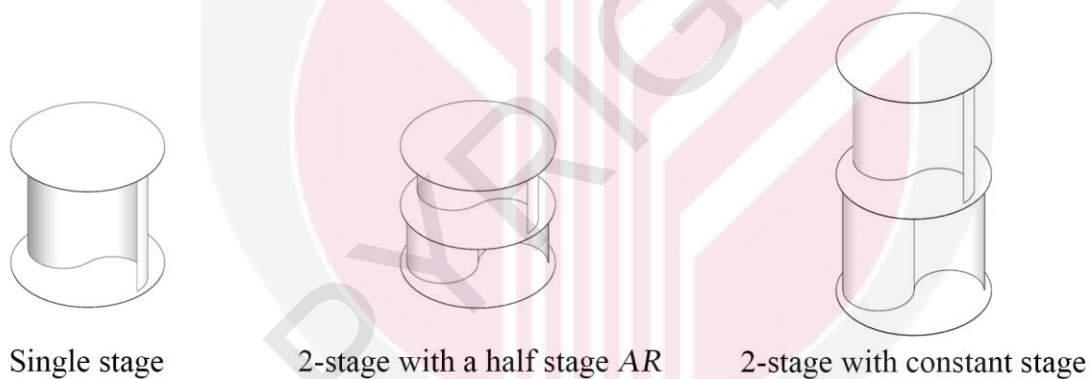


Figure 2.34 : Multiple stage configurations of Savonius turbine design

Table 2.9 : Summary of published works on the effect of the number of stages

Researchers	Methodology	Turbine	Detail of Stage	Finding
Mahmoud et al., 2012 [83]	Experiment, Wind turbine	2 buckets, semi-circular	1 and 2 stages	A two-stage rotor was better than a single stage.
Saha et al., 2008 [93]	Experiment, Wind turbine	2 and 3 buckets, semi-circular and twisted- bucket	1, 2 and 3 stages	The optimum stage was two.
Hayashi et al., 2005 [94]	Experiment, Wind turbine	2 buckets, semi-circular	1 and 3 stages with the different stage aspect ratio	One stage provided higher c_p , c_t but 3 stages provided less fluctuation of c_t , c_{ts}
Kamoji et al., 2008 [95]	Experiment, Wind turbine	2 buckets, semi-circular	1, 2, and 3 stages with a constant, and different stage aspect ratio	c_p , c_t decreased when the numbers of stages were increased for constant and different stage aspect ratios, 3 stages provided lower peak fluctuation of c_{st}

Mahmoud et al., 2012 [83] carried out wind tunnel testing on the single and double stages 2-bucket Savonius turbines with the constant overlap ratio ($\beta = 0$) and aspect ratio ($\alpha = 1$). The results in Figure 2.35 presented that a double-stage rotor provided higher specific power (W/m^2) defined as the power obtained from unit projected area of the rotor, and static torque coefficient (c_{ts}) compared to a single-stage rotor.

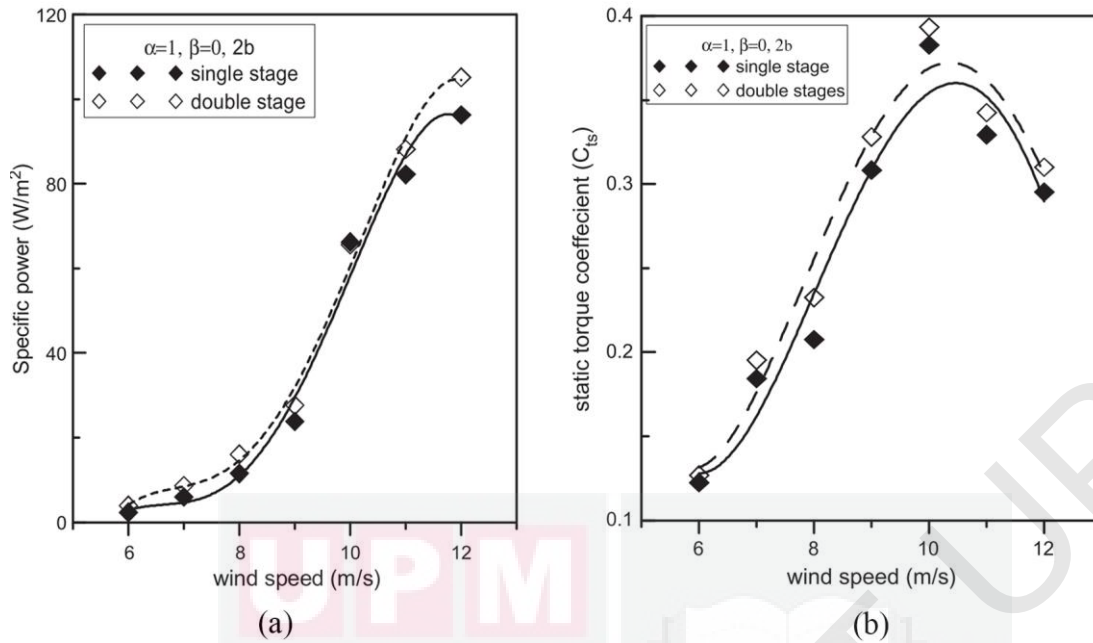


Figure 2.35 : Specific power and static torque coefficient of Savonius turbines with single and double stages [83]

Saha et al.,2008 [93] studied experimentally on the number of stages in a wind tunnel. The numbers of stages (from one to three) were varied by using turbines with two and three semi-circular (2sc, 3sc), and twisted (2tw, 3tw) bucket Savonius turbines. The results presented in Figure 2.36 show that the power coefficient of both sc and tw buckets increased when the number of stages was increased from one to two, the power coefficient nevertheless became worse when the number of stages was increased from two to three. Saha et al.,2008 analysed that the power coefficient (c_p) for both designs of the turbine was dropped because of the increase of rotor inertia from two to three stages. From these experimental studies, it was seen clearly that the optimum number of stages for a Savonius turbine was two.

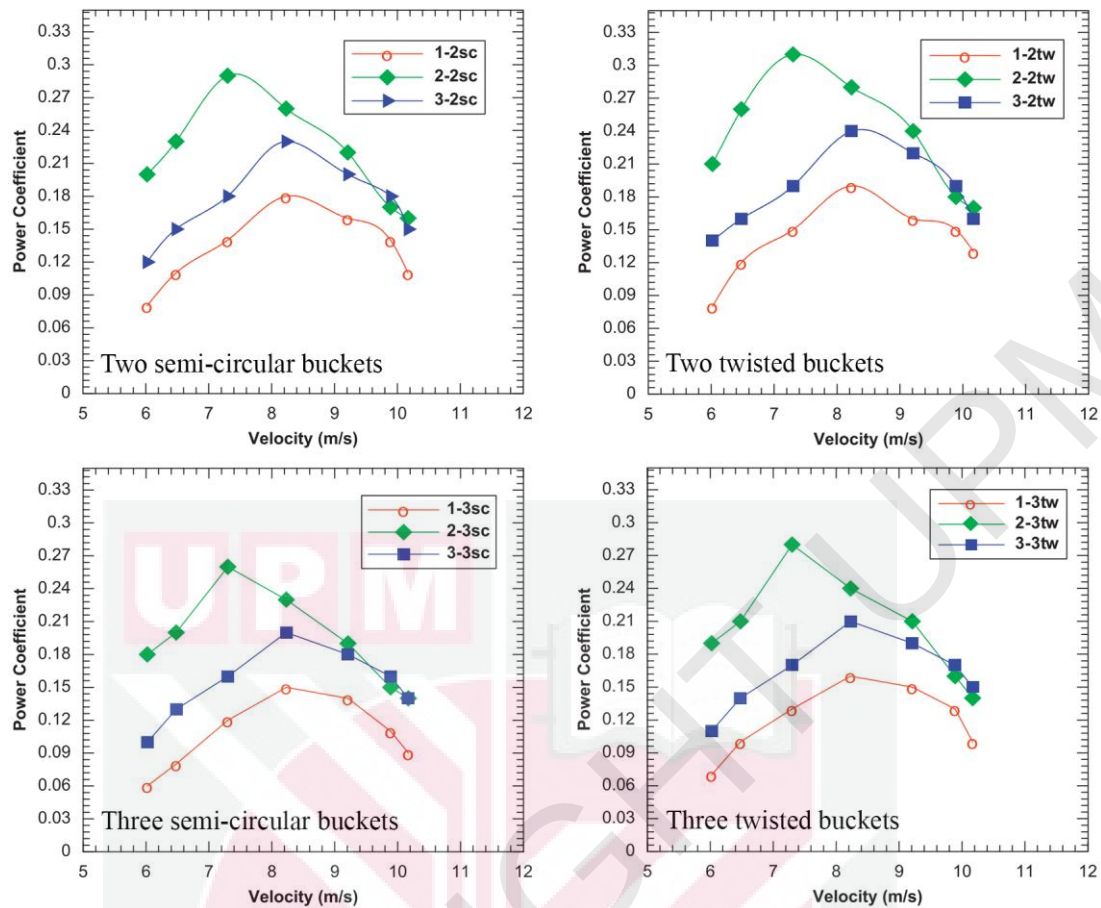


Figure 2.36 : Optimum number of stages [93]

Hayashi et al., 2005 [94] conducted wind tunnel tests using 2-semicircular bucket Savonius turbines with one and three stages. As seen in Figure 2.37(a), both cases were designed with the same overlap ratio of 0.2 and rotor aspect ratio (H/D). A single-stage rotor with a rotor aspect ratio defined as H/d of 1.25 and a three-stage rotor with a stage aspect ratio (h/d) of 0.2 were designed and tested under the steady wind speeds of 6, 9, 12, 15, and 18 m/s. The variations of averaged static torque coefficient (c_{ts}) in one revolution of both cases with different upstream velocities are depicted in Figure 2.37(b). It found that c_{ts} of the three-stage rotor was almost constant. Moreover, the torque and power coefficients presented in Figure 2.37(c) of a single-stage rotor showed better performances than a three-stage rotor. Hayashi et al. mentioned that the inferiority of the three-stage rotor was caused by the small aspect ratio of each stage.

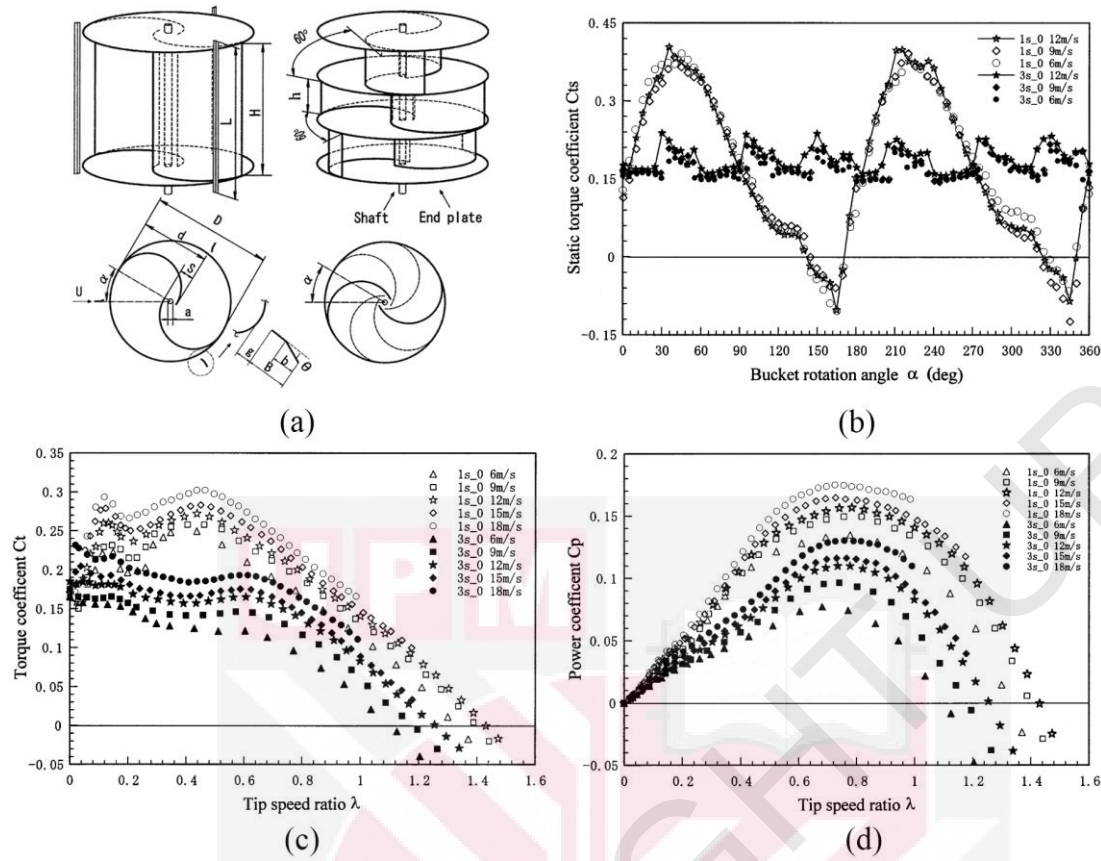


Figure 2.37 : (a) single- and three-stage configurations, (b) static torque coefficient (C_{ts}), (c) torque coefficient (C_t), and (d) power coefficient (C_p) from the study of Hayashi et al., 2005 [94]

For the three-stage rotor, the peak-to-peak value of C_t was lower and almost constant as depicted in Figure 2.38 while it changed significantly at a small tip speed ratio for a one-stage rotor. Figure 2.39 also shows the averaged dynamic torque coefficients (C_t) as functions of bucket rotational angles with four different tip speed ratios (λ) under the same upstream velocity of 12 m/s. It found that the number and amplitude of peaks varied with the increase in tip speed ratio. A three-stage rotor had six peaks in one revolution and the amplitude of C_t was smaller.

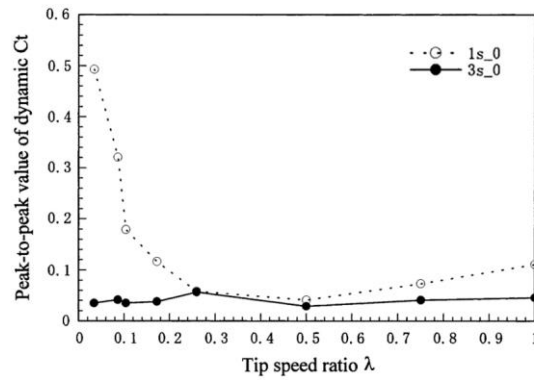


Figure 2.38 : Peak-to-peak values of dynamic torque coefficients at different tip speed ratios between a single-stage rotor and a three-stage rotor [94]

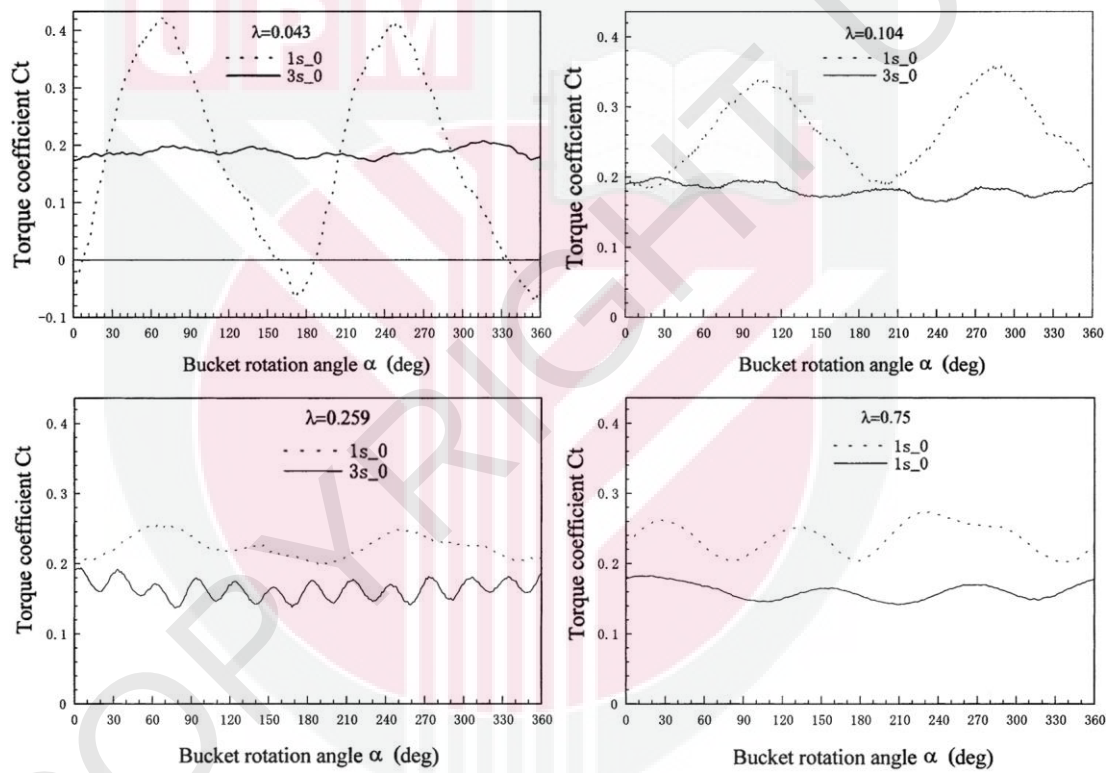


Figure 2.39 : Dynamic torque coefficient versus bucket rotation angles between a single-stage rotor and a three-stage rotor at different tip speed ratios [94]

Kamoji et al.,2008 [95] conducted experimental studies in the open jet wind tunnel under constant the Reynolds number of 80,000 by using the 2-bucket Savonius rotors with single-, two-, and three-stage rotors, and with constant, and different stage aspect ratios (h/D) as shown in Figure 2.40.

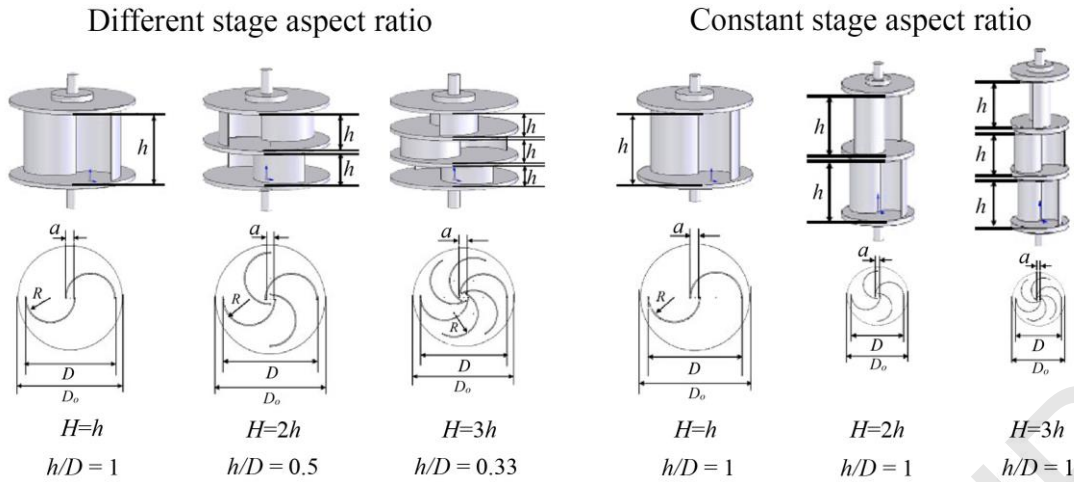


Figure 2.40 : Turbine configurations with different and constant stage aspect ratios [95]

Power and torque coefficients and coefficient of static torque of the turbines with different and constant stage aspect ratios were shown in Figure 2.41(a) and (b), respectively. In the case of different stage aspect ratios, it was found that power and torque coefficients deteriorated when the number of stages was increased. On the other hand, the power and torque coefficients of a single-stage rotor in case of constant stage aspect ratios were higher than ones of two- and three-stage rotors. However, there was no difference in the performances between the two- and three-stage rotors. The maximum power coefficient of a single-stage rotor (15.7%) was higher than a two-stage rotor (12.5%) and a three-stage rotor (12.2%), respectively.

Moreover, the curves of the static torque coefficient of both cases are also shown in Figure 2.41. It could be seen clearly that a three-stage rotor provided a lower peak of fluctuation than the rest configurations. In conclusion, the authors suggested that using a three-stage rotor was better than a two-stage rotor for a constant stage aspect ratio. Based on this study, it could be concluded that turbine fluctuation could be reduced by increasing the number of stages.

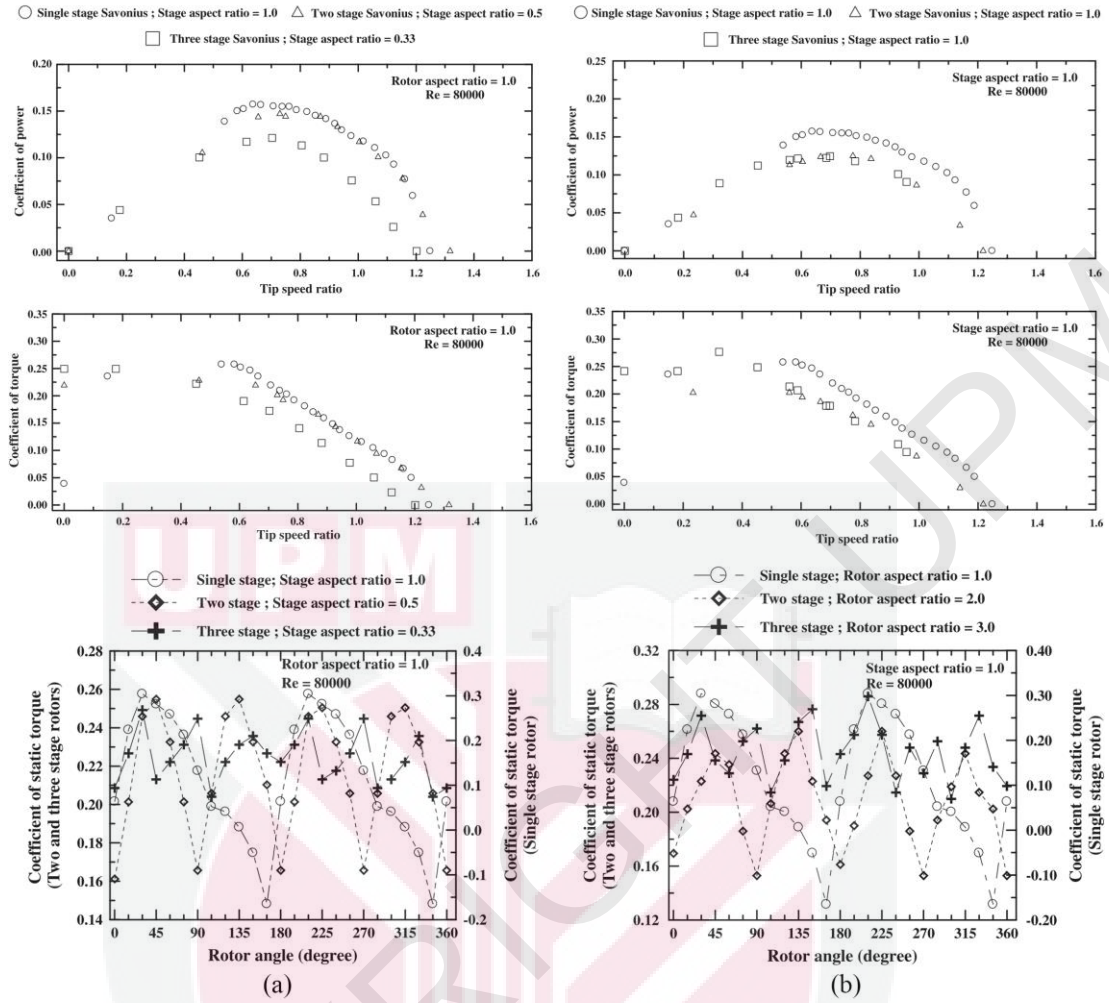


Figure 2.41 : Variation of power, torque, and static torque coefficient for single, two, and three-stage rotors of (a) varied stage aspect ratio and (b) constant stage aspect ratio [95]

2.6.7 Influence of Reynolds Number and Turbulence Intensity

The Reynolds numbers (Re) and turbulence intensity can represent flow conditions. Re is a dimensionless parameter defined as the ratio of inertial force to viscous force to categorize flow characteristics whether the flow is laminar or turbulent. For fluid flowing through the Savonius turbine, the Reynolds number based on the rotor diameter (D_s) can be expressed as in an equation (Eq. 2.26).

$$Re = \frac{\rho U_{\infty} D_s}{\mu} \quad (\text{Eq. 2.26})$$

Many studies with various designs of the Savonius turbines have been conducted experimentally and numerically to investigate the influence of the Reynolds number. The results in Figure 2.42 show that the power coefficient increased when the Reynolds numbers increased until the maximum value. An increase of Re after the maximum value might degrade the turbine performance [36]

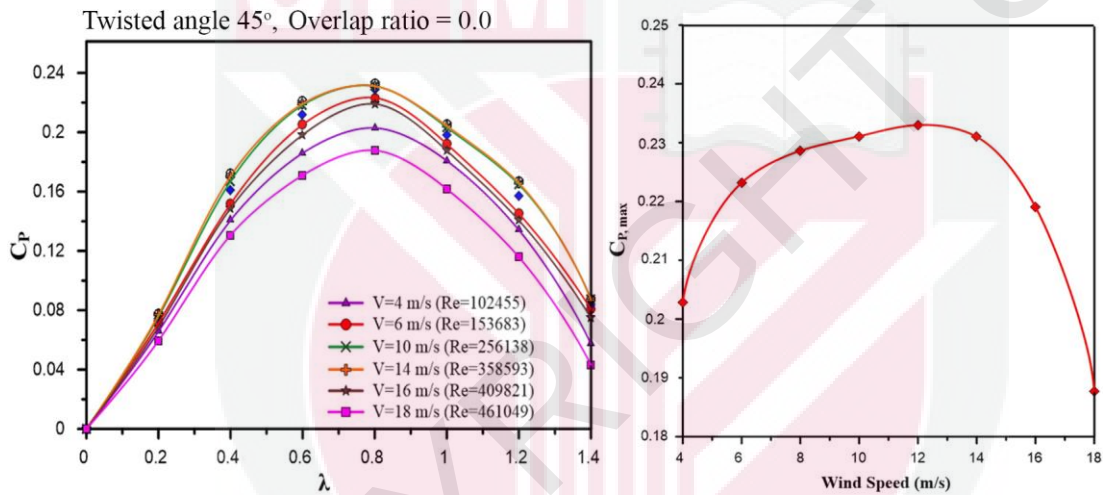


Figure 2.42 : Influence of the Reynolds numbers and wind speed [36]

As presented in Figure 2.43, power and torque coefficients increase corresponding to an increase of the Reynolds number [84]–[86],[94],[95]. Moreover, a lot of studies proved that the static torque coefficient was independent of the Reynolds number [94],[95] as seen in Figure 2.43(b) and (c).

Apart from the Reynolds number, turbulence intensity (I) as expressed in an equation (Eq. 2.27) [17] characterizes the quality of the flow around the turbine and also influences turbine performances.

$$I = \frac{1}{U_{\infty}} \sqrt{\frac{1}{3}(u'^2 + v'^2 + w'^2)} \quad (\text{Eq. 2.27})$$

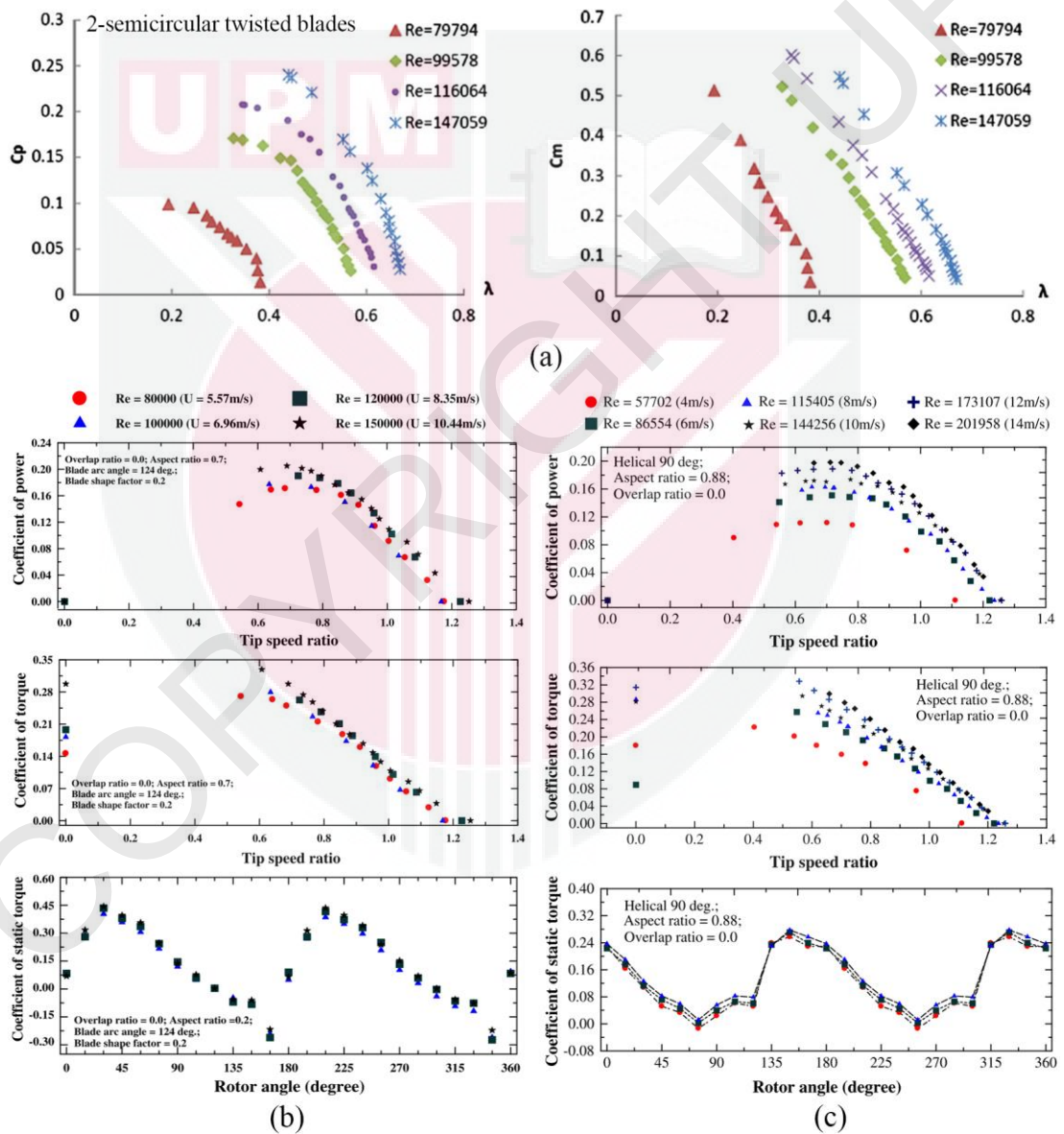


Figure 2.43 : The Influence of the Reynolds number on the turbine performances [85],[86]

As seen in Figure 2.44, the smaller turbulence intensity showed a better power coefficient.

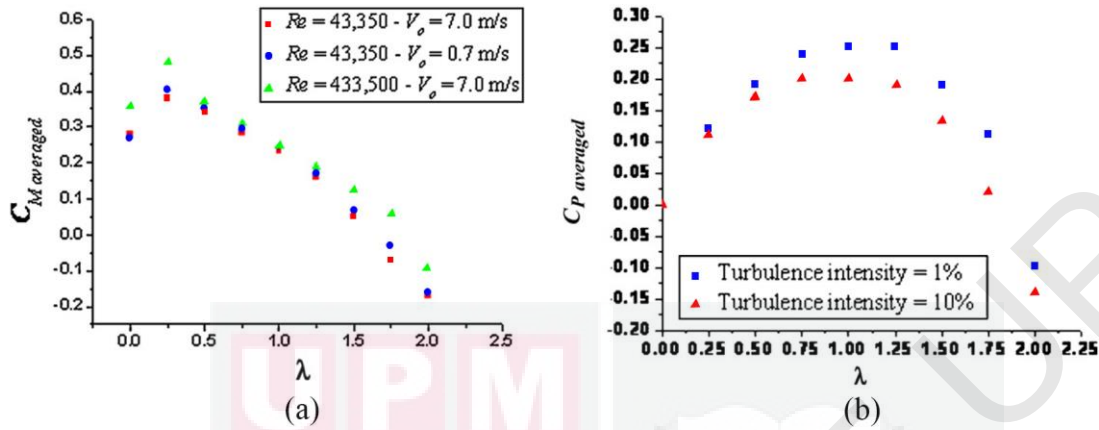


Figure 2.44 : Influence of (a) the Reynolds number and (b) turbulence intensity of medium flow [17]

2.7 Darrieus Hydrokinetic Turbine

The lift-based Darrieus turbine which is categorized as a vertical axis turbine has been designed, developed, and proposed by the French engineer named Darrieus [45] for almost a century [96]. Figure 2.45 shows the timeline of Darrieus turbine development that started with simple designs which were curved-blades in 1968 and straight-blades in the 1970s.

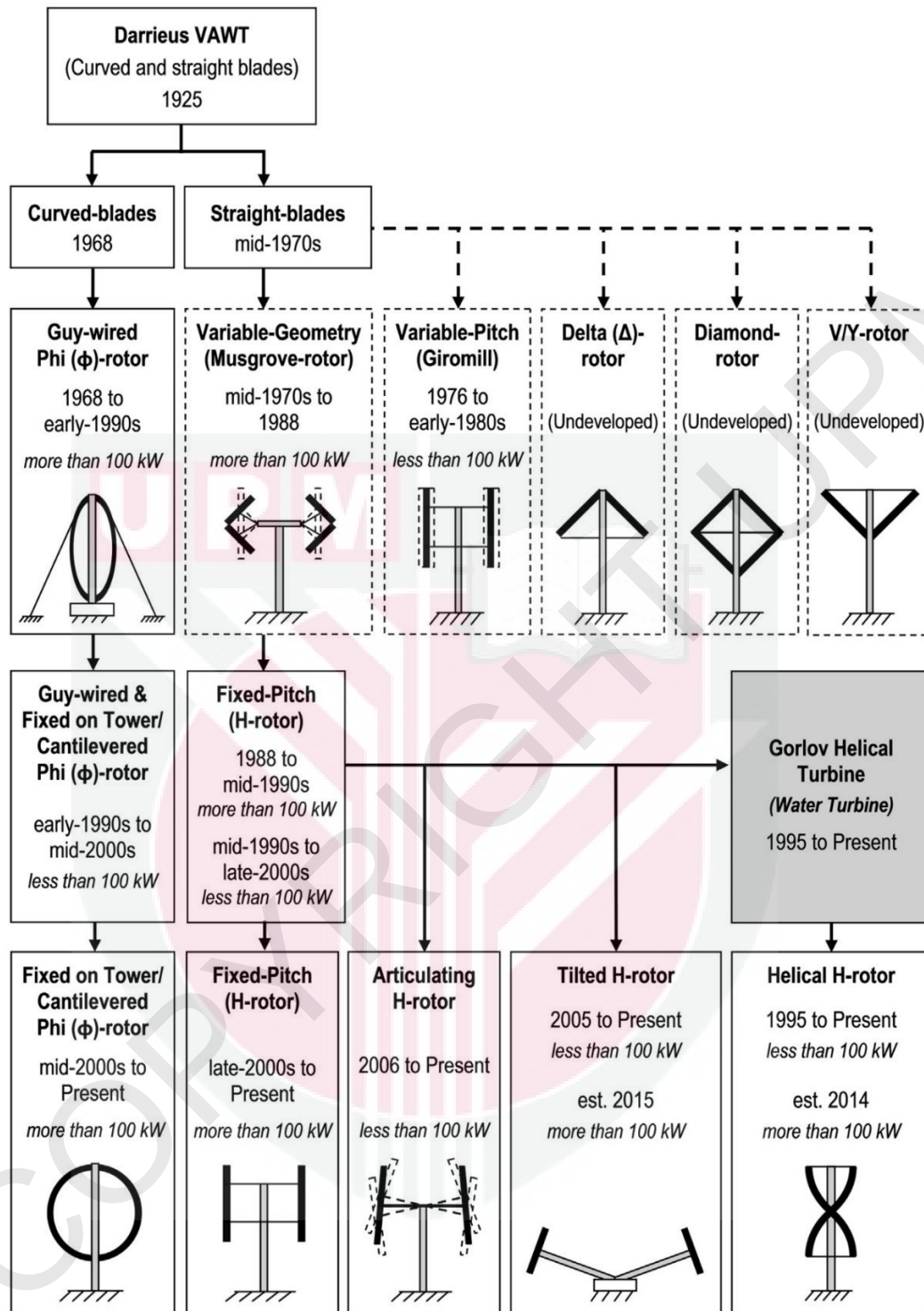


Figure 2.45 : Historical line of Darrieus VAWT development [96]

Several computational and experimental studies on turbine geometric parameters such as rotor diameter [40], chord length [40],[97], blade thickness-to-chord [98], blade profiles [31],[45],[52],[99]–[101], solidity [29],[101], pitch angles [101]–[103], aspect ratios [32],[104],[105], number of blades [22],[101], and blade shapes including the popular H-shape, helical-shape [106], egg-shape or ϕ -shape [75], J-shape turbines [107]–[109]. The studies showed that a general Darrieus turbine provides better efficiency at higher operational ranges (tip speed ratio) than other turbine types, but suffers with the problems of self-starting [51],[53],[110] and also suffers from high cyclic fluctuation [73].

A lot of research activity was also conducted on the turbine blade modifications and passive flow control techniques including the slotted airfoil [111],[112], a slotted deflective flap at the trailing edge [113],[114], the airfoil with different trapped vortex cavity layouts [115],[116], dimple designs and configurations [117],[118], leading-edge airfoil-slat [119], and the Gurney flaps [120],[121] to delay flow separation across the turbine blades and enhance their performance. The results showed that blade modifications could delay stall and enhance turbine power coefficient by around 18-28%.

The vorticity distribution which was produced by straight-, curved-, and helical-bladed turbines was also considered by Scheurich et al., 2010 [122]. It was found that the untwisted blades of the straight- and curved-bladed turbines caused a relatively symmetric distribution of vorticity in the wake downstream of the turbine. Whereas, a strong asymmetry in the distribution between the upper and lower halves was observed in the turbine with helically twisted blades.

Aside from turbine geometric parameters, turbine arrangement, and wake studies [70],[123],[124], installation of fixed guiding walls and upstream deflectors [125]–[127], or even the effect of an auxiliary or a double-bladed Darrieus turbine [128]–[131] utilizing wind or hydro turbines have been investigated since then with the similar purpose of enhancing the turbine performance. The finding results of those parameters will be described intensively in the next section.

2.7.1 Effect of Airfoil Profiles

Airfoil profiles have played an important role in the performance characteristics of the Darrieus turbines. This effect has been considered by using several profile series. Mohamed, 2012 [45] investigated by using two-dimensional CFD approaches on 20 different symmetrical and non-symmetrical airfoil shapes with 5 different profile series including NACA 00XX, NACA 63XXX, S-series, A-series, and FX-series as shown in Figure 2.46.

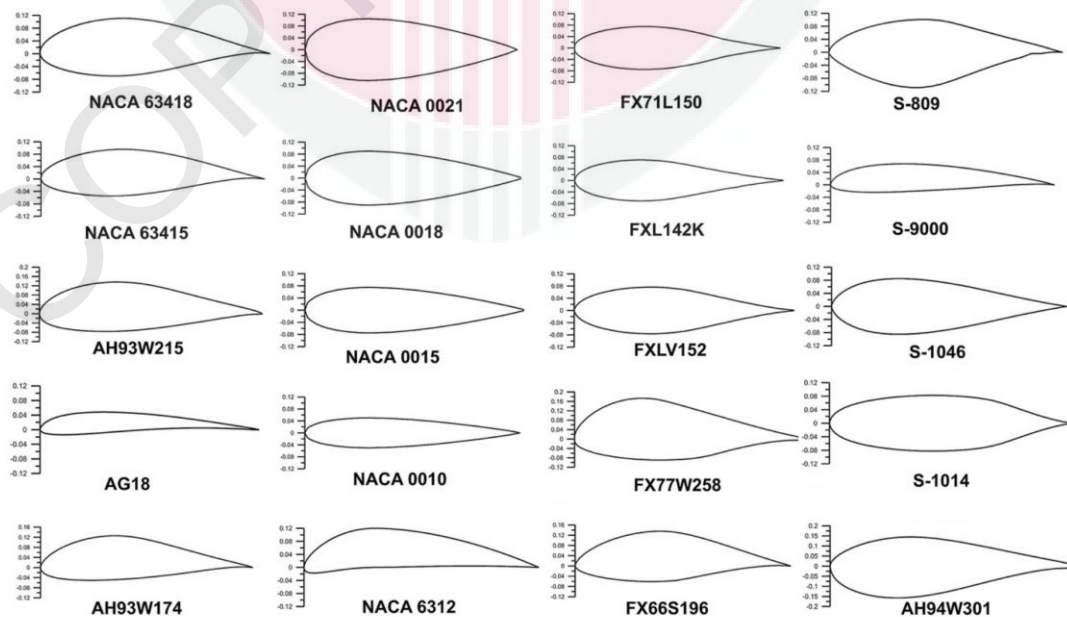
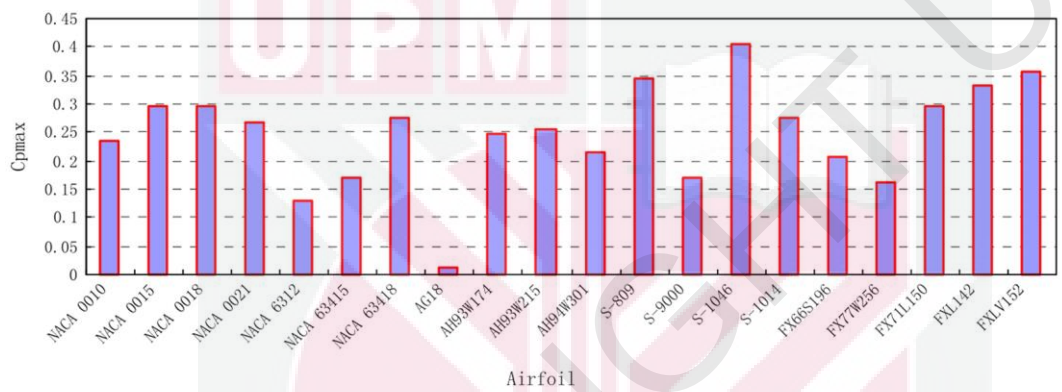
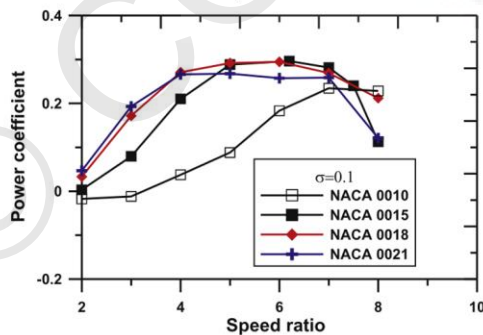
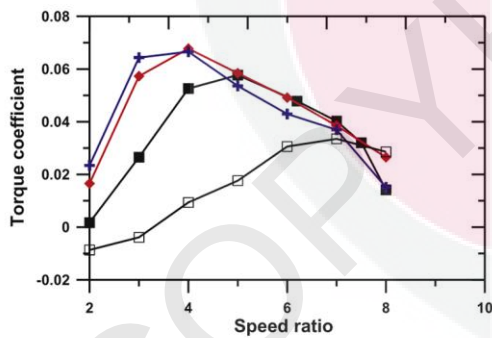


Figure 2.46 : Symmetric and non-symmetric airfoil shapes from the study of Mohamed, 2012 [45]

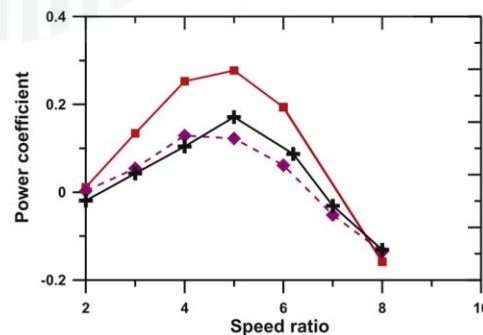
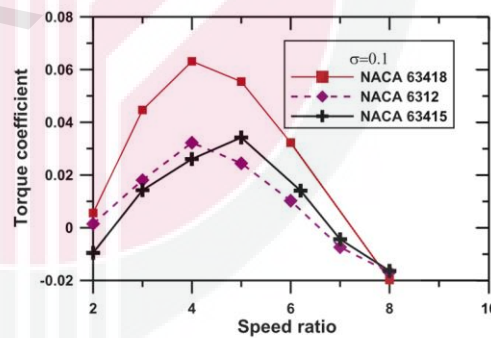
The maximum power coefficient of each airfoil profile from the study of Mohamed, 2012 [45] was presented in Figure 2.47(a) adapted by Jin et al.,2015 [132]. It found that the S-1046 profile showed the highest maximum power coefficient, ($C_{p,max} = 0.4051$). Furthermore, the CFD results as depicted in Figure 2.47(b) and (c) also showed that the performances of turbines with symmetric profiles were higher than the ones with non-symmetric profiles. The maximum power coefficient of the NACA series was 0.2964 corresponding to a NACA 0018 profile.



(a)



(b)



(c)

Figure 2.47 : (a) $C_{p,max}$ of each 20 airfoil profile [132], and torque and power coefficient of (b) symmetric (NACA00XX), and (c) non-symmetric (NACA63xx) airfoils of NACA-series [45]

Furthermore, Subramanian et al., 2017 [100] investigated numerically the effect of airfoil profiles by using two- and three-blade turbines with four different profiles that were NACA 0012, NACA 0015, NACA 0030, and AIR 001. The variation of c_p against λ is illustrated in Figure 2.48. It could be observed that each airfoil had its own operational range. The peak of NACA 0012 was right-shifted to a higher range of λ . Besides, a similar trend could be observed with three-blade turbines but the λ at which maximum c_p was able to be observed, shifted to a lower point.

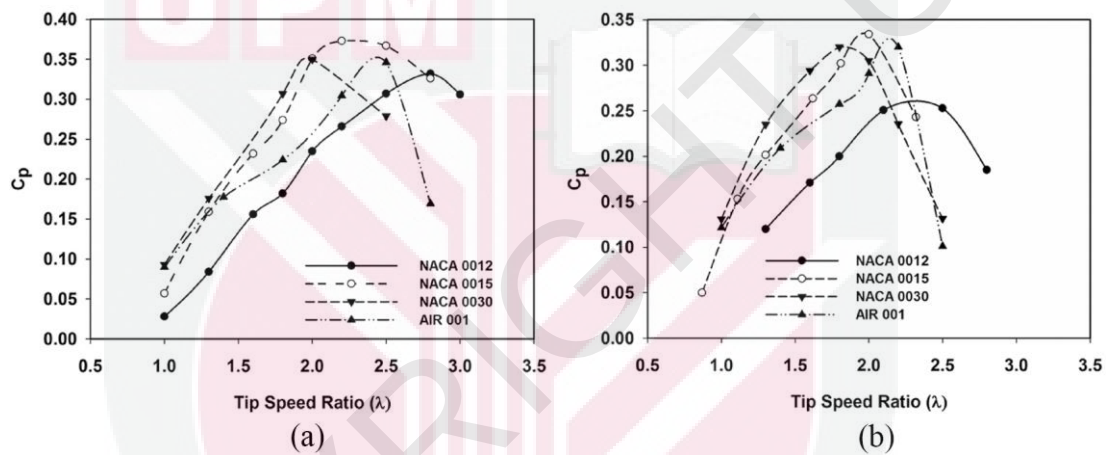


Figure 2.48 : Variation of c_p with λ of (a) 2-blade, and (b) 3-blade Darrieus turbines with four different airfoil profiles [100]

The numerical results as displayed in Figure 2.49(a) and (b) from the study of Mohamed et al., 2015 [52] using 25 airfoil profiles from different series showed that LS(1)-0413, NACA 63-215, NACA 63-415, NACA 64-415, and NACA 0015 were the top five of power coefficient among those 25 profiles. Besides, LS(1)-0413 was awarded to be the best power coefficient because of the highest peak of efficiency and wider operational range, while NACA 63-415 was awarded to be the best operating range. Blade profiles of LS(1)-0413 and NACA 63-415 are depicted in Figure 2.49(c).

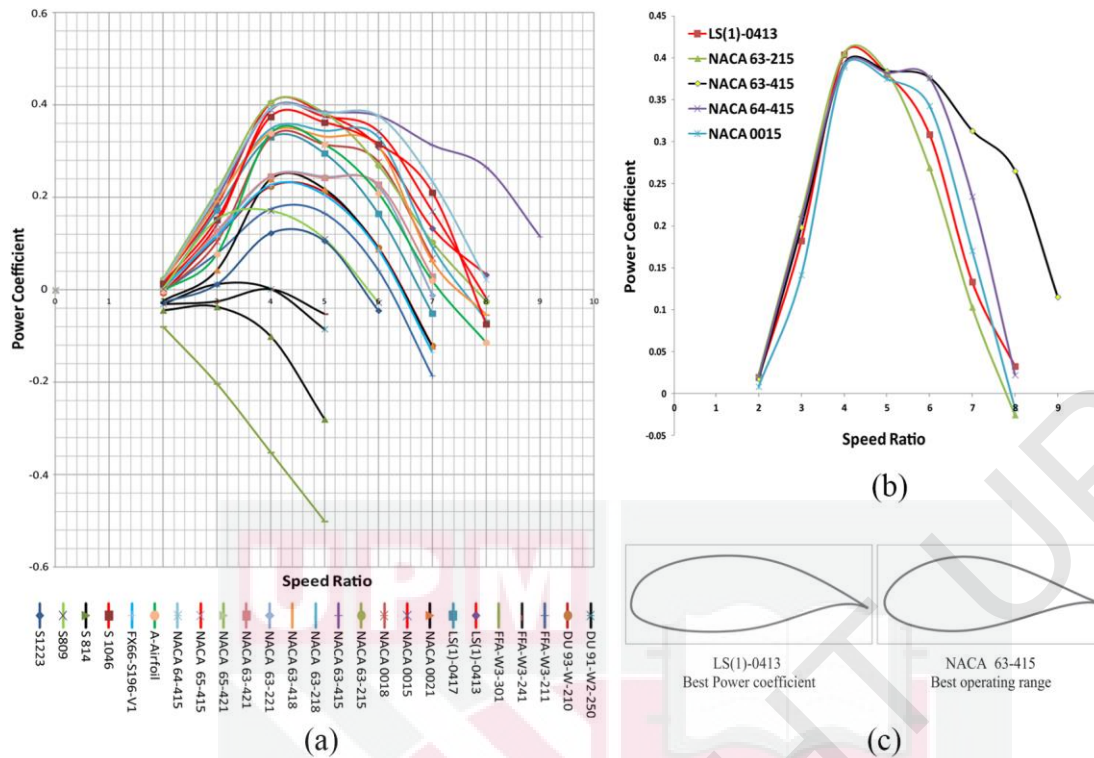


Figure 2.49 : (a) Variation of power coefficient with tip speed ratios of selected 25 airfoil profiles, (b) top five of power coefficient, and (c) shapes of LS(1)-0413 and NACA63-415 profiles [52]

2.7.2 Effect of Bladed Shapes

Bladed shapes of Darrieus turbines have also been investigated by Lee and Lim 2015 [103]. Four different values of twist or helical angles including 0° (A0), 10° (A10), 20° (A20), and 30° (A30) as illustrated in Figure 2.50 [103] were tested. The numerical results depicted in Figure 2.51 using three-dimensional simulations of 3-blade NACA0018 Darrieus wind turbines ($H_D = 1,600$, $D_D = 1,800$, and $c = 200$ mm.) showed that the helical angle had no significant influence on power performance.

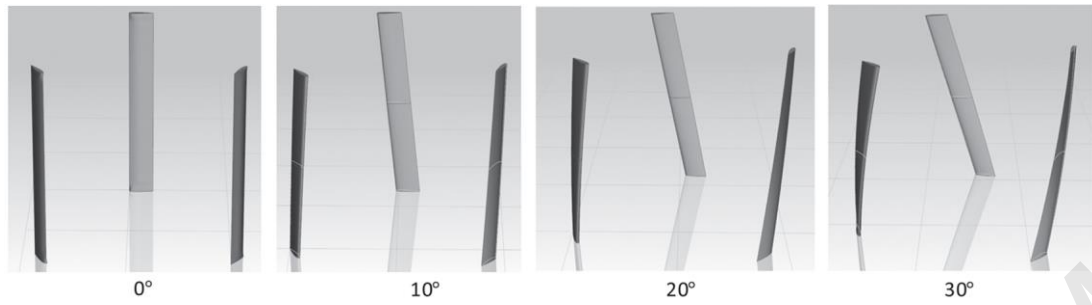


Figure 2.50 : Four different values of helical angles [103]

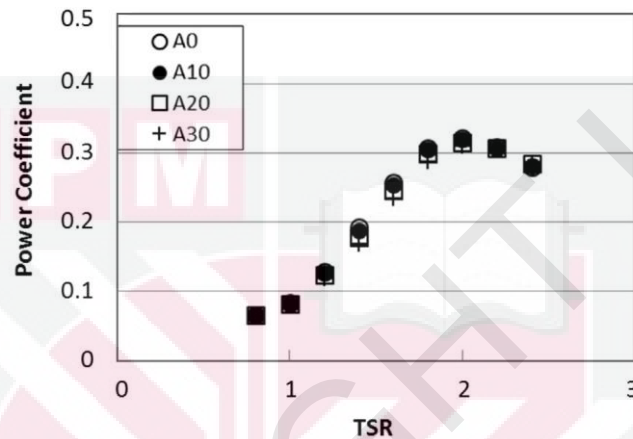


Figure 2.51 : Variations of power coefficient with TSR from different helical angles [103]

2.7.3 Effect of Blade Number

An important parameter of typical Darrieus turbines is the number of blades. Two- and three-straight blades of Darrieus turbine configurations are illustrated in Figure 2.52(a) and (b), respectively. The effect of blade number has been investigated using both experimental and CFD numerical approaches by many researchers. For example, two-dimensional simulations were conducted by Liang et al. [43] and Sabaeifard et al., 2012 [133], and 3D simulation by Subramanian et al., 2017 [100] has been carried out with different numbers of blades and airfoil profiles of wind turbines.

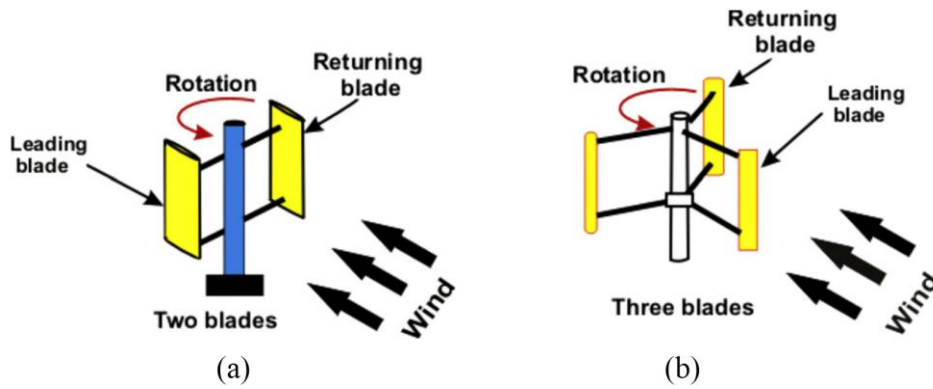


Figure 2.52 : Two and three-bladed H-rotor Darrieus turbines [45]

Based on studies of Sabaeifard et al., 2012 [133], the 3-, H-blade Darrieus wind turbine showed the highest c_p and widest operational range as depicted in Figure 2.53(a) reviewed by Ghasemian et al., 2017 [22] compared to the rest designs. However, the results of Liang et al. [43] and Subramanian et al. [100] in Figure 2.53(b) and (c), respectively showed that the highest c_p was of a 2-blade turbine. In addition, at the lower range of λ , the c_p of a 2-blade turbine was lower than the rest.

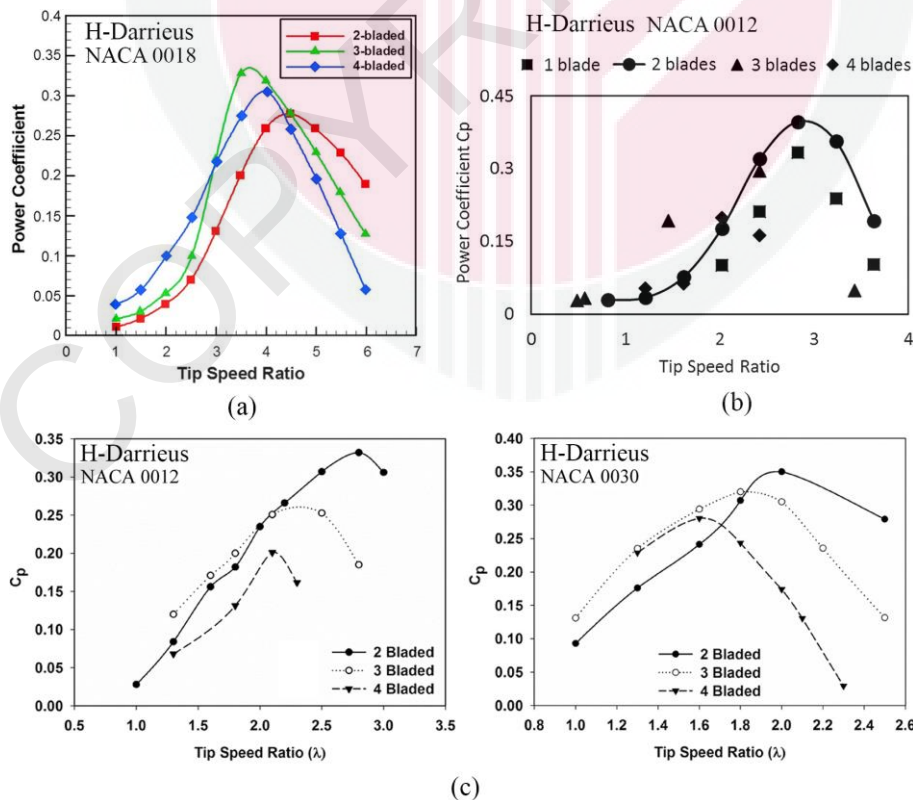


Figure 2.53 : Computational results on the effect of the number of blades from the previous studies [22],[43],[100]

Furthermore, the effect of the number of blades has also been investigated together with other parameters such as blade designs, aspect ratios, or pitch angles. Figure 2.54 [134] depicts the experimental results of straight-blade Darrieus wind turbines with 2, 3, 4, and 5 blades combined with various optimum pitch angles (6° , 8° , 8° , and 12°). The results studied by Li et al. 2015 [134] showed that the higher numbers of blades augment the power coefficient in the range of low tip speed ratios, however in the higher tip speed ratios, the lower numbers of blades performed better.

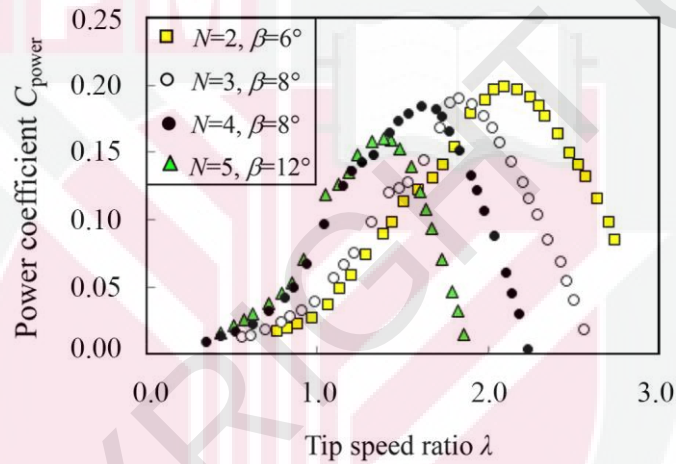


Figure 2.54 : Power coefficient for different numbers of blades according to the optimum pitch angle [134]

The finding results could be additionally explained by the vorticity contour as revealed in Figure 2.55 [100] obtained from the study of Subramanian et al. 2017 [100]. From Figure 2.55, the vorticity contours for the 3-bladed NACA 0012 profile and NACA 0030 at low λ are illustrated. It can be observed that flow fields across the turbines showed that a 3-blade turbine faced higher interactions with the vortex leading to the detaching of mechanical power. Therefore, the c_p of a 3-blade turbine was lower than a 2-blade turbine [100]. Besides, the increasing of blade numbers to 4 blades deteriorated the performance of the Darrieus turbine.

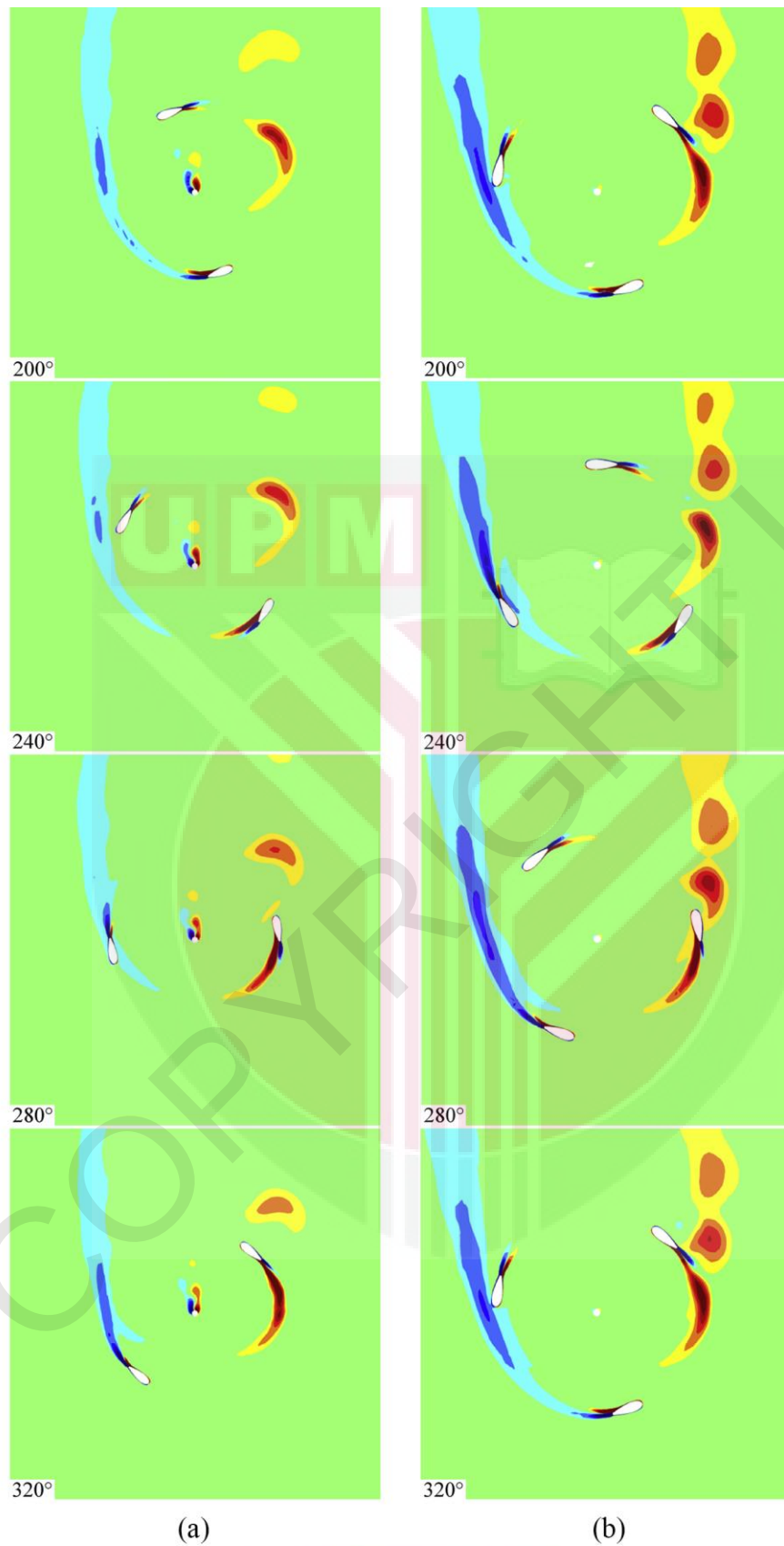


Figure 2.55 : Mid-span vorticity contours of (a) 2-bladed, and (b) 3-bladed NACA 0030 profiles [100]

2.7.4 Effects of Solidity

Solidity (σ) is defined based on the number of blades (n), chord length (c), and rotor radius of rotor (R_r) as expressed in equation (Eq. 2.9). Experimental and computational approaches have been conducted to investigate the effect of this geometric parameter. The numerical results reviewed by Ghasemian et al., 2017 [22] of various solidity values as illustrated in Figure 2.56(a) and (b) were collected by Sabaeifard et al., 2012 [133] and Lee and Lim, 2015 [103], respectively, and the results from the study of Mohamed, 2012 [45] were illustrated in Figure 2.56(c).

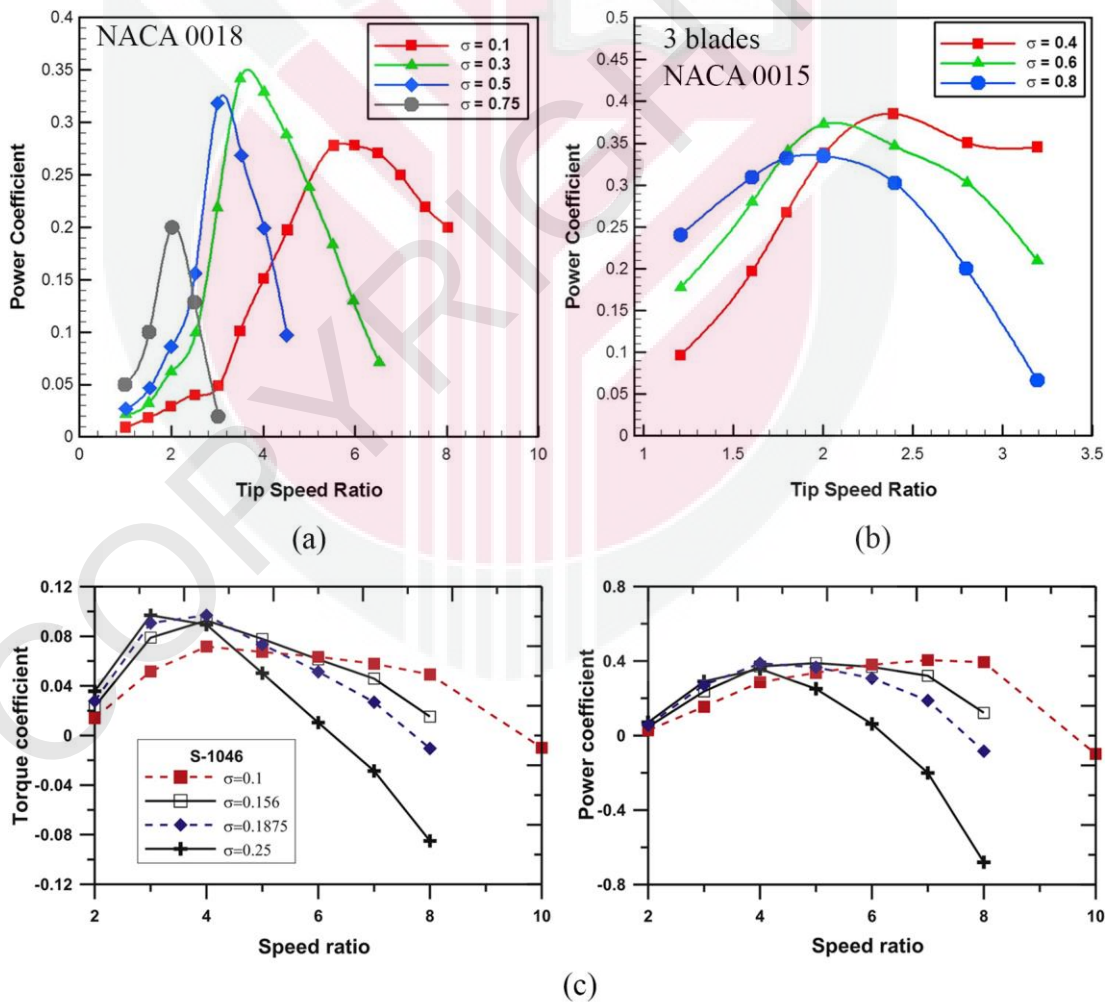


Figure 2.56 : Power coefficient versus tip speed ratio with various solidity values of (a) NACA0018, (b) NACA0015 [22], and (c) S-1046 airfoil profiles [45]

The instantaneous torque and power coefficients in one revolution of four different solidity values at a fixed speed ratio ($\lambda = 6$) were also depicted in Figure 2.57. Based on the literature, it was found that torque and power coefficients decreased when the solidity values increased. Therefore, the c_p curve was shifted left when solidity values were increased.

The results in Figure 2.56 were also consistent with the study of Eboibi et al., 2016 [135], Joo et al., 2015 [136], and Gang and Kau, 2015 [137]. It confirmed that a lower solidity showed a wider operating range. According to the studies mentioned above, it can be concluded that a turbine with low solidity (shorter chord length and bigger rotor diameter) showed better performances for a high range of tip speed ratios. However, during a lower range of tip speed ratio, a higher solidity turbine performed better [60]. Besides, decreasing the solidity, the maximum power coefficient decreased accordingly. The power curves became more flattened and the power capture remained constant for wider values of tip speed ratios [138].

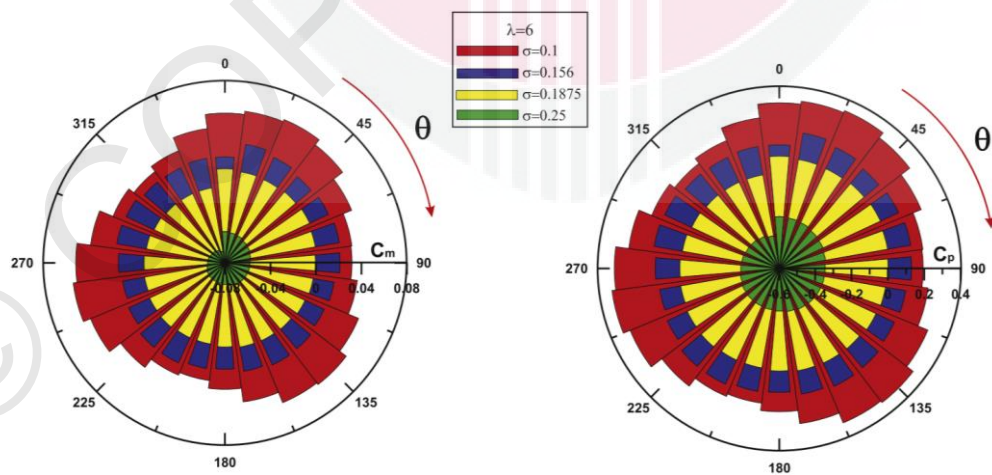


Figure 2.57 : Instantaneous torque (c_m) and power (c_p) coefficients of S-1046 H- Darrieus turbines with four different solidities [45]

2.7.5 Effects of Aspect Ratio

Generally, an aspect ratio (H_D/D_D) of a typical Darrieus turbine is defined as the ratio of turbine height (H_D) and diameter (D_D) as similar as mentioned in the aspect ratio of the Savonius turbine. Numerical [32] and experimental [139],[104] investigations have been carried out on this effect so far. Figure 2.58(a), and (b) present the experimental results of the unsymmetrical S1210 and symmetrical NACA0018 three-blade Darrieus wind turbine, respectively.

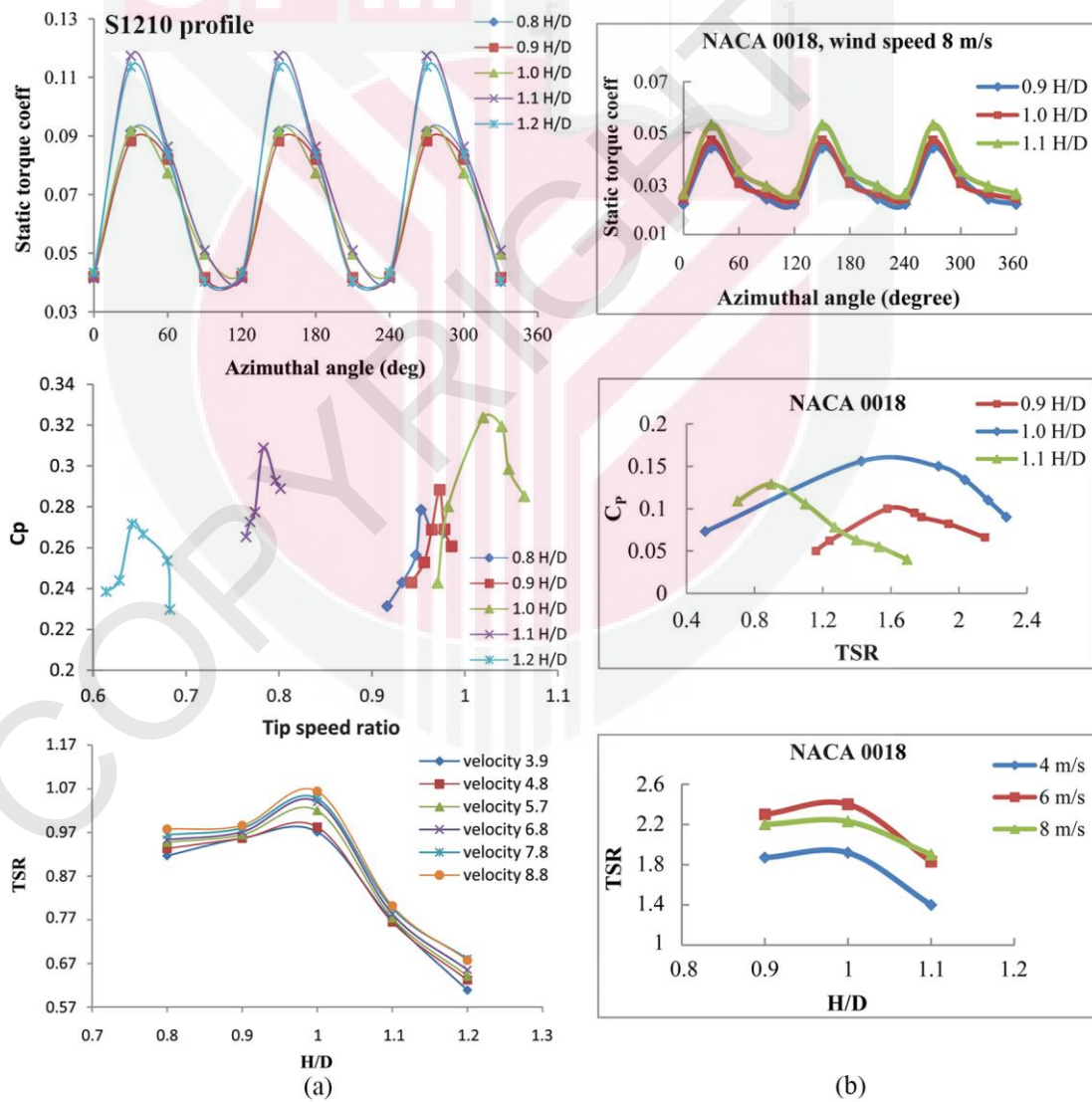


Figure 2.58 : Experimental results on the effect of aspect ratios of Darrieus wind turbines [139],[104]

As seen in Figure 2.58, static torque coefficient versus azimuth angles, the power coefficient (c_p) with respect to tip speed ratios, and tip speed ratios versus aspect ratios were displayed. It could be observed that the starting torque coefficient increased as the aspect ratio increased. It meant that the self-starting ability of Darries turbines during start-up might be enhanced with the increasing of aspect ratio.

From c_p curves versus tip speed ratios (TSR) of turbines with different aspect ratios, it found that c_p values of every aspect ratio increased as TSR value increased until the maximum value, after that c_p decreased with further increase of TSR . In addition, the optimum aspect ratio value ($H_D/D_D = 1$) could be found in both S1210 and NACA0018 profiles.

2.7.6 Effect of Pitch Angle

A pitch angle (β) of the Darrieus turbine is defined as an angle between the tangential line and chord line of a turbine blade. A pitch angle is a conventional sign to describe blade characteristics. There are two systems to define either positive or negative pitch angles. For the first system as depicted in Figure 2.59(a), the positive pitch angle is defined as an inward rotation of the leading edge (toe-in) [61] and the negative β is for the leading edge outwards [61],[102],[103],[140]–[144]. On the other hand, as shown in Figure 2.59(b), the pitch angle convention is signed in the opposite direction from the first one [52],[145],[146]. Variations of pitch angles including positive values (inward), negative values (outward), and zero were investigated by experimental and numerical methods as presented in Table 2.10.

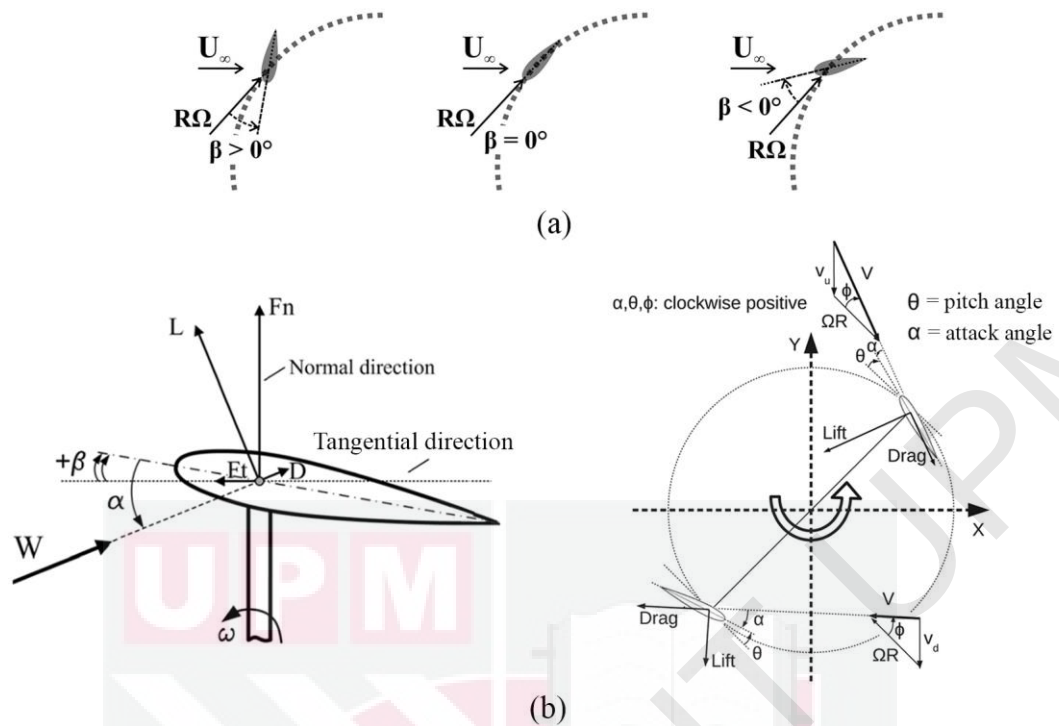


Figure 2.59 : (a) Variation of pitch angle [61], and (b) Schematic of pitch angle [145],[146]

Table 2.10 : Summary of published literature on the effect of pitch angles

Researchers	Methodology	Turbine	Range used	Optimum β
Chen and Zhou, 2009 [141]	2D simulation Wind turbine	3 blades NACA0018	-5° to 5°	-2° outward
Armstrong et al., 2012 [102]	Experiment Wind tunnel	3 straight blades NACA0015	-9°, -6°, -3°, 0°, and 3°	-6° outward
		3 canted blades NACA0013	-5.5°, -3.5°, -1.5°, and 2.5°	-3.5° outward
Sumantraa et al., 2014 [142]	2D simulation Wind turbine	3 blades NACA0015	-6°, 0°, and +6°	-6° outward
Mohamed et al., 2015 [52]	2D simulation Wind turbine	3 blades LS(1)-0413	-10° to 10°	0°
Lee and Lim, 2015 [103]	3D simulation Wind turbine	3 blades NACA0015	-6°, -3°, -2°, -1°, 0°, and 3°	-2° outward
Rezaeiha et al., 2017 [61]	2D simulation Wind turbine	3 blades NACA0015	-7° to 3°	-2° outward
Abdalrahman et al., 2017 [144]	2D simulation Wind turbine	3 blades NACA0018	-6°, -4°, 0°, 4°, and 6°	0°

From the investigations, the results could be observed that the optimum performances occurred when the turbines had a negative pitch angle [61],[102],[103],[141],[142] as illustrated in Figure 2.60. The other conclusion as depicted in Figure 2.61 mentioned that the optimum performances occurred when pitch angle was zero [52],[144].

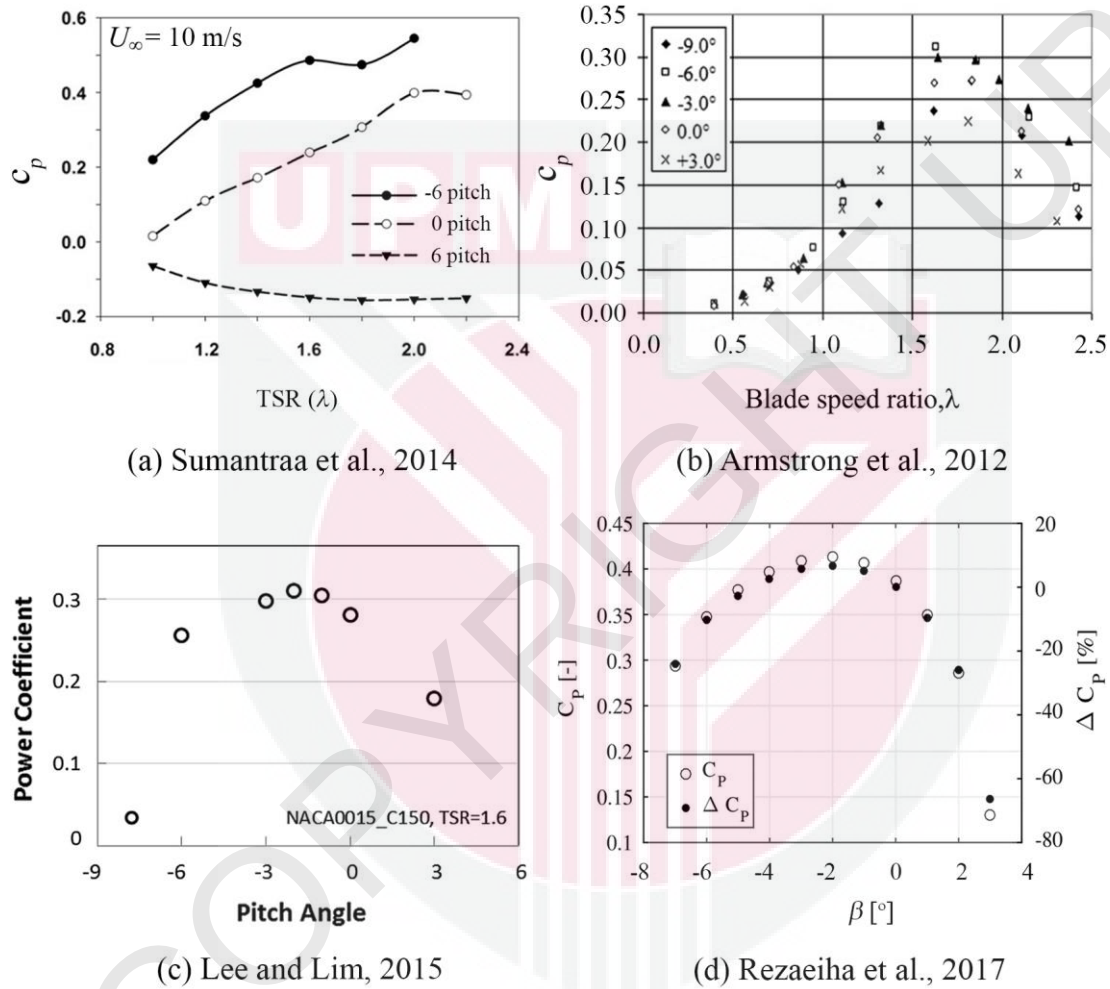
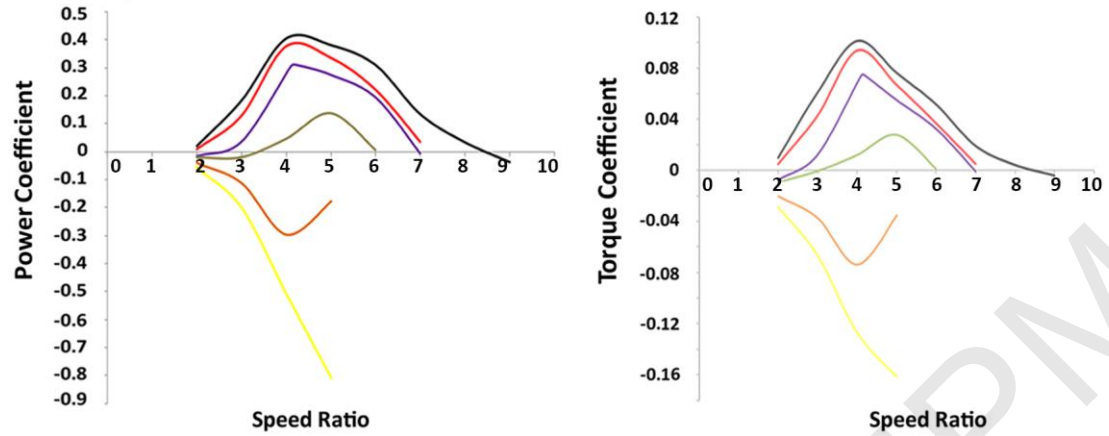
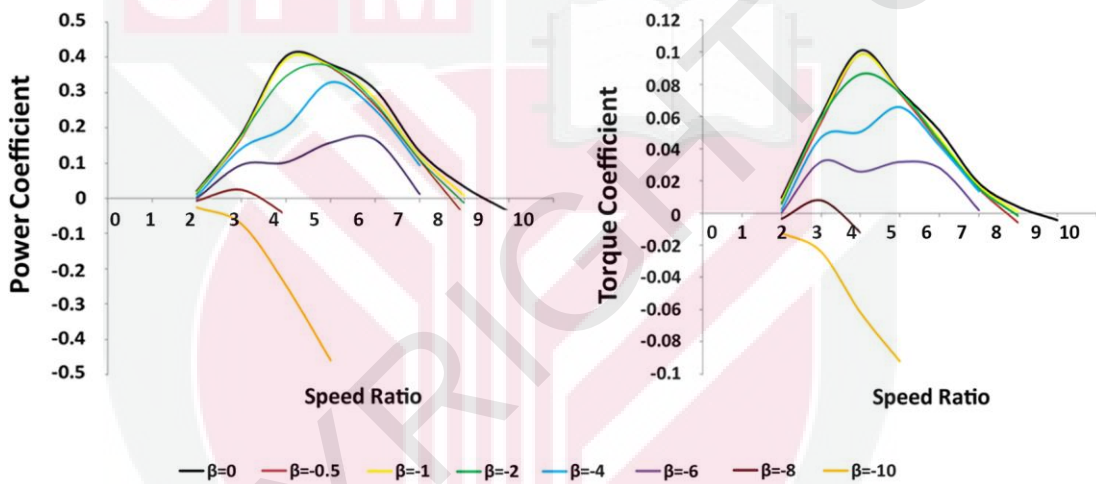


Figure 2.60 : The optimum power coefficient occurred when pitch angles were negative (outward) [61],[103],[102],[142]

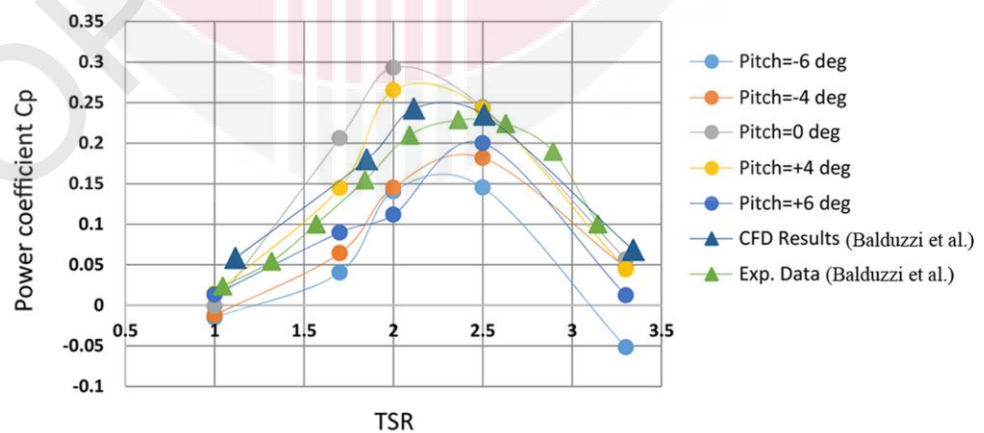
Positive β



Negative β



(a) Mohamed et al., 2015



(b) Abdalrahman et al., 2017

Figure 2.61 : The optimum power coefficient occurred when the pitch angle was zero ($\beta = 0$) [52],[144]

Moreover, Li et al., 2015 [134] conducted wind tunnel experiment on the effect of pitch angles as well as numbers of blades by using NACA0021 straight-blade Darrieus turbine with 2, 3, 4, and 5 blades and several pitch angles. As depicted in Figure 2.62(a), the results showed that the optimum power coefficient of the turbines with different numbers of blades was obtained at different pitch angles. The optimum configuration of each condition was $\beta = 6^\circ$ when $n = 2$, $\beta = 8^\circ$ when $n = 3$, $\beta = 8^\circ$ when $n = 4$, and $\beta = 12^\circ$ when $n = 5$. As seen in Figure 2.62(b), the power coefficient curves of the optimum configuration of each condition were plotted once again. Therefore, the 2-blade turbine with $\beta = 6^\circ$ showed the best performance among the rest configurations.

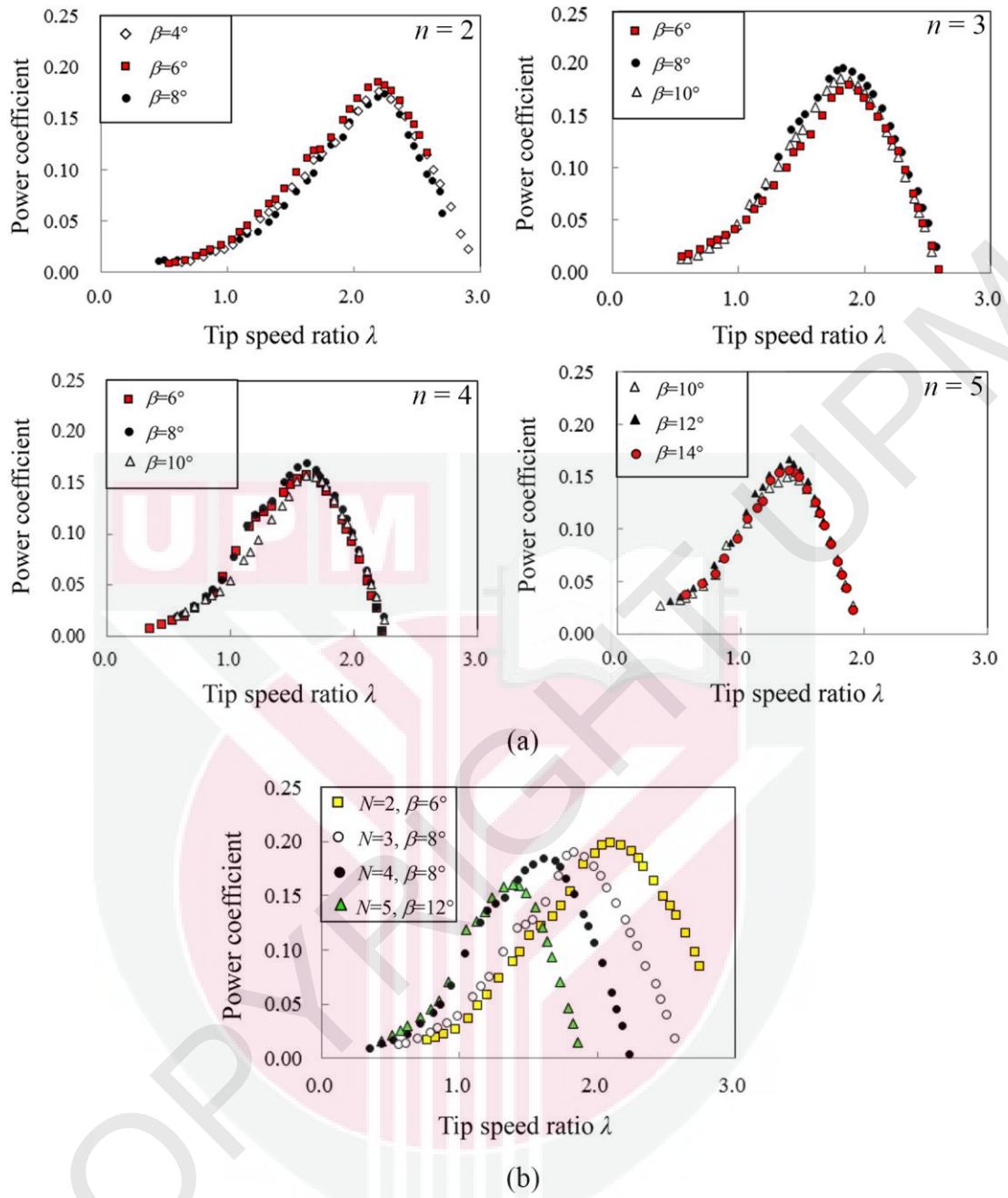


Figure 2.62 : Power coefficient curves from different pitch angles [134]

2.8 Combination of Hydrokinetic Turbine

As mentioned in the previous sections, each conventional vertical axis hydrokinetic turbine has its own specific ranges of operation [2],[8] as illustrated in Figure 2.63 and specific operational characteristics. The conventional Savonius turbines for example are good at providing an excellent torque or self-starting capability, and perform better during a low range of tip speed ratio, while the Darrieus hydrokinetic turbines give good dynamic efficiencies and perform better during a high range of tip speed ratio. Therefore, the combination of the Savonius and Darrieus turbines called the combined Savonius-Darrieus hydrokinetic turbine or hybrid turbine may complement each other and get better overall performances.

The combination aims to combine the self-starting torque ability of a Savonius turbine with the high dynamic efficiency of a Darrieus turbine. To enhance the performances of typical combined Savonius-Darrieus turbines, some geometric, and operational parameters such as the effects of compact designs, radius ratios, attachment angles, and rotational velocities need to be regarded. The determination of the parameters will be described in the next sections.

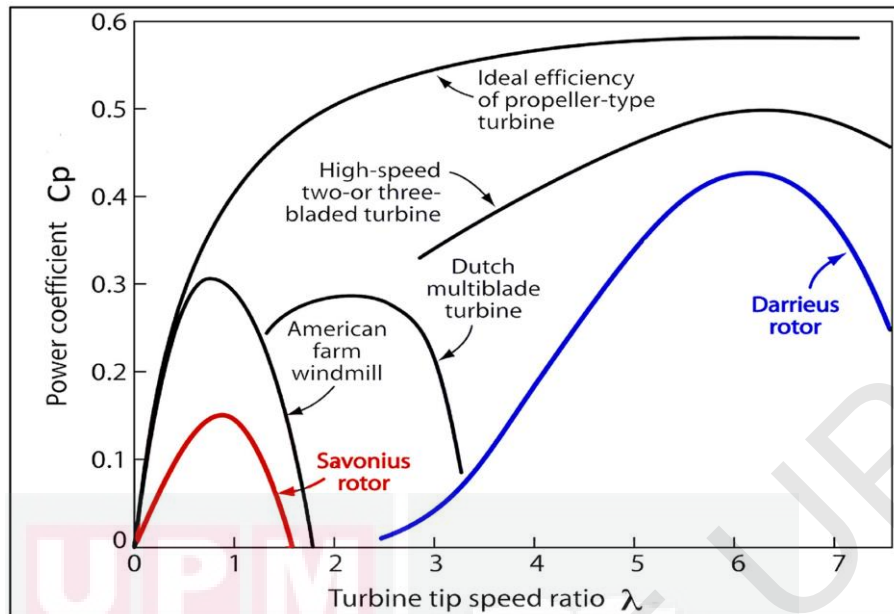


Figure 2.63 : Power coefficient of different wind turbines as functions of the tip speed ratio [21]

2.8.1 Effect of the Compact Design

Compact designs known as combined configurations have been studied with two design configurations as shown in Figure 2.64. The first design configuration depicted in Figure 2.64(a) was called the nested coupling. A Savonius rotor was mounted at the center and inside the Darrieus rotor [23],[46],[147]–[150]. The second one depicted in Figure 2.64(b) was called the stage coupling. The Savonius and Darrieus rotors were connected serially [47],[87],[150]–[152].

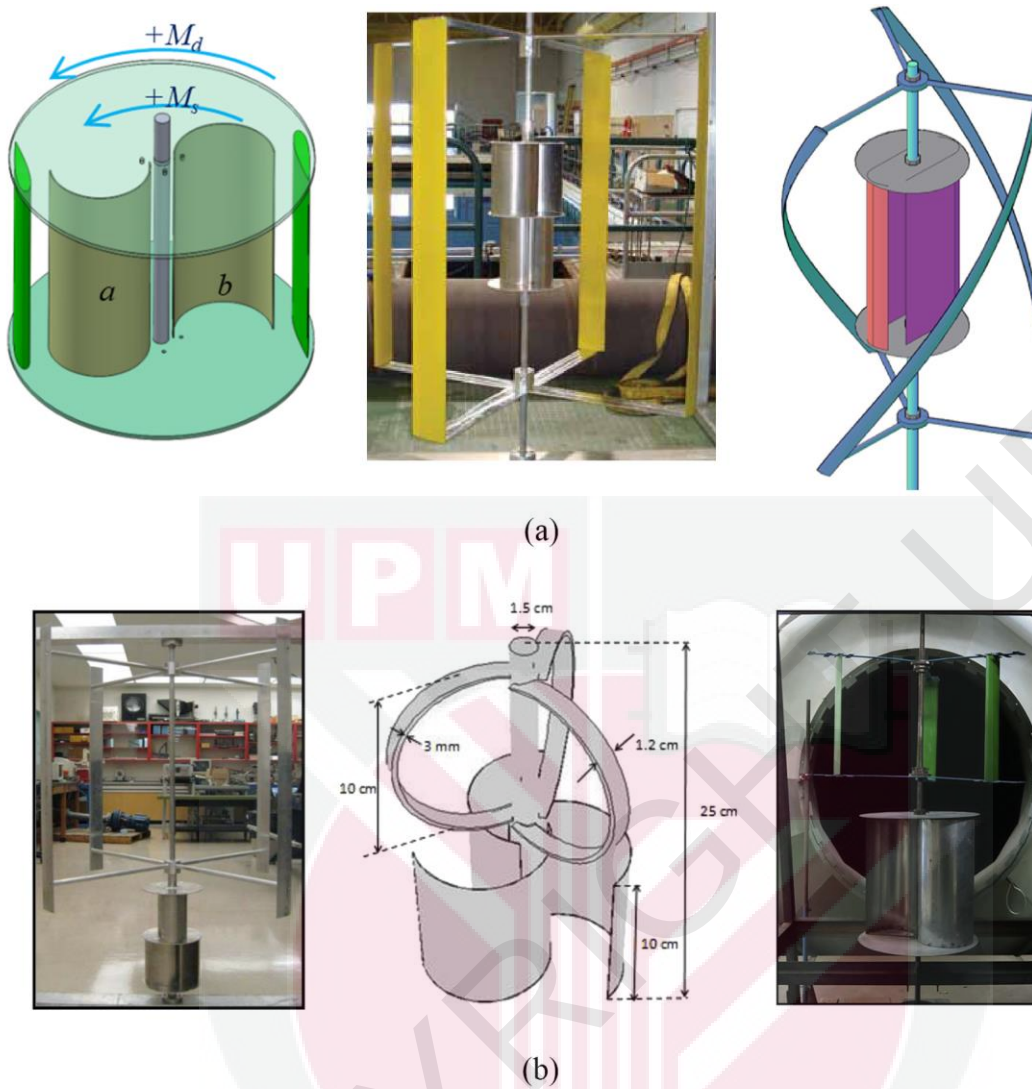


Figure 2.64 : Turbine configurations of the combined systems; (a) nested coupling [21],[23],[148], and (b) stage coupling [39],[47],[87]

Wakui et al., 2005 [150] conducted numerical studies on the effect of compact designs with 2 different configurations as illustrated in Figure 2.65. A 2-bladed, Troposkien bladed shape Darrieus turbine and a 2-stage, 2-semicircular bucket Savonius wind turbine were assembled with a nested coupling design for Type-A and with a stage coupling design for Type-B. The results in Figure 2.65 presented that the Type-B design provided a rated output about 1.5 times higher than the Type-A design.

Moreover, flow interference could be observed from a Type-A configuration due to Savonius and Darrieus turbines. Even though Type-B configuration produced lower wake quantities, however, this configuration had to lengthen the turbine height which might not be appropriate for the shallow river. Based on many consideration factors, pros, and cons, the authors concluded that a Type-A configuration was more appropriate in terms of compact size and operating behaviour than a Type-B design.

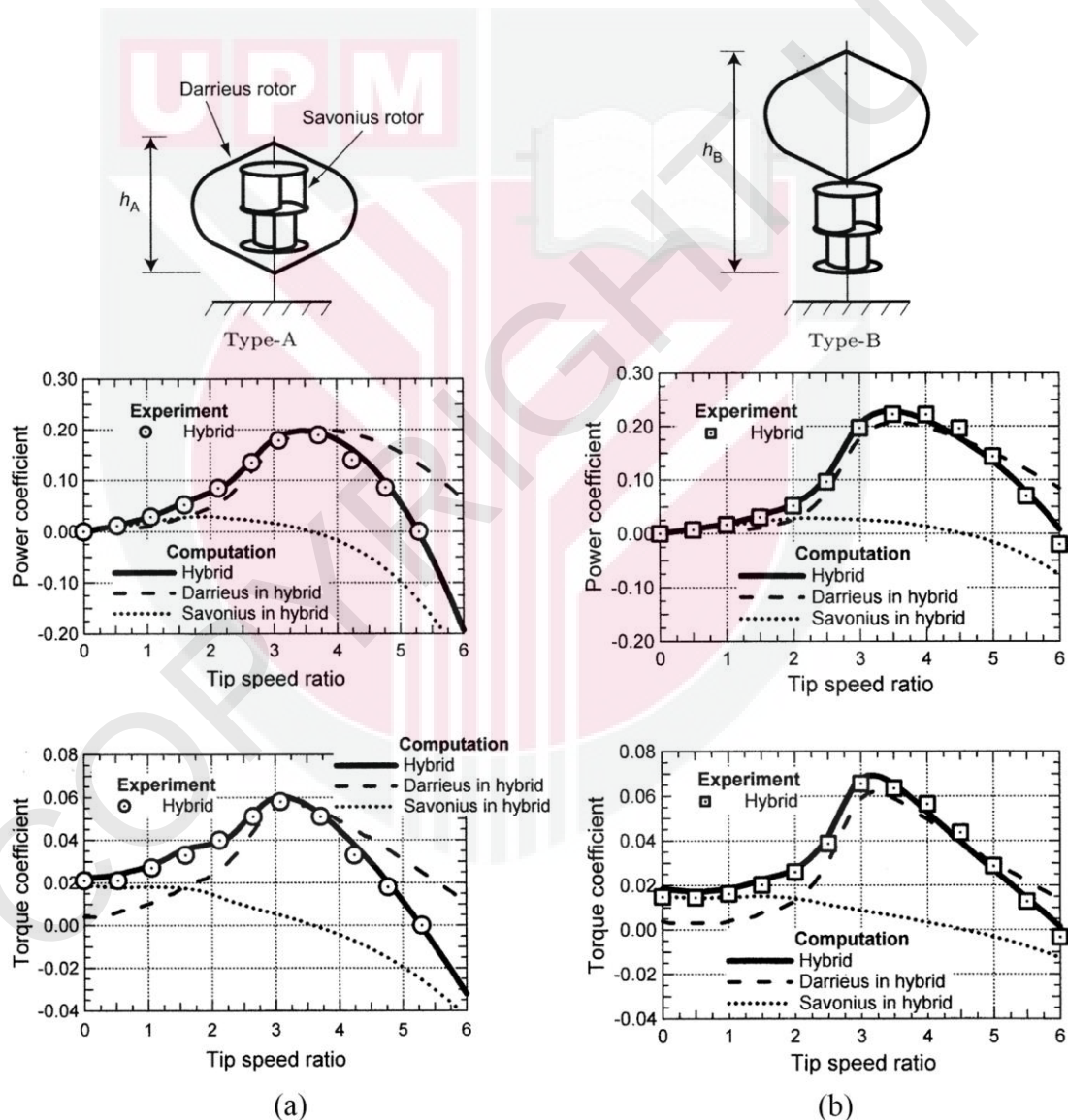


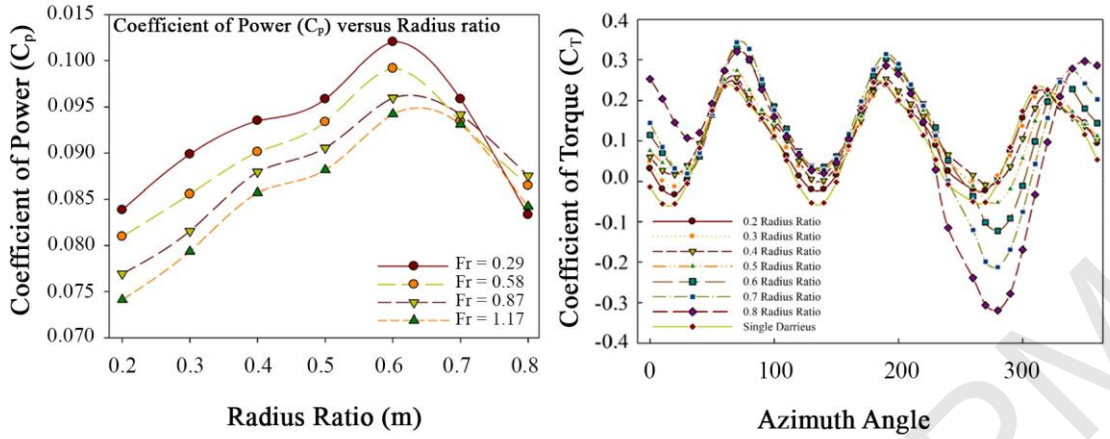
Figure 2.65 : Effect of compact designs [150]

2.8.2 Effects of the Radius Ratio

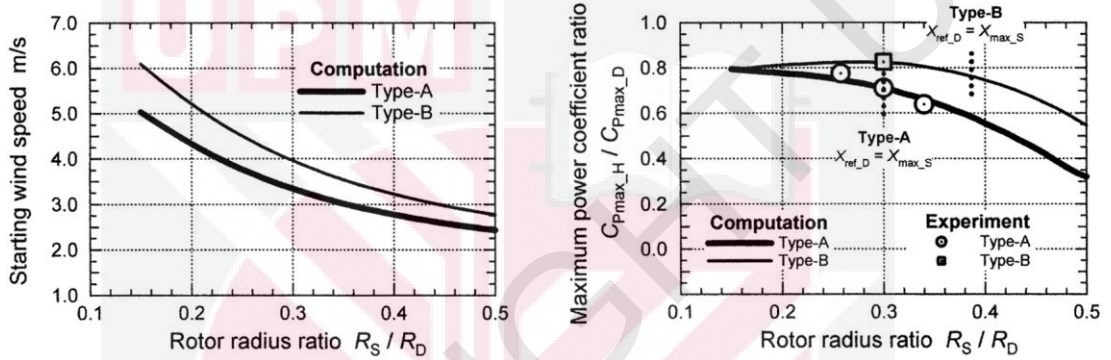
Radius ratio (R_S/R_D) is a parameter used to specify the geometries of a combined turbine and influences importantly turbine performances. It is a ratio of the radius (R) or swept area (A_S) of the Savonius rotor to the Darrieus rotor [43]. Furthermore, it also represents the ratio of drag force to lift force due to the Savonius and Darrieus rotor, respectively, and influences significantly the generated torque and thus power generation [42]. It refers that, the participation of a Savonius part in a combined system is increased with an increase of a radius ratio.

The optimum R_S/R_D of combined turbines has been determined by numerical methods [42],[43],[46],[150] and experiments [153],[154]. Turbine performances including c_p and c_t were plotted with respect to values of R_S/R_D as illustrated in Figure 2.66 and Figure 2.67.

As seen in Figure 2.66(a), the c_p was increased with an increase of R_S/R_D until reaching the maximum value. Beyond the maximum point, c_p decreased with an increase of R_S/R_D . The optimum R_S/R_D was 0.6 for every Froude number (Fr). Instantaneous c_t was also plotted with respect to the azimuthal angle. It found that R_S/R_D of 0.6 provided higher instantaneous c_t . Information on starting wind speed was provided in Figure 2.66(b). When R_S/R_D was increased, the starting wind speed dropped down for both compact designs. Similarly, $c_{p,max}$ ratio was decreased corresponding with an increase of R_S/R_D .



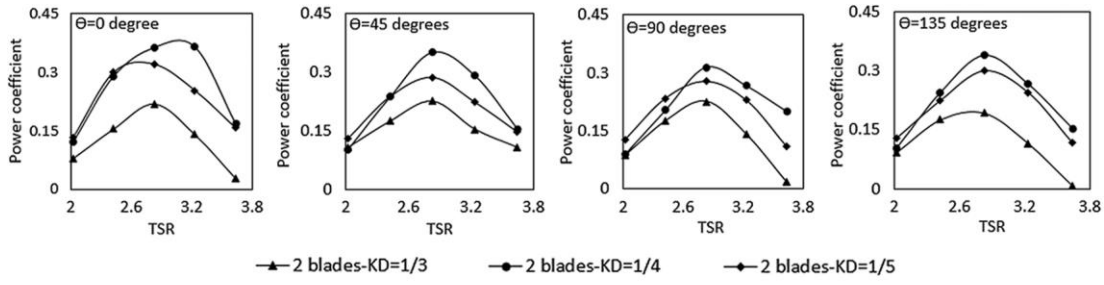
(a) Saini and Saini, 2020



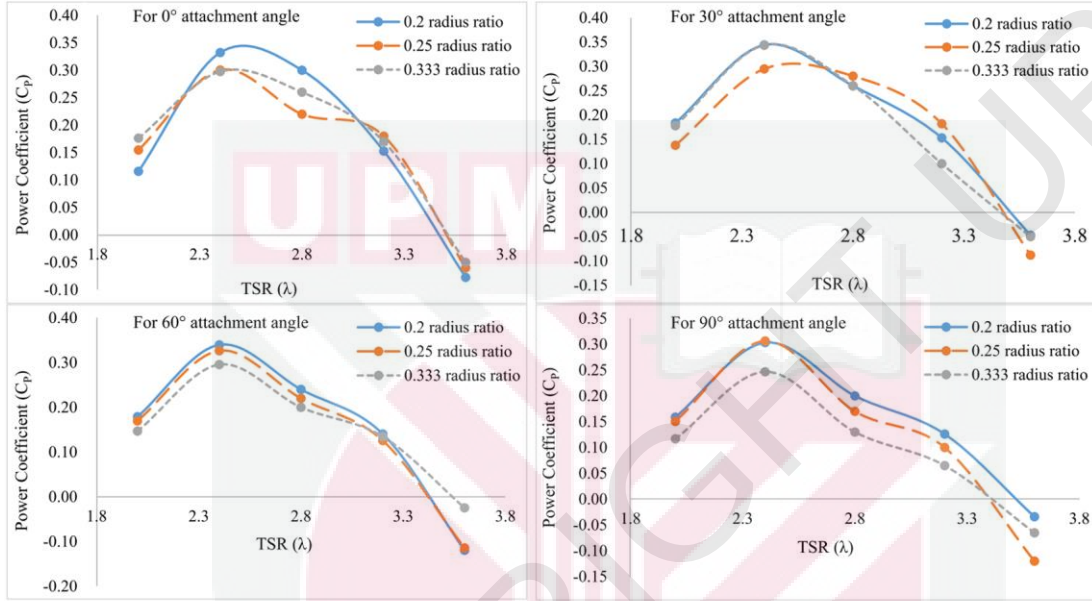
(b) Wakui et al., 2005

Figure 2.66 : Turbine performances versus radius ratios from the studies of Saini and Saini, 2020 and Wakui et al., 2005 [46],[150]

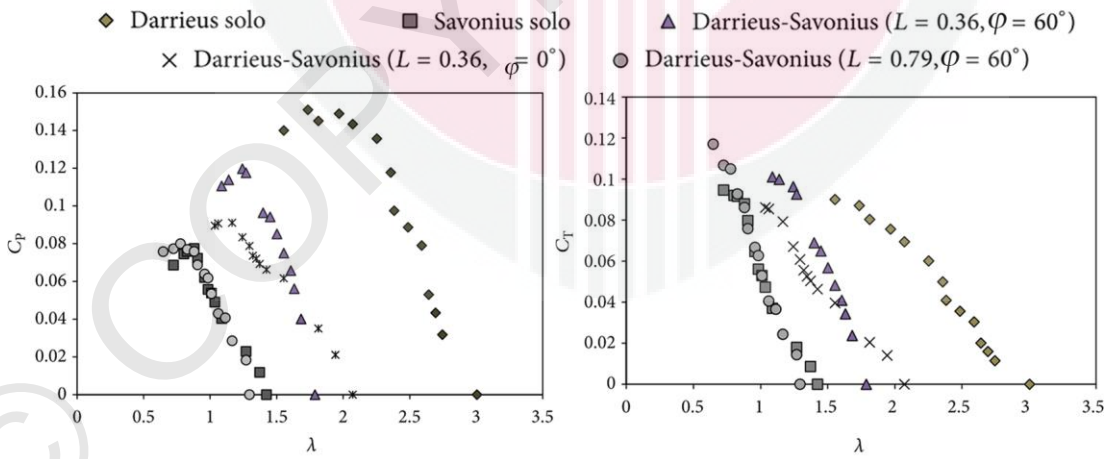
Three different radius ratios were numerically studied by Liang et al., 2017 [43] and Saini and saini, 2018 [42] for the combined turbines with various attachment angles. The findings showed that the ratio of 0.25 provided the highest c_p from the study of Liang et al., 2017 [43] and 0.2 from the results of Saini and saini, 2018 [42] as illustrated in Figure 2.67(a) and (b), respectively. Moreover, an experimental investigation on the combined turbines with 3 different ratios was also carried out by Sahim et al. 2013 [154]. The results as illustrated in Figure 2.67(c) were plotted and compared with the results of Savonius, and Darrieus stand-alone. The finding presented that c_p curves had the same tendency in every radius ratio. The optimum or maximum c_p with respect to λ of each radius ratio could be observed.



(a) Liang et al., 2017



(b) Saini and saini, 2018



(c) Sahim et al. 2013

Figure 2.67: Effects of radius ratios on the c_p and c_t of the Darrieus turbines from previous studies [42],[43],[154]

2.8.3 Effect of the Attachment Angle

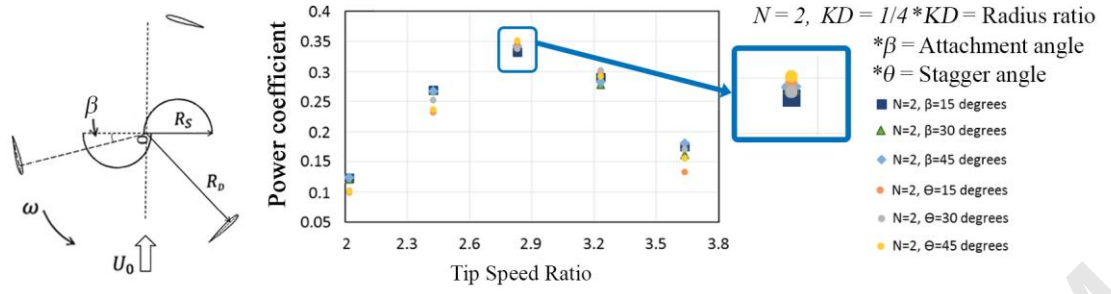
In the combined system, an attachment angle (φ) used to describe turbine positioning or relative configurations between the Savonius and Darrieus turbines corresponding to each other is defined as an angle between the radius arms of a Darrieus blade with respect to a Savonius bucket. Numerical and experimental investigations were conducted on this effect with various designs and configurations of combined turbines. Details of the turbines used and some findings are presented in Table 2.11 and Figure 2.68.

From the numerical studies of Liang et al., 2017 [43], an optimum φ was found to be zero, whereas $\varphi = 60^\circ$ was an optimum value from the numerical studies of Saini and Saini, 2018 [42], Saini and Saini, 2020 [46], and the experimental study of Sahim et al., 2013 [154]. Further, the experimental studies of Kyoizuka, 2008 [149] found that the combined turbine provided the best performances when an attachment angle equaled 135° . Based on numerical and experimental studies mentioned earlier, it can be concluded that the attachment angle between both Savonius and Darrieus radius arms in a combined system should not be co-axial (0°) or orthogonal (90°) to each other in order to attain better performances.

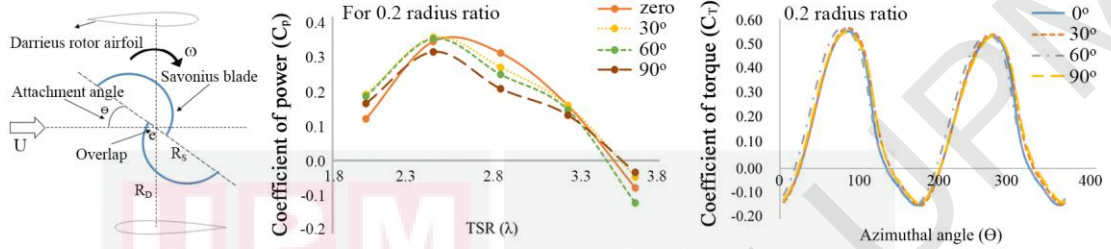
Table 2.11 : Summary of published literature on the effect of attachment angles

Researchers	Methodology	Turbine size	Range used	Optimum ϕ	Optimum c_p and c_t
Kyozuka, 2008 [149]	Experimental water turbine	<u>D</u> ; 2 blades, NACA0018, $c = 100$ mm., $D_D = 1500$ mm., $H_D = 600$ mm. <u>S</u> ; 2 Semicircle buckets, $H_S = 475$ mm.	0°, 45°, 90°, 135°	135°	$c_p > 0.3$ at $\lambda = 2.2$
Sahim et al., 2013 [154]	Experimental water turbine	<u>D</u> ; 3 blades, NACA0015, $c = 63$ mm., $H_D = 300$ mm. <u>S</u> ; 3 semielliptic buckets, $H_S = 300$ mm.	0°, 60°	60°	$c_p = 0.12$ at $R_S/R_d = 0.36$ $c_t = 0.12$ at $R_S/R_D = 0.79$
Liang et al., 2017 [43]	2D Numerical, Wind turbine	<u>D</u> ; 2 blades, NACA0012, $c = 220$ mm, $R_D = 750$ mm. <u>S</u> ; 2 Semicircle buckets	0°, 45°, 90°, 135°	0°	$c_p = 0.363$ at $n = 2$, $R_S/R_D = 1/4$
Saini and Saini, 2018 [42]	2D Numerical, water and wind turbine	<u>D</u> ; 2 blades, NACA0012, $c = 220$ mm., $D_D = 1500$ mm. <u>S</u> ; 2 Semicircle buckets	0°, 30°, 60°, 90°	30°, 60°* *(from the reference x-axis)	$c_p = 0.34$ at $R_S/R_D = 0.2$ & 0.33 $\lambda = 2.4$,
Saini and Saini, 2020 [46]	3D Numerical, water turbine	<u>D</u> ; 3 blades, S-1046, $c = 55$ mm., $D_D = 175$ mm., $H_D = 160$ mm., <u>S</u> ; 2 Semicircle buckets $H_S = 160$ mm.	60°, 90°, 120° from x-axis	90° *(from the reference x-axis) or 60° (between 2 turbines)	$c_p = 0.109$, $c_{t,max} = 0.336$, $c_{t,avg} = 0.1273$ at $\lambda = 0.703$, $R_S/R_D = 0.6$, $Fr = 0.29$

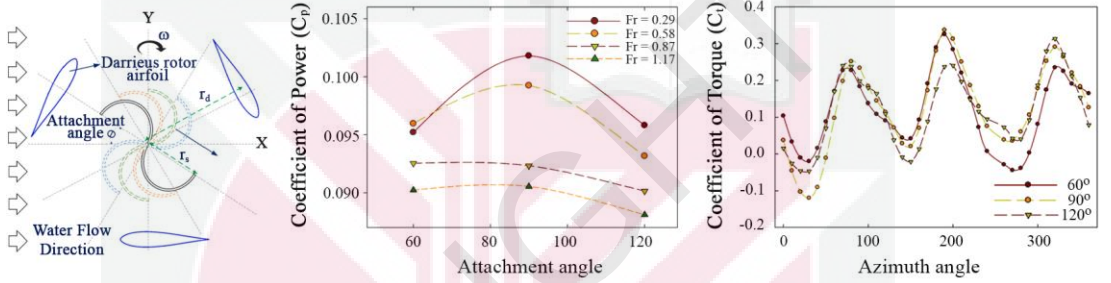
S = Savonius turbine, D = Darrieus turbine



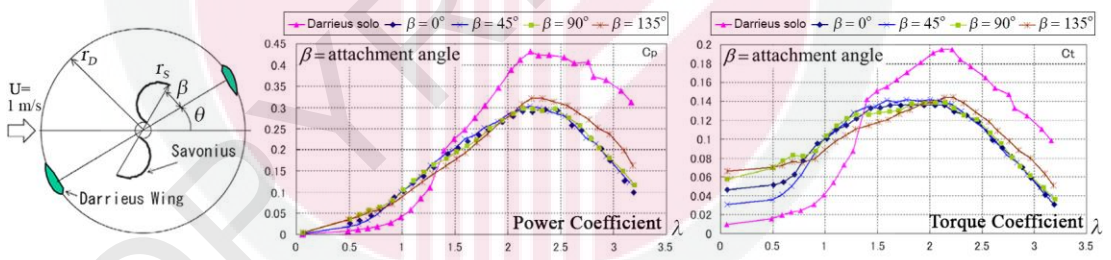
(a)



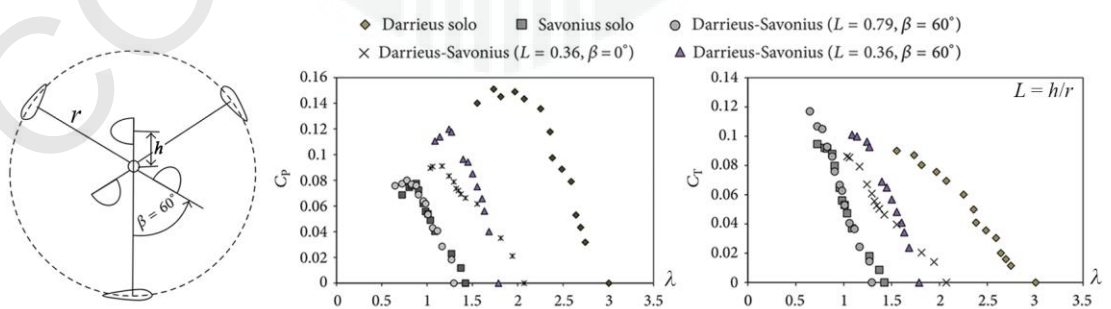
(b)



(c)



(d)



(e)

Figure 2.68 : Numerical (a) [43], (b) [42], and (c) [46] and experimental (d) [149], and (e) [154] results of the effect of attachment angles on the combined turbine performances from previous studies

2.8.4 Effect of the Rotational Velocity

In order to enhance the overall efficiency, the effect of the rotational velocities on the combined Savonius-Darrieus turbines was conducted by S. ed-Dîn Fertahi et al., 2018 [48]. The two-dimensional numerical studies were carried out on the three different rotational configurations which were 1). Synchronous coupling ($\Omega_S = \Omega_D$) and identical rotational direction, 2). Asynchronous coupling ($\Omega_S \neq \Omega_D$) and opposite rotational direction, and 3). Asynchronous coupling ($\Omega_S \neq \Omega_D$) and identical rotational direction as shown in Figure 2.69(a) - (c), respectively. The rotational speeds of each turbine were adapted by using the planetary gear train and the epicyclic trains to fix a ratio between the angular velocities of both turbines. The results could be concluded that a combined turbine with asynchronous coupling and identical rotational direction presented in Figure 2.69(c) provided the best performances compared to other configurations.

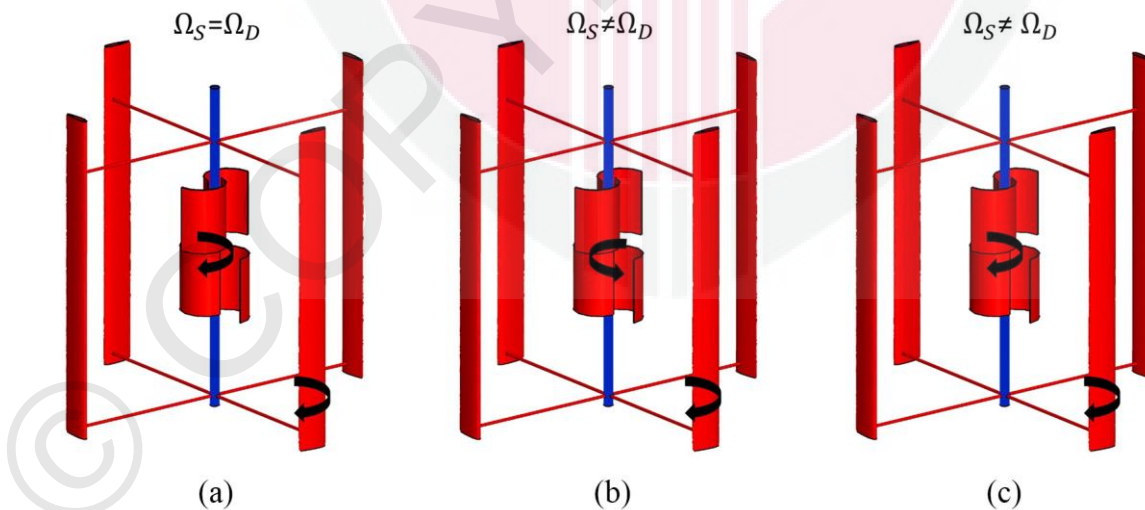


Figure 2.69 : Three different operational configurations of hybrid turbines on the effect of the rotational velocities [48]

2.9 Summary and Research Gap

After a comprehensive review of the important and recently published research that is directly relevant to the topic of this thesis. It can be summarized that the vertical-axis hydrokinetic turbines including the Savonius and Darrieus turbines are a promising solution used as renewable energy to solve the global warming situation and reduce GHG emissions.

Several experimental studies and CFD simulations have been carried out to study the effect of turbine geometry on the performance of the Savonius turbine. It was found that, the Savonius turbine is good at starting ability at lower flow speeds. However, it suffers from low performance at a high rate of flow. Several experimental and computational studies on geometric parameters of the Darrieus turbine for example, rotor diameter, chord length, blade thickness-to-chord, blade profiles, solidity, pitch angles, aspect ratios, number of blades, and blade shapes including the popular H-shape, helical-shape, egg-shape or ϕ -shape, J-shape turbines were conducted. The studies showed that a general Darrieus turbine provides better efficiency at higher operational ranges (tip speed ratio) than other turbine types but suffers from the problems of self-starting and also suffers from high cyclic fluctuation. To overcome the aforementioned problems and complement the advantages of each other, a combination of Savonius and Darrieus turbines has been proposed. The effects of different geometric parameters and turbine configurations for example compact designs, radius ratios, and attachment angles were the main concern.

Based on the review, the research gaps have been explored. The published works on the effect of the number of stages of the Savonius turbines were found mostly about the experimental works conducted on wind turbines as mentioned in table 2.9. There is however no information on the performance of hydro turbines and works using 3D CFD simulations.

Most research works on the performance of the Darrieus turbines have been carried out with straight-bladed, curved-bladed, J-shape-bladed, and helical-bladed Darrieus turbines. There was no information or study on the turbine with the delta-shaped blade.

CFD simulations on the effect of attachment angles were conducted mostly by using 2D simulations and investigating the large value of attachment angles i.e. 60° , 90° , and 120° as presented in table 2.11. Besides, simulations on the effect of the ratios of swept area known as the radius ratio were conducted and found widely only using 2D simulations. There is still a lack of work in determining the configuration parameters including attachment angles from small to large values and the ratios of swept area by using 3D CFD simulations. As a result, the current study will attempt to fill the gaps.

CHAPTER 3

METHODOLOGY

3.1 Introduction

In this chapter, the primary purpose is to present the approaches following the aforementioned objectives. The whole processes in these studies are represented by a flow chart as illustrated in Figure 3.1. In this study, there are six major procedures. To optimize the combined Savonius-Darrieus hydrokinetic turbine, the processes are going to begin with technical specification consideration and turbine selection. Plenty of literature on those turbines will be considered to compare the pros and cons during the design and conceptualization processes. Following that, the selected turbines will be designed and modeled using the Solidworks available software. Mainly, there are three types of selected models which are Savonius, Darrieus, and combined Savonius-Darrieus hydrokinetic turbine models. After that, several model designs will be exported to the next process which is the CFD optimization process. Several parameters of those turbine models will be simulated and optimized using ANSYS Fluent commercial software. Simulation data from the CFD software will be calculated and analyzed. Turbine performances including power and torque coefficient several parameters and turbines will be compared. To ensure that the simulation results are reliable, those results need to be validated and verified with previous studies either experimental or numerical results. Finally, the most optimum design of a combined Savonius-Darrieus hydrokinetic turbine will be concluded.

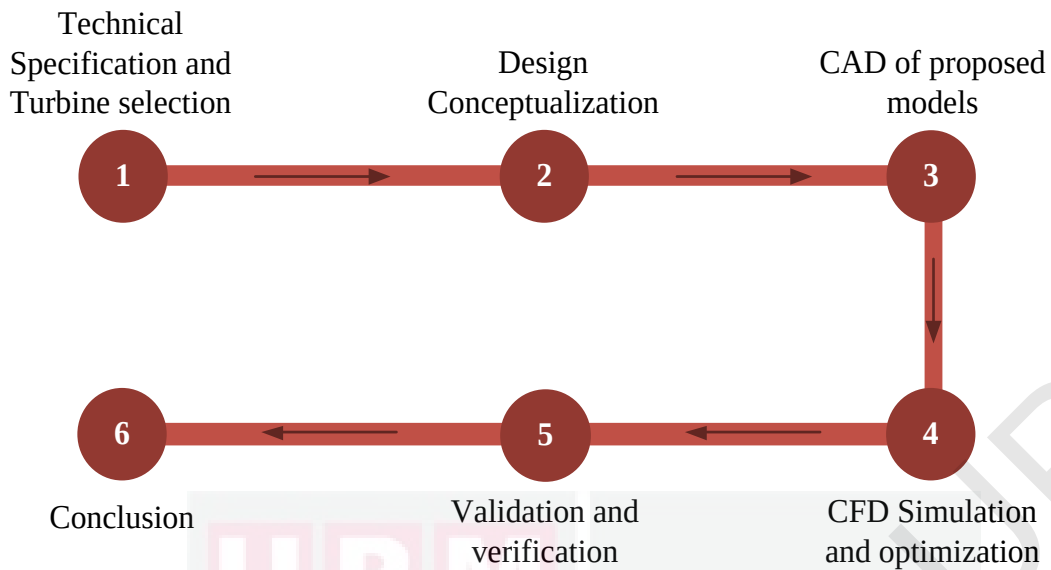


Figure 3.1 : The flow of whole methodological approaches in these studies

3.2 Turbine Design and Optimization Processes

To design and Savonius (SHKT), Darrieus (DHKT), and combined Savonius-Darrieus hydrokinetic turbines (SDHKT), information of several parameters for instance types of turbines, turbine shape and size, performances, pros and cons, and characteristic curves, need to be considered. The details of those parameters have been explained intensively in Chapter 2. In addition, an irrigation canal where a turbine will be installed also needs to be considered. Figure 3.2 is an example of turbine installation in a typical irrigation canal. The turbine usage will have one or multiple one as a turbine system as shown in Figure 3.2(a). In general, irrigation canal shapes can be straight or incline walls as depicted in Figure 3.2(b) and (c), respectively.

Size of turbine is determined according to size of an irrigation canal and a geometric parameter that is an aspect ratio (AR). Based on previous literature, optimal AR of hydrokinetic turbine provides the best performances.

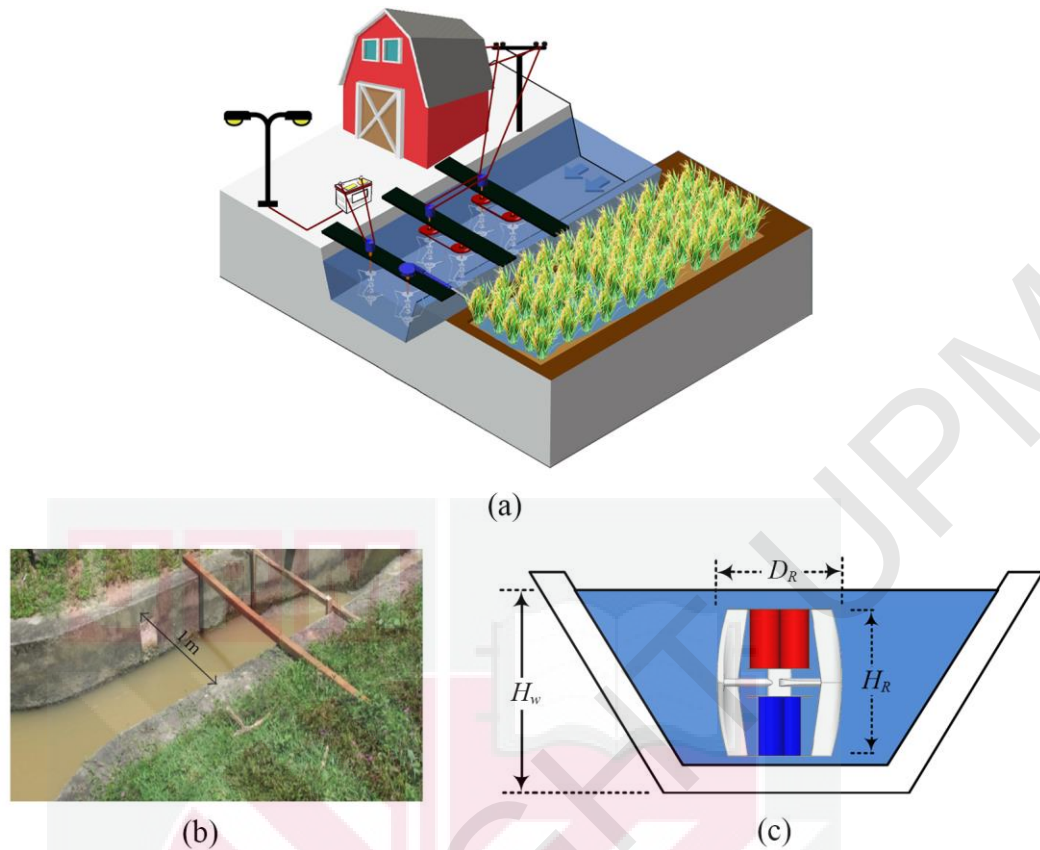


Figure 3.2 : (a) concept of turbine installation in an irrigation canal, and typical configuration of an irrigation canal with (b) straight wall and (c) inclined wall

However, turbine height and diameter are also limited due to structural reasons of canal geometry. To submerge, turbine height therefore should be shorter than height of water level. Average water flow velocity in an irrigation canal is a factor needed to be considered. In a potential site, an irrigation canal should have ample water all year round and flow inside the canal should be controllable. It is the fact that a general hydrokinetic turbine cannot operate well if the flow velocity is too low and lower than the starting point that a turbine starts to rotate. Minimum water velocity requirement depends on geometric parameter of turbine such as, shape, size as well as dynamic behavior of turbine. To optimize hydrokinetic turbines, several parameters will be investigated systematically. The flow chart of turbine design and optimization processes from the beginning until the end is presented in Figure 3.3.

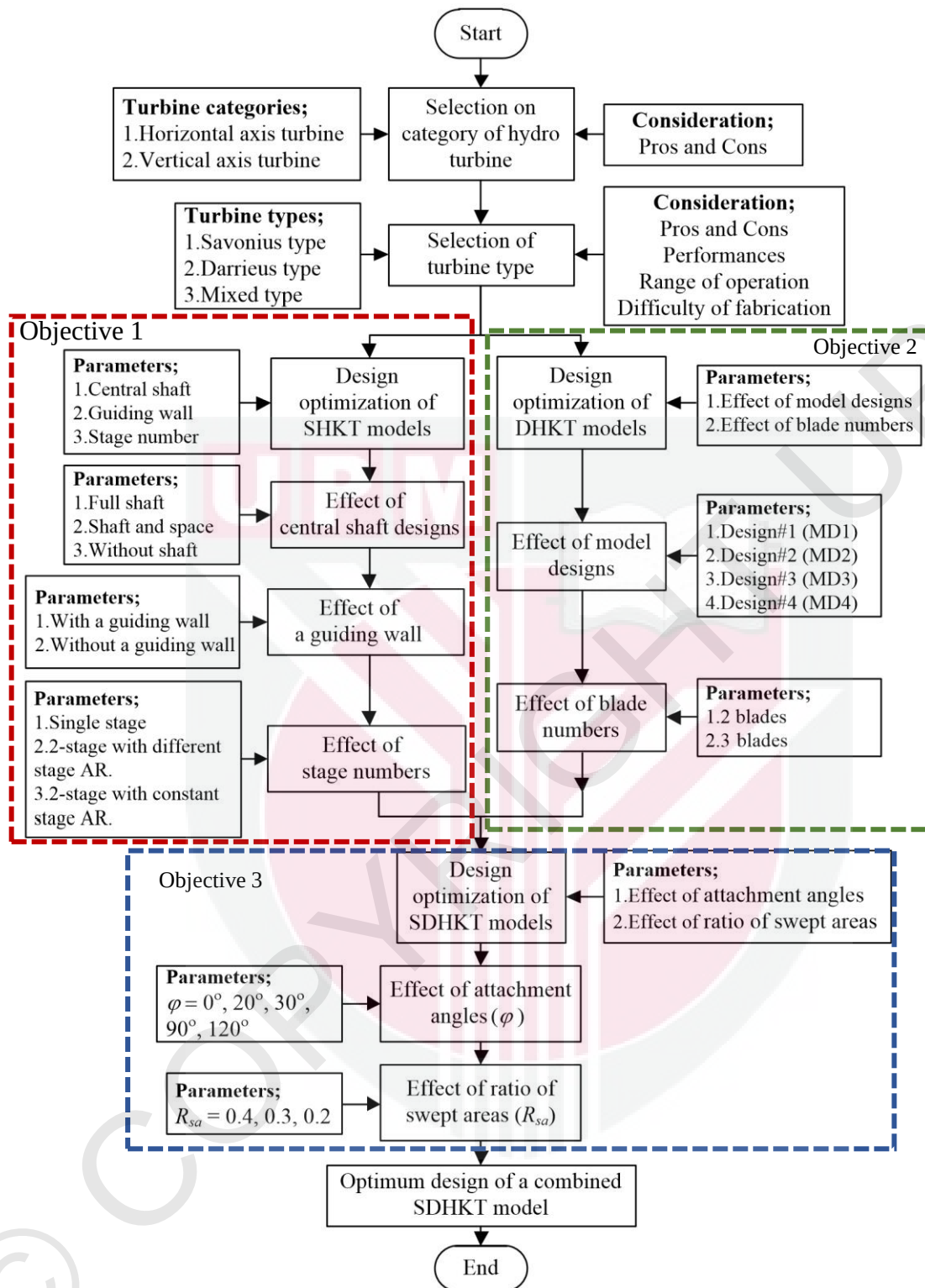


Figure 3.3 : Flow chart of turbine design and optimization processes

3.3 CAD Design

After the conceptual designs of SHKT, DHKT, and combined SDHKT models have been done. Computer Aids Design (CAD) of the proposed models including two- and three-dimensional models will be conducted next by using CAD commercial software. The details of CAD designs of those turbines will be explained in the next sections.

In this study, optimization processes of all turbine designs have been carried out by the method of computational fluid dynamic (CFD) based on numerical investigation. Basically, CFD is a branch of fluid mechanics using numerical methods and algorithms to solve and analyze problems involving fluid flow. As shown in Figure 3.4, CFD simulation processes are composed of 3 main steps [19]. Begin with pre-processing steps, this process involves problem formulation, modeling the proposed hydrokinetic turbines using the Solidworks software, defining the flow domain by the ANSYS workbench, and mesh generation employing the ANSYS mesh. Next are the computational solver steps. Physical models (material properties turbulence model), physical conditions (boundary and initial conditions), and convergence conditions will be imposed. The final is the post-processing steps. The CFD results will be calculated and plotted. Contour plots are visualized. Due to pros and cons, the ANSYS commercially available software at CAD/CAM laboratory, Faculty of Engineering, Universiti Putra Malaysia has been employed for an entire study. Therefore, parallel computing is implemented using a Dell Precision T7500 workstation with an Intel Xeon® processor of 2.3 GHz, 36-core, and 128-GB installed memory.

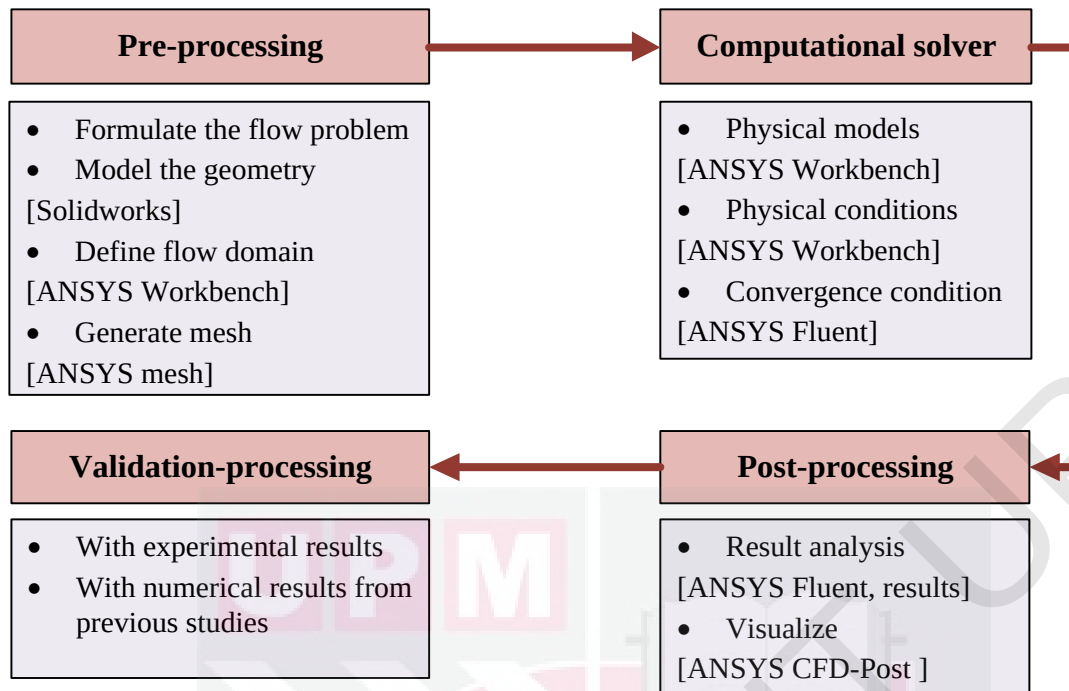


Figure 3.4 : CFD simulation processes

3.4 CFD Optimization Procedure

The flow chart as illustrated in Figure 3.5 is the entire CFD optimization process in this study. CFD simulations will be utilized to optimize the designs of the SHKT and DHKT models. Several parameters of both models have been studied so far by many researchers. However, three parameters of SHKT and DHKT models would be selected to design and evaluate the effect of those parameters. After that, optimized designs of both models would be combined as a combined SDHKT model. Finally, CFD simulations would be conducted once again with two parameters involving the combined model configurations in order to optimize the effects of attachment angles and the ratios of swept area.

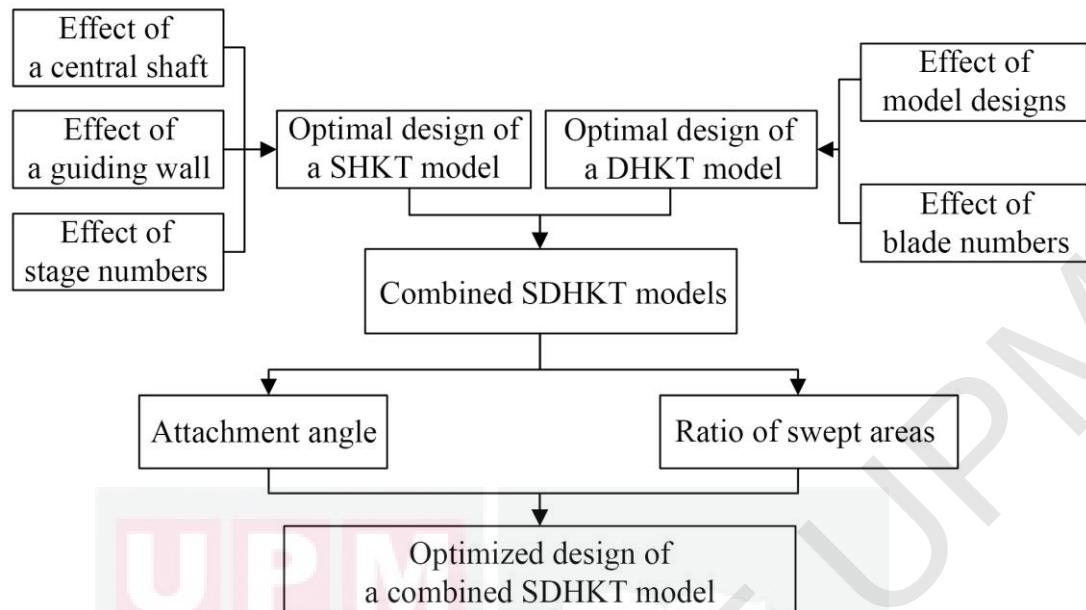


Figure 3.5 : A flow diagram of the optimization processes in the current studies

As depicted in Figure 3.6, the effects of central shaft designs and the presence of a guiding wall will be conducted using two-dimensional simulations because of the symmetric shapes of the models. The effect of central shaft designs will be considered first. The optimum design would be used to design the models with the effect of the presence of a guiding wall. In addition, three-dimensional CFD simulations would be carried out on the models having asymmetric shapes to investigate the effect of stage numbers. CFD results were also compared with results from previous studies.

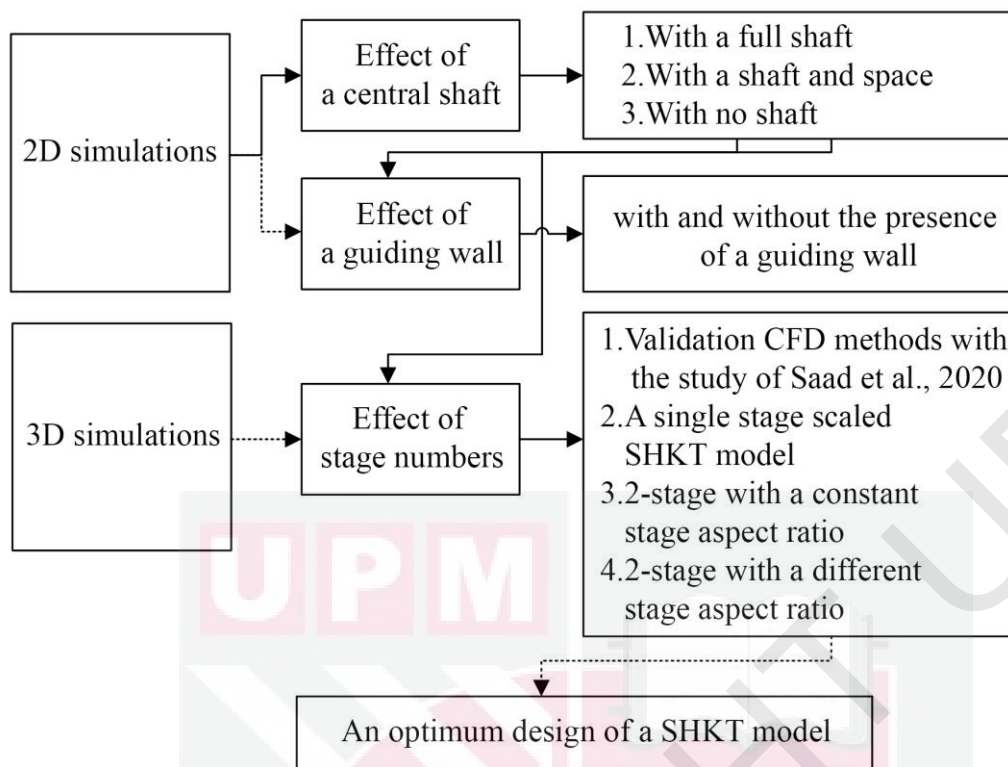


Figure 3.6 : A flow chart of the simulation procedure of SHKT models

The whole DHKT optimization process displayed in Figure 3.7 will be carried out by using 3D CFD simulations. The validation process will be conducted in order to validate and verify the CFD method that is used in this study. Then, the effect of model designs would be considered first. The two blade models with four different designs (*MD1-MD4*) corresponding with the turbine blade characteristics and cross-sectional area will be evaluated. After that, the optimum two-blade model design would be compared with the DHKT models having three blades (*MD5-MD8*). Finally, the most optimum design of DHKT would be found.

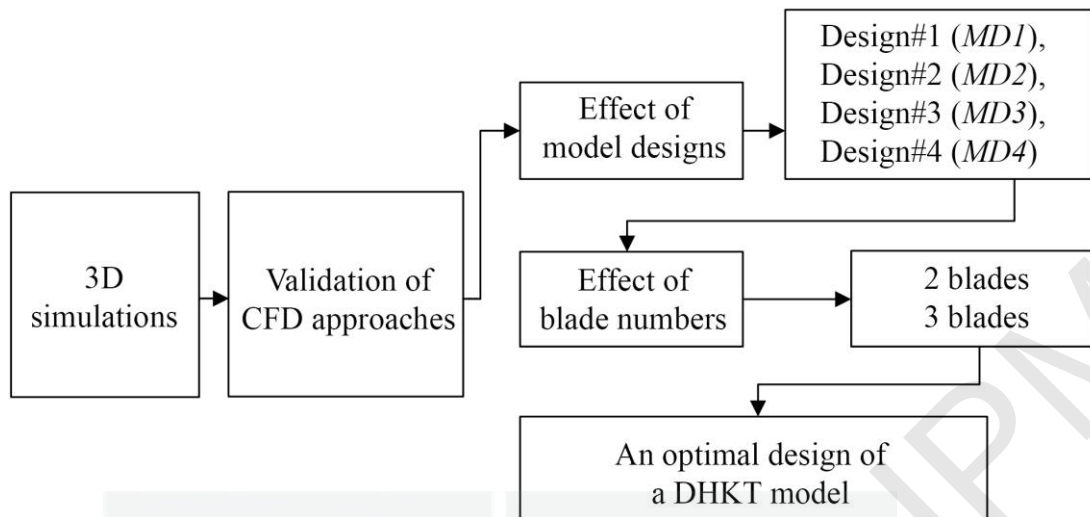


Figure 3.7 : DHKT investigation processes

The CFD optimization processes of the combined SDHKT models as presented in Figure 3.8 will be finally conducted. After combining the optimized SHKT and DHKT models, 3D simulations using ANSYS Fluent will be carried out once again with two main configured parameters which are the attachment angles (φ) and the ratios of swept area (R_{sa}). The effects on the turbine configurations between both SHKT and DHKT models will be considered. Finally, the most optimum configuration of the combined SDHKT model will be concluded.

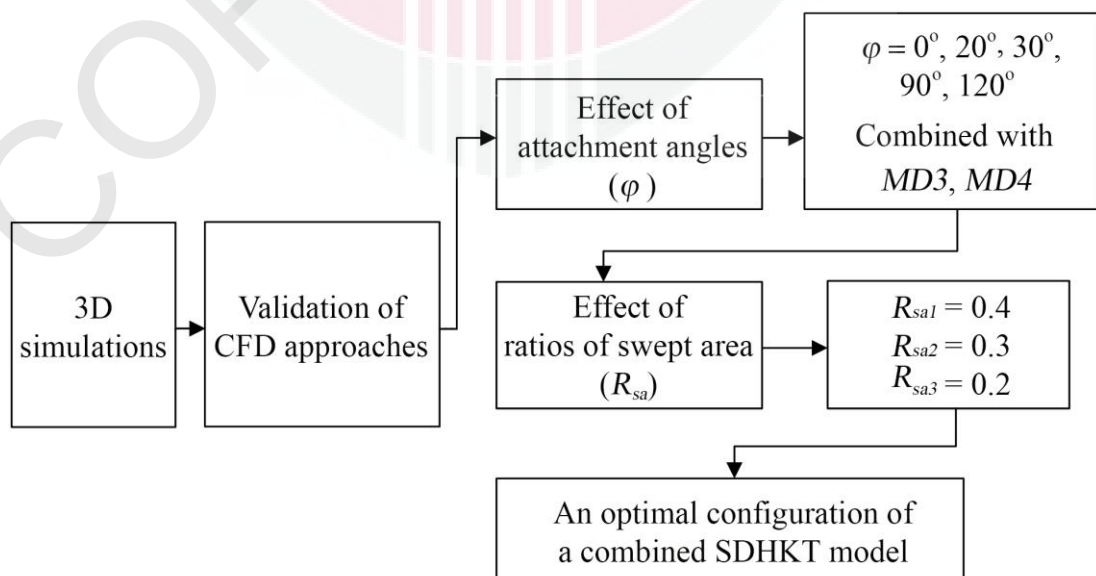


Figure 3.8 : Investigation procedure of the combined SDHKT models

3.5 Computational Methodology

To simulate 3D models, the ANSYS Fluent available commercial software based on the finite volume method was employed. The flow around the models was assumed to be three-dimensional, turbulent, and transient incompressible fluid flow. The time-dependent Unsteady Reynolds-Averaged Navier Stokes (URANS) equations with the pressure-based formulation were solved with the pressure-velocity coupling solution scheme employing the Semi-Implicit Method for Pressure Linked Equation (SIMPLE) algorithm. Moreover, the spatial discretization of the pressure, momentum, and turbulence equations was computed with the second-order upwind scheme.

3.5.1 Governing Equations

The governing equations including the continuity and momentum equations of the flow for unsteady Newtonian incompressible turbulent flow are given in the equation (Eq. 3.1) and (Eq. 3.2) by the form of index notation.

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \quad (\text{Eq. 3.1})$$

$$\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{P}}{\partial x_i} + \frac{\mu}{\rho} \frac{\partial^2 \bar{u}_i}{\partial x_j^2} - \frac{\partial}{\partial x_j} (\bar{u}'_i \bar{u}'_j) \quad (\text{Eq. 3.2})$$

3.5.2 Time Step Size

To obtain an accurate result, a time step (Δt) size is determined by the azimuthal increment. For CFD simulations using the sliding mesh technique, it is calculated based on the degree of revolutions. It means that the unit time step equals to a time that the turbine rotates in one degree. Hence, the time step size is varied depending on the desired rotational speed of a turbine model or a tip speed ratio (λ).

The optimal time step size needs to be considered using the time step independency study. Time step of 0.5° increment can be calculated by equation (Eq. 3.3). Based on the previous study, the time step size of 0.5° of rotation was precise enough [31],[70] and acceptable to use as a time step size for all simulations.

$$\Delta t (0.5^\circ) = \frac{\pi}{360\omega_o} \quad (\text{Eq. 3.3})$$

3.5.3 Convergence Criteria

In order to consider whether the simulation results have already been converged or not, the repeated average power coefficients ($c_{p,avg}$) were calculated and used. In fact, when the convergent criterion (ε) expressed in equation (Eq. 3.4) is less than 1% [33] at least two consecutive revolutions, it means that the simulation results have already reached the convergence criterion.

$$\varepsilon = \frac{c_{p,avg(n+1)}(\theta) - c_{p,avg(n)}(\theta)}{c_{p,avg(n)}(\theta)} < 1\% \quad (\text{Eq. 3.4})$$

3.6 CFD Procedure of the Savonius Hydrokinetic Turbine

3.6.1 Two-Dimensional Simulations of the SHKT Models

Two dimensional simulations are conducted to determine the effect of central shaft designs and evaluated with and without the presence of a guiding wall on performances of the designed SHKT models. Details of the model designs for both parameters and computational approaches for 2D simulations are described in this subsection.

3.6.1.1 Design of 2D SHKT Models

A conventional SHKT model with a central shaft and its designed abbreviation is depicted in Figure 3.9(a). To investigate influences of central shaft designs, A conventional SHKT model was modified with 3 different designs including (I) with a full shaft, (II) with a shaft and space, and (III) without shaft as shown in Figure 3.9(b). As depicted in Figure 3.9(b), the overlap length (e) of 0.006 m. between two buckets was fixed, while the turbine shaft diameter (D_s) was varied. Therefore, the overlap ratios (β) of the design I, design II, and design III were 0, 0.0009, and 0.06, respectively. The model initial position (θ) where $\theta = 0^\circ$ was set as in Figure 3.9(a). The models were made from the ABS (Acrylonitrile Butadiene Styrene) with $\rho_{abs}=1,020 \text{ kg/m}^3$.

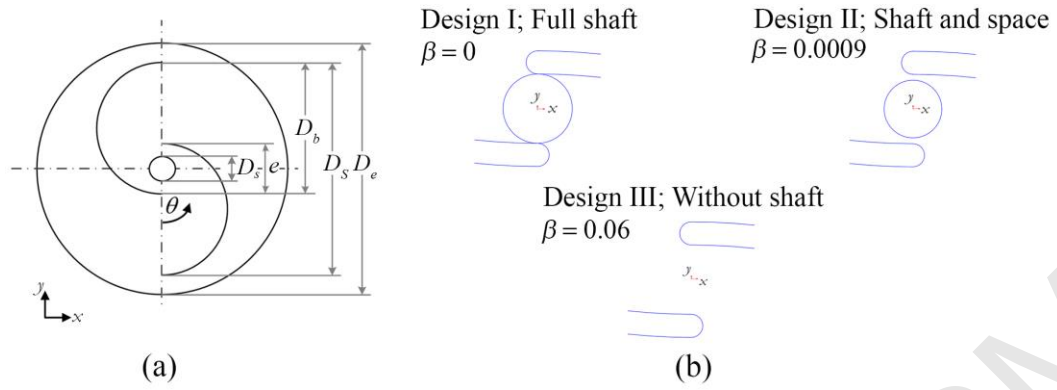


Figure 3.9 : (a) a schematic diagram of a 2D SHKT model, and (b) Designs of a central shaft

The optimum central shaft design from the previous simulation was modified by adding a guiding wall. As shown in Figure 3.10, one end of a guiding wall was located along at the center half of domain and far away from the turbine center (S_G), while another end was mounted at a side of domain walls. The incline angle (α) was fixed at 45° . The design specifications of the designed 2D SHKT models used in current studies are described in Table 3.1.

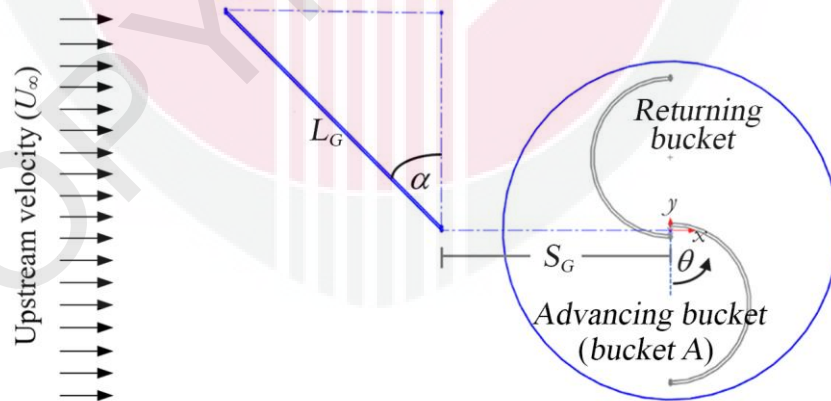


Figure 3.10 : The design of a guiding wall configuration

Table 3.1 : Detail specifications of the designed 2D SHKT models

Parameters	Value		
SHKT diameter (D_s)	0.2 m		
Bucket diameter (D_b)	0.100 m		
End plate diameter (D_e)	0.202 m		
Thickness (t)	0.002 m		
Shaft diameter (D_s)	0	0.00591 m	0.006 m
Spacing ($e - D_s$)	0.006 m	0.00009 m	0
Length of a guiding wall (L_G)	0.424 m		
Location of a guiding wall (S_G)	0.150 m		
Incline angle (α)	45°		

3.6.1.2 Computational Methodology

The time-dependent Unsteady Reynold-Averaged Navier–Stokes (URANS) method was utilized to solve the complex flow problems. For a realistic simulation of flow across the 2D SHKT model, several turbulence models have been considered. There was no agreement on the most appropriate turbulence model adopted in CFD adopted in CFD simulations even in two or three-dimension simulations. However, the standard $k-\varepsilon$ model as mentioned intensively in Chapter 2 was the simplest turbulence model and widely accepted [19],[79],[88]. The model was employed to solve two separated transport equations which transport equations which are the turbulent kinetic energy turbulent kinetic energy (k) equation and dissipation rate (ε) of the turbulent kinetic energythe turbulent kinetic energy equation as expressed as equations (Eq. 3.5) and (Eq. 3.6).

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} + G_k - \rho \varepsilon \quad (\text{Eq. 3.5})$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j} \left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} \quad (\text{Eq. 3.6})$$

This model was suitable for flow with fully turbulent [58]. Due to many advantages e.g. robustness, economy, and reasonable accuracy [40], the standard k - ε model was employed in several previous CFD studies and those results were compared with other turbulence models [36],[54],[85],[88]. It found that the results from the standard k - ε model showed a general agreement with experimental data. Therefore, the model with a scalable wall function could be accepted and employed to simulate all cases in the present studies.

3.6.1.3 Computational Domain

The domain size was described in terms of a proportion of the model diameter (D_s). The dimension of the domain was considered by following the published literature such as Saad et al., 2020 [36], Yaakob et al., 2010 [91], and Mendoza et al., 2020 [68] to confirm the negligible effects of domain walls. To ensure the negligible domain wall effect, the bigger the domain size, the better the simulated accuracy. However, the oversize domain might take more computational costs such as bigger hard disk spaces and longer simulation time. Kumar et al., 2016 [8] and Kumar and Saini 2017 [54] set the domain width the domain width of $3.4D_r$ and length of $18.75D_r$ in their studies and found that the minimum domain width and length did not interfere with the flow field across the turbine.

In the present studies, therefore the domain size for all simulations was imposed as illustrated in Figure 3.11. The domain was symmetrical rectangular, and a SHKTThe domain was symmetrical rectangular, and a SHKT model was placed at the center of the domain which separated the domain into front-back regions and upper-lower regions equally. The domain width and length were $3D_s$ and $6D_s$, respectively. The domain was divided into two main regions which were a rotating region or an inner domain and a stationary region or stationary region or an outer domain. Both regions were connected regions were connected by the interface line to ensure the continuity of the flow field across the domain [54].

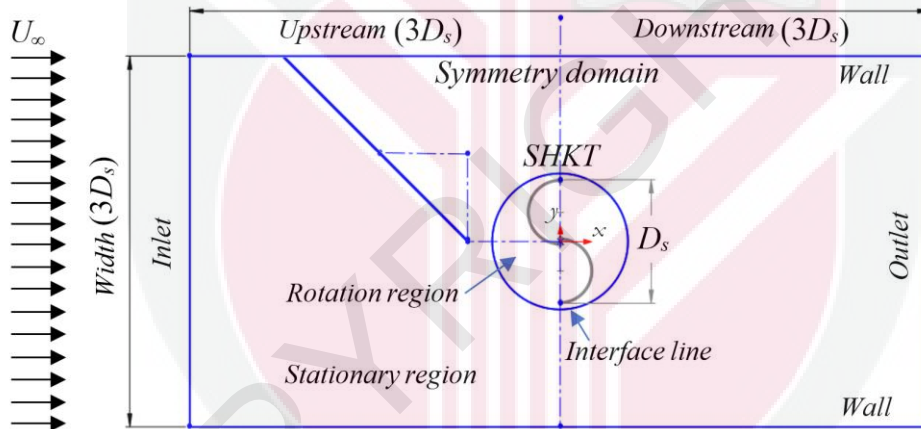


Figure 3.11 : A symmetric computational domain for 2D simulations

3.6.1.4 Mesh Configurations and Boundary Conditions

An ANSYS mesh module was used to generate 2D meshing. The unstructured triangular meshes as illustrated in Figure 3.12 in both the outer and inner domains were applied. Moreover, the boundary layers were set using inflation with a smooth transition rate of 1.2 on the turbine bucket surfaces. In the dynamic mesh motion technique, the turbine rotated freely corresponding with the upstream flow velocity together with three dynamic mesh methods; Smoothing, Layering, and Remeshing. It

is suited for boundary movement. In addition, the symmetrical boundary conditions were imposed for all lateral walls. The uniform upstream water flow velocity (U_∞) with four different variations e.g. 0.3, 0.5, 0.8, and 1.1 m/s entered perpendicularly at the inlet boundary. The total pressure at the outlet boundary as well as 5% turbulent intensity (I_t) and turbulent viscosity ratio of 10 were imposed for all simulations. Finally, the no-slip boundary condition was defined at all surfaces of all SHKT models and lateral walls.

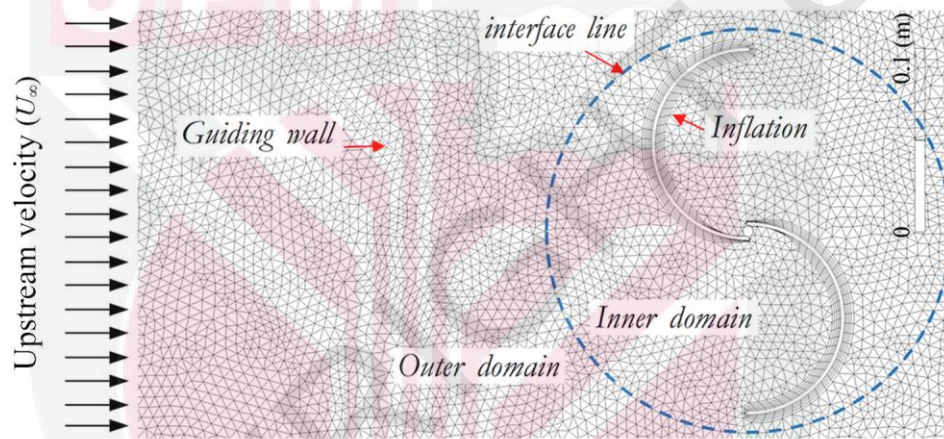


Figure 3.12 : The 2D mesh configuration

3.6.1.5 Mesh Independency Study

To ensure that the mesh configuration was valid, the mesh independence study was conducted. Six mesh refinement levels were tested. It found that the average torque of refinement level 5 was 10% higher than that of refinement level 4. Besides, average torques from mesh refinement level 5 to 7 as depicted in Figure 3.13 were almost similar with differences lower than 1%. Therefore, mesh refinement level 6 which was the medium value between the level 5 to 7 was selected to employ as the mesh size for all simulations.

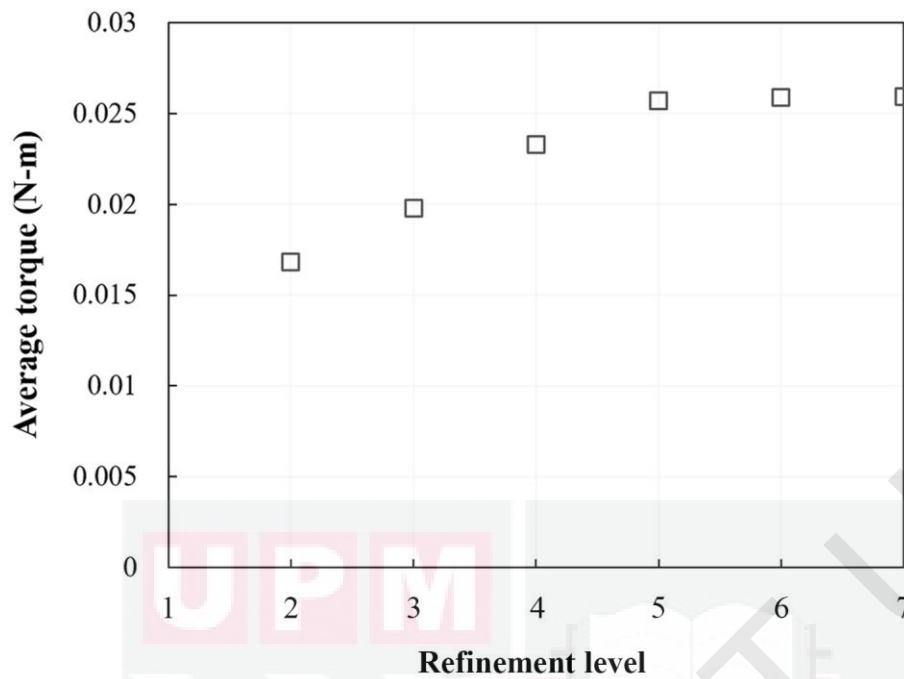


Figure 3.13 : Mesh independency study of a 2D SHKT model

3.6.2 Three-Dimensional Simulations of the SHKT Models

3D simulations of SHKT models were conducted to mainly investigate the effect of stage numbers. Simulation processes could be classified into four main cases. Case 1 was carried out in order to validate and verified 3D-CFD approaches with results from the previous study. Case 2-4 were additionally performed to investigate the effect of stage numbers on the performances of SHKT models. Details of all cases are as follow;

Case 1; Verify CFD methods with the study of Saad et al., 2020 [36].

Case 2; A single stage scaled SHKT model.

Case 3; A 2-stage SHKT model with a different stage aspect ratio.

Case 4; A 2-stage SHKT model with a constant stage aspect ratio.

3.6.2.1 Physical Design of 3D SHKT Models

To get an optimum designedget an optimum designed shape of 3D SHKT models, first of all, several parameters were considered using information from the previous studies. It was found that the optimum bucket number was two [38],[58],[83]. The semicircular shape of buckets was better than the elliptic [58],[81]. A Savonius turbine with upper and lower end plates [36], and with circular shape of end plates [82] showed better performance. Besides that, the optimum end plate diameter (D_o) was 1.1 times the rotor diameter (D_r) [36],[155],[156]. The aforementioned parameters were combined and designed to develop a 3D SHKT model called a conventional 3D SHKT model as illustrated in Figure 3.14. A conventional 3D SHKT model depicted in Figure 3.14 was modified by using the Solidworks software to be able to verify and validate CFD approaches, and to compare the results with the study of Saad et al., 2020 [36].

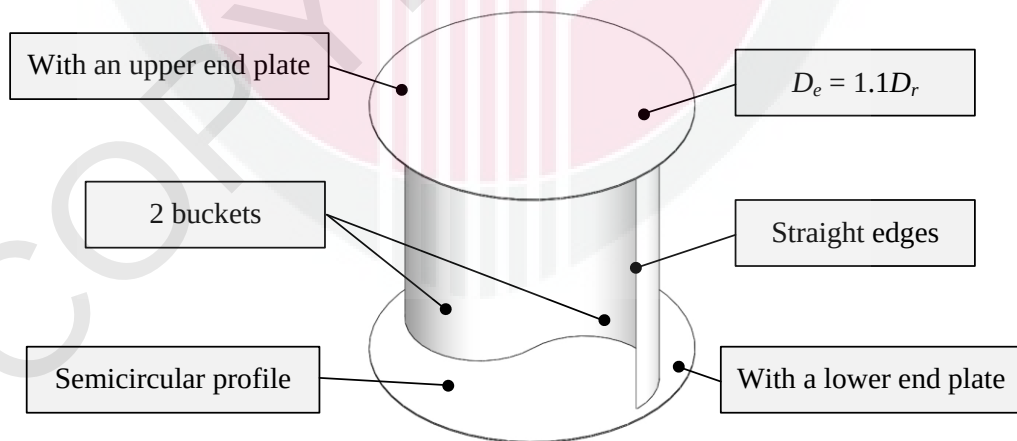


Figure 3.14 : A 3D conventional SHKT model

The SHKT model used in case 1 was made from Acrylonitrile Butadiene Styrene (ABS) ($\rho_{\text{abs}}=1,020 \text{ kg/m}^3$). The rotor height (H_r) and diameter (D_r) were 400 mm, with an aspect ratio of one. The overlap ratio ($\beta = e/D_b$) and the twist angle (ψ) were designed to equal to zero. Every part of the model had a thickness of 2 mm. Schematic diagrams and detailed specifications of the model of case 1 are presented in Figure 3.15, and Table 3.2, respectively.

Case 1

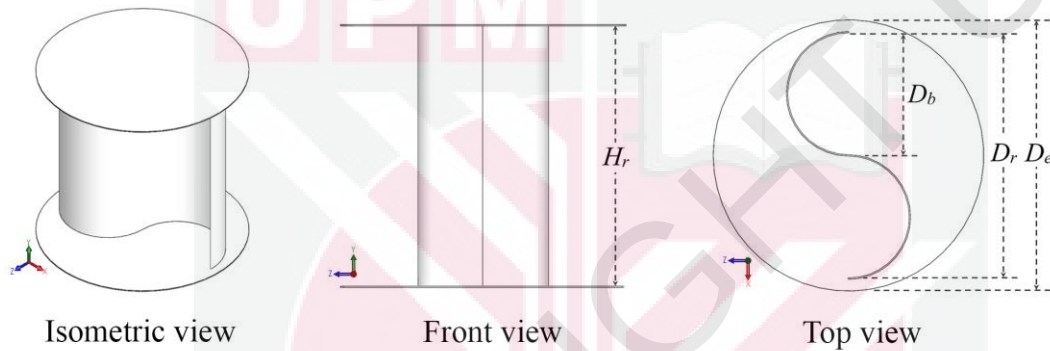


Figure 3.15 : Isometric, front, and top views of the SHKT model of case 1

Table 3.2 : Design parameters for the SHKT model of case 1

Parameter	Value
Rotor height (H_r)	400 mm.
Rotor diameter (D_r)	400 mm.
Endplate diameter (D_o)	440 mm.
Thickness	2 mm.
Circular end plate size ratio (D_o/ D_r)	1.1
Number of buckets	2
Bucket shape	Semi-circular
Aspect ratio (H_r / D_r)	1
Overlap ratio (β)	0
Twist angle (ψ)	0°

To investigate the effect of stage numbers, the conventional SHKT model as depicted in Figure 3.14 and Figure 3.15 was scaled down firstly by a factor of 0.2. It was called the scaled SHKT model. Therefore, the model presented in Figure 3.16(a) had a rotor height (H_r) and diameter (D_r) of 80 mm. The model was employed in the simulation of case 2 which was a single-stage scaled SHKT model. In addition, other parameters for instance number of buckets, aspect ratio (H_r/D_r), end plate size ratio (D_o/D_r), overlap ratio (β), and twist angle (ψ) were kept constant as similar to the design of a model in case 1.

The scaled SHKT model was modified once again becoming two-stage SHKT models. Based on a stage aspect ratio (H_s/D_r), basically there were two designs as illustrated in Figure 3.16 that were the two-stage SHKT models with the different stage aspect ratio (case 3), and constant stage aspect ratio (case 4). The referred term of different, or constant stage aspect ratio was compared with the stage aspect ratio of a model in case 2 (H_{s2}/D_r). The stage aspect ratio of the model used in case 3 (H_{s3}/D_r) was 0.5 of each or a half of that one of case 2. While the model with a constant stage aspect ratio (H_{s4}/D_r) was equal to 1 for each stage. The details of the geometric parameters of each case are presented in Table 3.3.

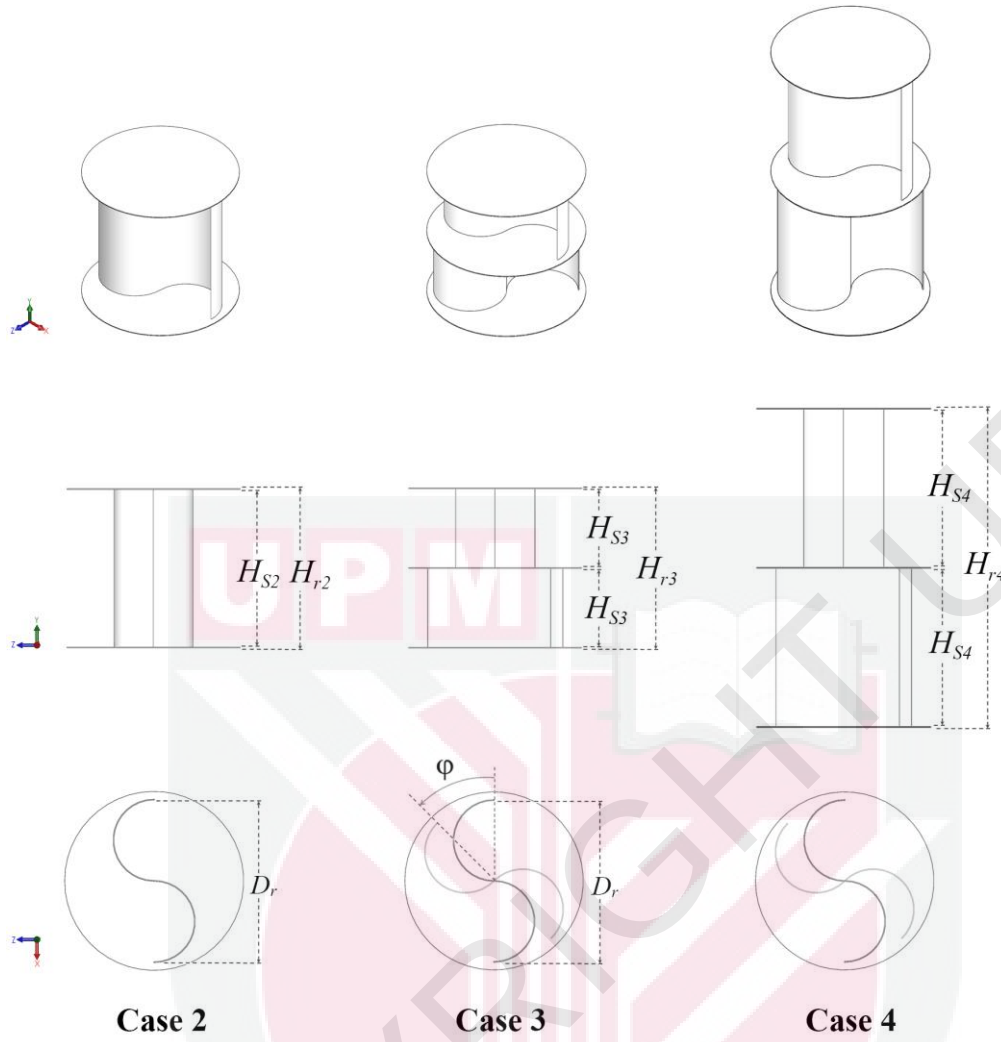


Figure 3.16 : Schematic configurations of the SHKT model of the case 2 - 4

To sum up, 3D-CFD simulations of SHKT models with the effect of stage numbers presented in this chapter were set into four main cases as presented in Figure 3.17. Case 1 was the CFD simulations of a conventional SHKT model validated and verified the CFD approaches and results with the study of Saad et al., 2020 [36]. Case 2 was CFD simulations of a single-stage scaled SHKT model. Simulations of case 3 were conducted on the model with different stage aspect ratios. Finally, case 4 was the CFD investigation of a two-stage SHKT model with a constant stage aspect ratio.

Table 3.3 : Design parameters of the SHKT models of the case 2 – 4

Parameter	Case 2	Case 3	Case 4
Stage height (H_s)	$H_{s2} = 80$ mm.	$H_{s3} = 40$ mm.	$H_{s4} = 80$ mm.
Rotor height (H_r)	$H_{r2} = 80.8$ mm.	$H_{r3} = 81.2$ mm.	$H_{r4} = 161.2$ mm.
Rotor diameter (D_r)	80 mm.	80 mm.	80 mm.
Stage aspect ratio (H_s/ D_r)	1	0.5	1
Rotor aspect ratio (H_r/ D_r)	1.01	1.015	2.015
Attachment angle (φ)	-	45°	45°
Endplate diameter (D_o)		88 mm.	
Thickness		0.4 mm.	
Number of buckets		2	
End plate size ratio (D_o/ D_r)		1.1	
Overlap ratio (β)		0	

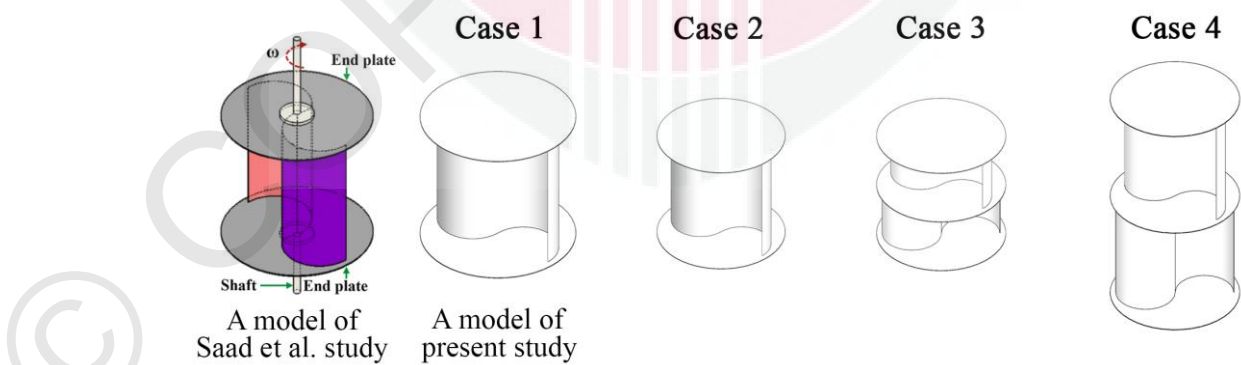


Figure 3.17 : The SHKT models of the case 1 – 4

3.6.2.2 Computational Turbulence Model

There is no agreement on the most appropriate model adopted in CFD simulations. However, according to previous studies mentioned in Chapter 2, the Shear Stress Transport (SST) $k-\omega$ turbulence model which is a hybrid model consisting of the $k-\epsilon$ and $k-\omega$ combination has abilities to capture flow structures in the boundary layer as well as the free stream regions [31]. It has been widely used in simulations of the vertical axis turbine. The studies of Saad et al., 2020 [36] showed the comparison between the predicted results using the SST $k-\omega$ turbulence model and the experimental measurements of Damak et al. 2018 [127], and the experimental and numerical results of Kothe et al. 2020 [37]. Those results as seen in Figure 3.18 showed good agreement among those results.

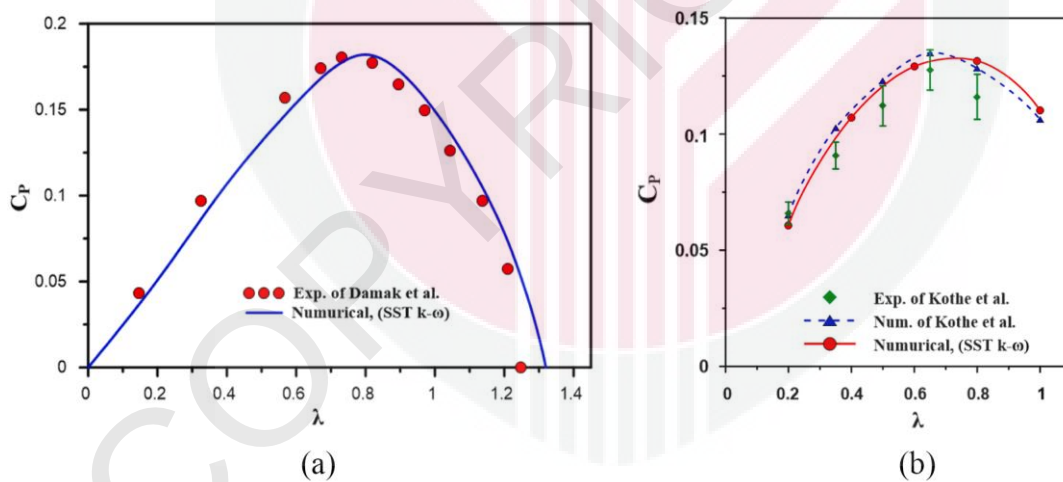


Figure 3.18 : Comparison between the predicted power coefficient versus tip speed ratio and experimental results of (a) Damak et al. 2018 [157] and experimental and numerical results of (b) Kothe et al. 2020 [37]

Therefore, the SST $k-\omega$ turbulence model will be employed for all simulations as an acceptable turbulence model to investigate the performance and flow characteristics of the designed SHKT models. The transport equation for the SST $k-\omega$ can be expressed as equation (Eq. 3.7) and (Eq. 3.8).

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k - Y_k \quad (\text{Eq. 3.7})$$

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_j}(\rho \omega u_j) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + G_\omega - Y_\omega \quad (\text{Eq. 3.8})$$

3.6.2.3 Computational Domain

A rectangular cube computational domain was created to simulate all cases of 3D SHKT models. The dimensions of a domain as depicted in Figure 3.19(a) were represented in the form of multiples of the model diameter (D_s) which were $10D_s$, $10D_s$, and $25D_s$ in X, Y, and Z directions, respectively. The SHKT model was located at the origin of a Cartesian coordinate system (0, 0, 0) which was downward $5D_s$ from the inlet boundary. The model initial position or an azimuth angle (θ) where $\theta = 0^\circ$ is depicted in Figure 3.19(b). The domain consisted of two main regions, a stationary domain, and a rotating domain surrounded a SHKT model. Both regions were separated by a rotating cylindrical surface called an interface boundary where the continuity of absolute velocity was set to ensure the continuity of the flow field and the rotating region was allowed to rotate freely with different rotational speeds whereas the rest was kept as fixed [19],[23],[53],[54].

As seen in Figure 3.19(c), (d), and (e), the model was located inside and at the center of a rotating cylinder which has a diameter of $1.5D_s$ and 0.02 m. of top and bottom gaps from the models. Because the model sizes used in this study were different, the domain size of case 1 hence was bigger than the rest cases whereas the size of the case 2-4 was equal.

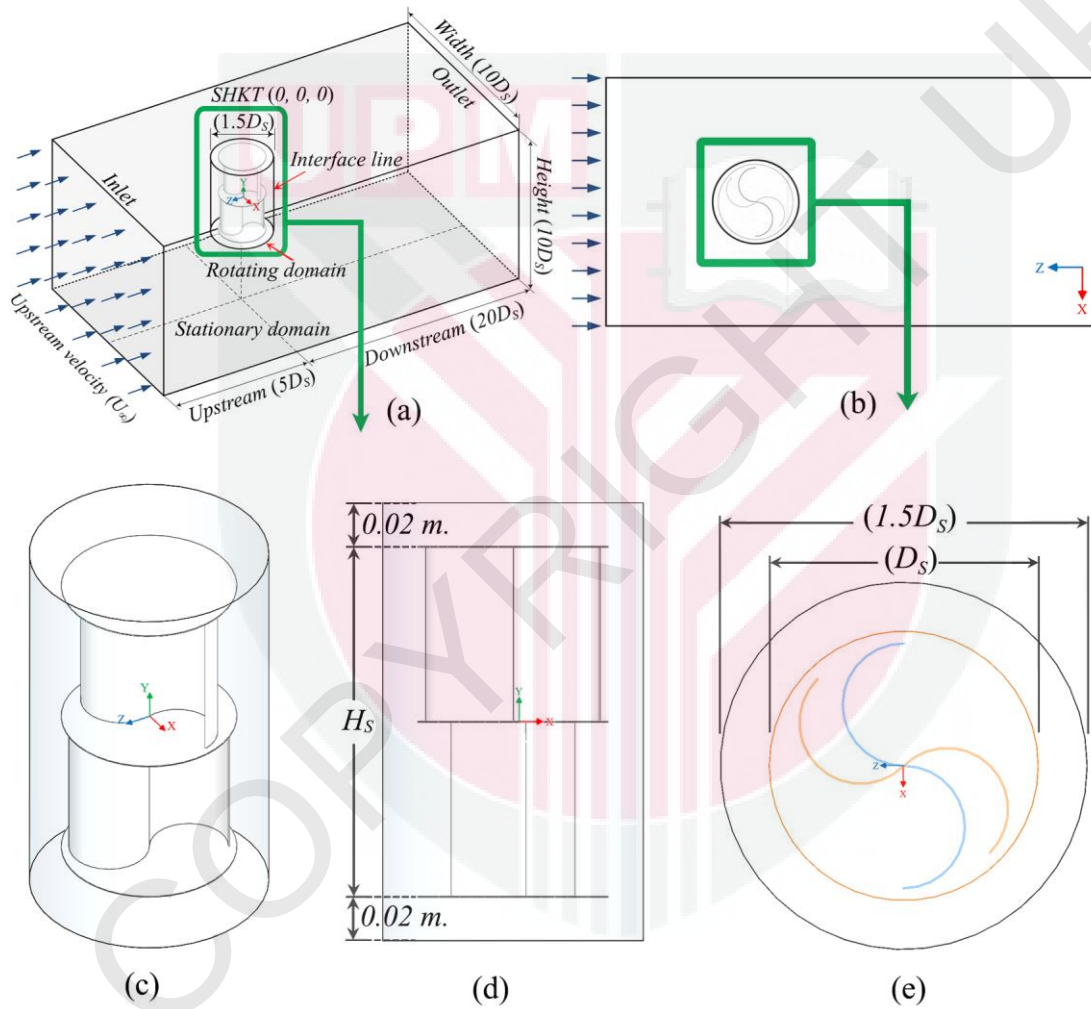


Figure 3.19 : Schematic representation of SHKT computational domain

3.6.2.4 Boundary and Initial Conditions

Due to the fact that the sliding mesh technique was employed for all simulations of the designed SHKT model. Therefore, the boundary and initial conditions utilized in this study will be set specifically for the sliding mesh method. For simulations using this technique, SHKT models will be forced to rotate constantly in the positive direction with different rotational speeds (ω_b) under constant upstream flow velocities (U_∞). Therefore, a variation in the tip speed ratio can be observed.

To validate the CFD approaches used in his study, an airflow velocity (U_∞) of 6 m/s was imposed similar to Saad et al.,2020 for the simulation of case 1. For the case 2-4, a water flow velocity of 1.5 m/s was set. The outlet total pressure was set as zero. The turbulence intensity (I) of 0.1% and turbulent viscosity ratio of 10 were imposed. For all surfaces of a model and flow channel, the standard wall roughness and non-slip condition were assumed. The boundary and initial conditions used in the 3D simulations of SHKT models are represented in Table 3.4.

Table 3.4 : The boundary and initial conditions for 3D simulations of combined SDHKT models

Location	Boundary and initial conditions	Condition	Value
Inlet	Velocity inlet	Uniform flow, Normal to the boundary	Case 1; 6 m/s, airflow Case 2-4; 1.5 m/s water flow
Outlet	Pressure outlet	Total pressure	0 (gauge pressure)
Domain channel	No slip walls Stationary wall	Standard wall roughness	0.5
DHKT surfaces	No slip walls Stationary wall	Standard wall roughness Rotate at the setting speed	0.5
Flow condition		Turbulent intensity (I)	0.1%
		Turbulent viscosity ratio	10
Interface		The link of the rotating and stationary region	Coupled surface

The convergence criteria will be imposed to be equal to 10^{-5} for the residual of all equations including continuity and momentum equations. The maximum iterations were set as 100 per time step. The summary of the CFD solver setting used for the simulations of the case 1 – 4 and used in the study of Saad et al., 2020 [36] is represented in Table 3.5.

Table 3.5 : Comparison of solver setting between the case 1 and the study of Saad et al., 2020

Parameters	Saad et al., 2020	Case 1	Case 2 - 4
Flow			
-Upstream velocity	6 m/s	6 m/s	1.5 m/s
-Medium fluid	Wind flow	Wind flow	Water flow
Material of model	Not mentioned	ABS	ABS
Turbulence model	SST $k-\omega$	SST $k-\omega$	SST $k-\omega$
Pressure-velocity coupling			
-Scheme			
Spatial Discretized	SIMPLE	SIMPLE	SIMPLE
-Pressure equation,			
-Momentum equation,	Second order	Second order	Second order
-Turbulent kinetic energy	upwind	upwind	upwind
k equation,			
-Specific dissipation rate			
ω equation,			
-Transient formulation			
Mesh motion technique	sliding mesh technique	sliding mesh technique	sliding mesh technique
Maximum iterations per time step	100	100	100
Convergence criterion	5 revolutions	$\varepsilon < 1\%$	$\varepsilon < 1\%$
-Continuity equation	10^{-5}	10^{-5}	10^{-5}
-The rest equation	10^{-5}	10^{-5}	10^{-5}

3.6.2.5 Mesh Configurations

The ANSYS mesh module was employed to generate 3D unstructured tetrahedral meshes. To validate CFD approaches of the 3D conventional SHKT model simulations with the numerical results of Saad et al., 2020 [36], firstly mesh configurations in this study were set following the Saad et al., 2020 [36] mesh methods that already conducted mesh independency study using seven different mesh resolutions. It was found that there were no significant variations in all computed parameters with increasing mesh numbers over 2.5 million elements. As a result, mesh elements equal to or over 2.5 million elements were good enough to be imposed for the 3D SHKT model simulations. Moreover, the inflation with the first layer height of 0.5 mm., and 10 total layers with a growth rate of 1.1 were also applied to the rotor walls. Figure 3.20 and Table 3.6 show mesh configuration and mesh details for 3D SHKT simulations of case 1. It can be seen that finer meshes have been created in the rotational region as compared to the stationary zone.

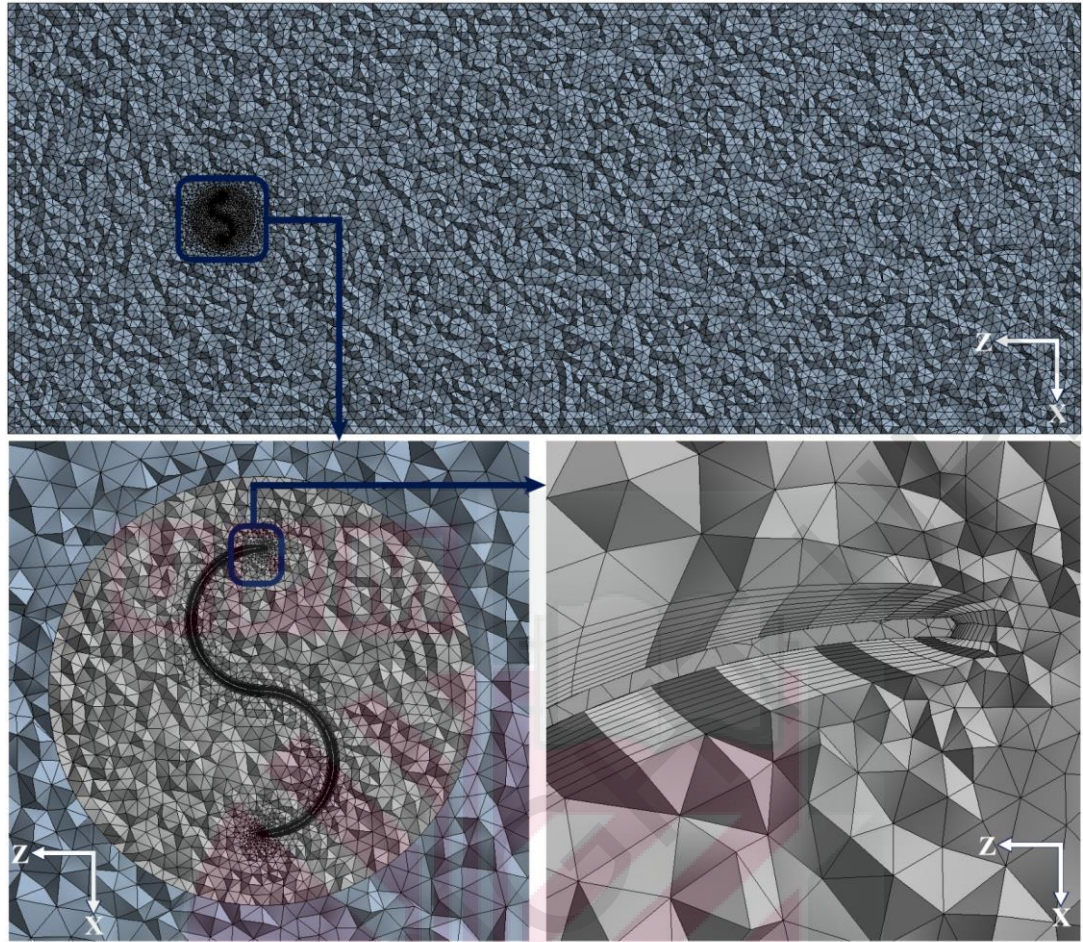


Figure 3.20 : Mesh configuration of a 3D SHKT model using in case 1

Table 3.6 : Mesh details of case 1 compared with the study of Saad et al., 2020

Parameters	Case 1	Study of Saad et al., 2020
Mesh configurations		
<i>-Element size</i>		
; Rotating region	35 mm.	No information
; Stationary region	115 mm.	No information
<i>-Number of elements</i>		
; Rotating region	2.54 million cells	No information
; Stationary region	0.96 million cells	No information
; Total element	3.50 million cells	2.5 million cells
<i>-Inflation</i>		
First layer cell	0.5 mm.	0.5 mm.
thickness	10	10
Total layers	1.1	1.1
Growth rate		

Mesh configurations used for the 3D SHKT simulations of the case 2-4 are depicted in Figure 3.21. Finer meshes were generated in the rotational region as compared with the stationary zone. the inflation for each case was fixed as similar to one of case 1. A total number of mesh elements was kept over 2.5 million cells, though the domain sizes of the case 2-4 were smaller than case 1.

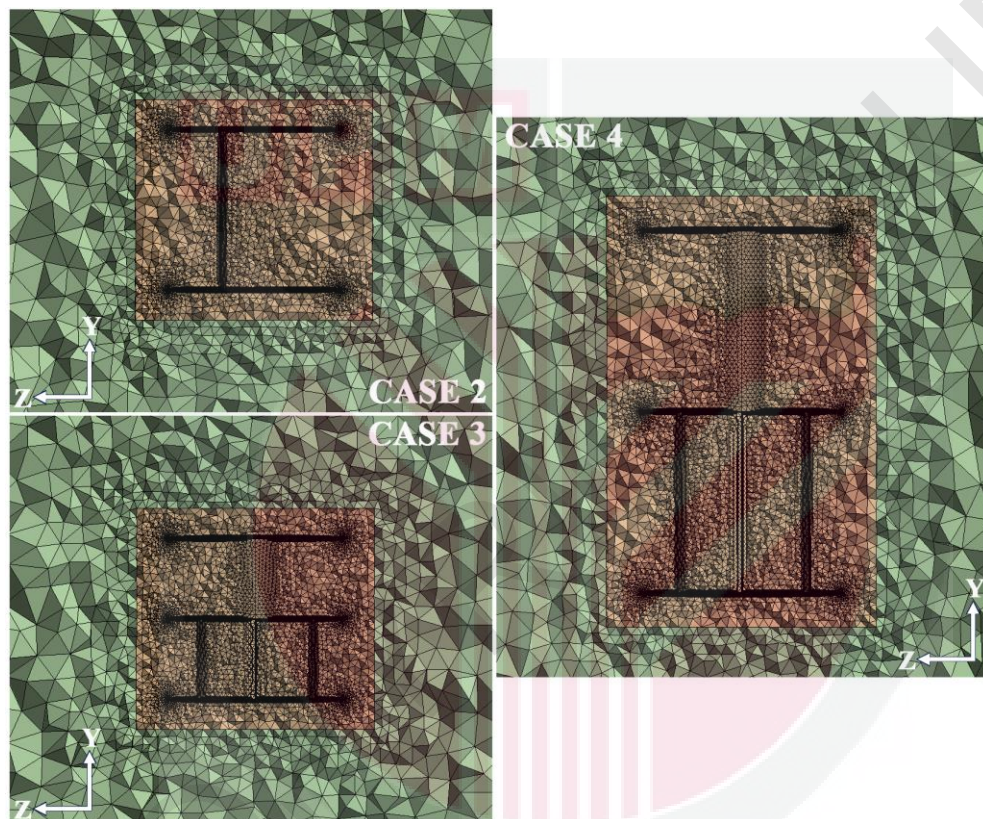


Figure 3.21 : Mesh configuration of 3D SHKT models used in the case 2-4
3.7 CFD Approaches of the Darrieus Hydrokinetic Turbine

This section will present the CFD approaches using ANSYS Fluent. The simulations have been carried out systematically to investigate and optimize new designs of the delta-shaped blade Darrieus hydrokinetic turbine (DHKT) models employing NACA0012 hydrofoils. The 2 and 3-blade models of varying blade characteristics and cross-sectional areas will be simulated. The models will be positioned fully submerged inside a water flow domain and be forced to rotate with different rotational speeds under a constant upstream velocity of 1.5 m/s by utilizing the sliding mesh technique. CFD methodology including setting up methods, and analytical methods will also be explained in this section.

3.7.1 Designs of the 3D Delta-Shaped Blade DHKT Models

To design the DHKT models, several parameters were considered. The blade designs were the main concern in this study. Firstly, blade profiles were considered from the literature on the one that provided the most optimized performances, or has been found lacking. From consideration, a symmetrical NACA0012 profile in which the pivot point was located at a $0.2c$ unit from the frontal edge as shown in Figure 3.22(a) was selected to utilize as a blade profile for all models. Additionally, blade shapes were also considered with a similar concept. Plenty of computational or experimental research works have been carried out using turbines with straight blades, known as H-shaped blades such as in [5],[45],[31]. However, studies on the delta-shaped blade have been found lacking. The delta-shaped blade accordingly was selected.

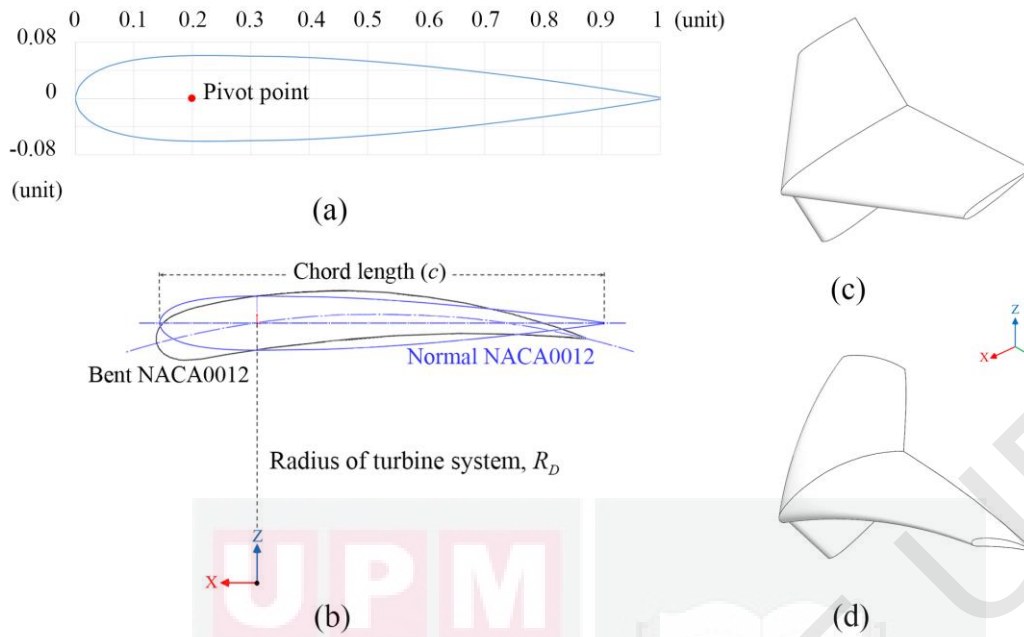


Figure 3.22 : (a) NACA0012 profile, (b) Normal and bent NACA0012 profiles, (c) In-plane, and (d) Twisted delta-shaped blade designs

There were two designs of the delta-shaped blade. The blade created from the normal NACA0012 profile depicted in Figure 3.22(a) was called an in-plane delta-shaped blade. Whereas the blade created from the bent NACA0012 profile arranged along the circular line of the turbine system as shown in Figure 3.22(b) was called a twisted delta-shaped blade. The blade designs of an in-plane and a twisted delta-shaped blade are illustrated in Figure 3.22(c) and (d), respectively.

As depicted in Figure 3.23, the models consisted of three main parts including a central shaft, supporting arms, and blades. The blades were connected to a central shaft by supporting arms. All supporting arms were also shaped as a NACA0012 profile. The shape, and size of the supporting arm as well as the shaft design were kept constant to avoid any effect from those factors. The model relative positions were represented by azimuth angle (θ). Blade shapes were symmetry along the x-z plane. The incline-blade angle (γ) of 60° was fixed.

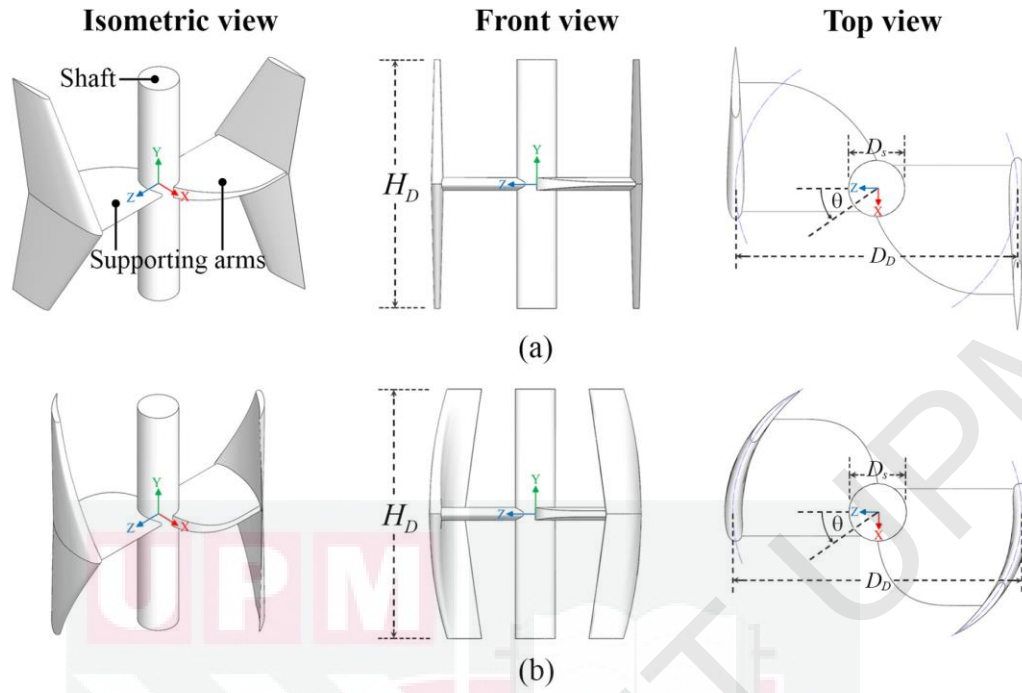


Figure 3.23 : Schematic sketches of the isometric, front, and top views of the 2-blade (a) In-plane, and (b) Twisted delta-shaped blade with reduced cross-sectional areas models

Turbine blades were also modified with two different designs corresponding to the turbine cross-sectional area. First, the blades were composed of two trapezoidal wings which its cross-sectional areas were reduced constantly by the factor of 0.5 from the x-z plane. It meant that the cross-sectional areas at both ends of the wings were half of the area at the x-z plane. Second, the blades are composed of two constant cross-sectional area wings. The combination of those two design parameters, for the 2-blade DHKT models ended with four iterations which were called *MD1-MD4*. As presented in Figure 3.24, the in-plane blade models with reduced and constant cross-sectional areas were called *MD1* and *MD2*, respectively. Whereas *MD3* and *MD4* represented the twisted blade models with reduced and constant cross-sectional areas, respectively.

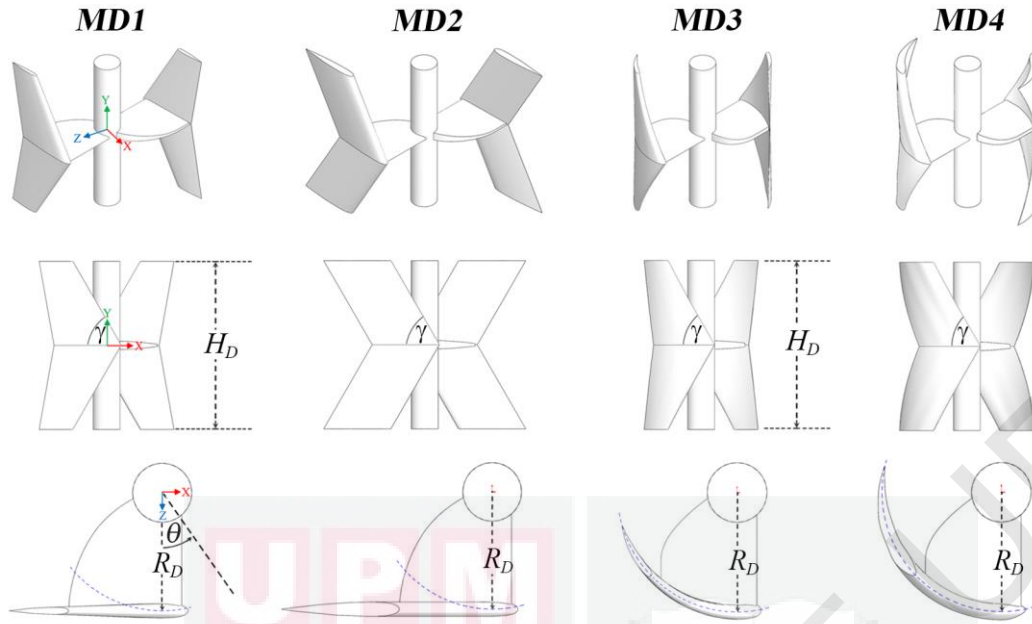


Figure 3.24 : The 2-blade DHKT model designs (MD1-MD4)

To investigate the effect of blade number, the DHKT models were also modified with four designs named MD5-MD8 as depicted in Figure 3.25. The model height (H_D) and diameter (D_D) were 0.15 mm. and 0.12 mm., respectively with the turbine aspect ratio of 1.25. As indicated from the literature, the Darrieus turbine performed better when an aspect ratio is greater than 1 [64]. The incline-blade angle (γ) was kept constant at 60° . The models were made from Acrylonitrile Butadiene Styrene (ABS) with a density of $1,020 \text{ kg/m}^3$. The general geometric parameters of a DHKT model are presented in Table 3.7.

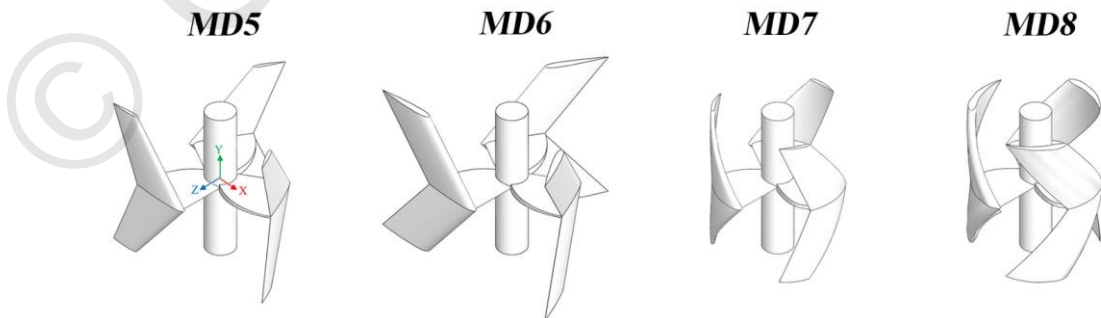


Figure 3.25 : Designs of 3-blade DHKT models

Table 3.7 : Geometric features of DHKT models

Parameters	Results
Blade profile	NACA0012
Characteristic of blade	Delta-shaped blades
Number of blades, n	2, 3
Incline-blade angle (γ)	60°
Chord length at $Y = 0$	0.10 m.
Chord length at $Y = 0.75$	0.05 m.
Height (H_D)	0.15 m.
Diameter (D_D)	0.12 m.
Shaft diameter (D_s)	0.024 m.
Turbine aspect ratio (H_D/D_D)	1.25
Material	Acrylonitrile Butadiene Styrene (ABS)
Density of material	1,020 kg/m ³

To optimize the model designs, CFD simulations using the ANSYS Fluent commercial software were conducted. There were two main parts. Each part follows the general functional form as presented in Figure 3.26.

Turbine performances including coefficient of power (c_p), and torque (c_t) were the dependent variables (DV) while the tip speed ratio (λ) was an independent variable (IV). For the first part, four different designs of 2-blade models ($MD1 - MD4$) were studied. The model designs were the variable parameters (VP) and other parameters such as number of blades (n), incline-blade angle (γ), and upstream water velocity (U_∞) were kept as constant parameters (CP).

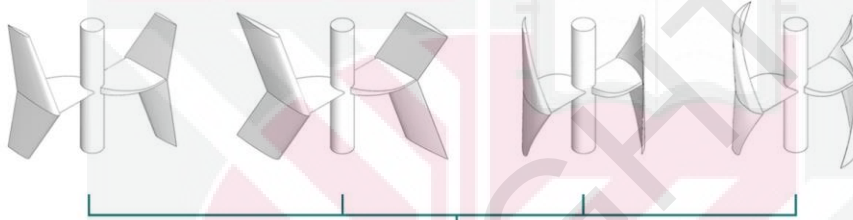
After getting the optimum designs from the 2-blade design, CFD simulations were conducted to investigate the effect of the number of blades (n). The results of the optimized 2-blade designs from the previous part were compared with those of 3-blade designs ($n = 3$).

General functional form

$$DV = f(IV; VP; CP)$$

Part 1; Test on the model designs (MD)

$$c_p, c_t = f(\lambda; MD1, MD2, MD3, MD4; n = 2, \gamma = 60^\circ, U_\infty)$$



Part 2; Test on the number of blades (n)

$$c_p, c_t = f(\lambda; n = 2, n = 3; MD, \gamma = 60^\circ, U_\infty)$$

$n = 2$

$n = 3$

* DV = Dependent variable

VP = Variable parameters

IV = Independent variable

CP = Constant parameters

Figure 3.26 : Optimization procedures of the Darrieus hydrokinetic turbine

3.7.2 Computational Methodology

3.7.2.1 Turbulence Model

In general, there is no agreement on the most appropriate turbulence model adopted in CFD simulations. However, according to previous studies of the Darrieus turbine, the Shear Stress Transport (SST) $k-\omega$ turbulence model which was a hybrid model consisting of the $k-\varepsilon$ and $k-\omega$ combination has abilities to capture flow structures in both the boundary layer as well as the free stream regions [31],[158]. It has been widely used in the simulations of vertical axis turbines and showed good agreement with experimental measurements [31],[123],[159],[160]. Hence, the SST $k-\omega$ turbulence model was utilized for all simulations as an acceptable model to consider the performances and flow characteristics of the DHKT models. The transport equation for the SST $k-\omega$ can be written as equation (Eq. 3.9) and (Eq. 3.10).

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k - Y_k \quad (\text{Eq. 3.9})$$

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_j}(\rho \omega u_j) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + G_\omega - Y_\omega \quad (\text{Eq. 3.10})$$

3.7.2.2 Computational Domain

A computational domain depicted in Figure 3.27(a) used for all simulation cases was created as a rectangular cube. The domain dimensions were represented in the form of multiples of the model diameter (D_D) which were $10D_D$, $10D_D$, and $23D_D$ in x-, y-, and z- directions, respectively. The DHKT model was positioned at the origin of a Cartesian coordinate system (0, 0, 0). It was $5D_D$ downward from the inlet boundary. The model's initial position where $\theta = 0^\circ$ is depicted in Figure 3.27(b). The domain comprised two main regions which were a stationary domain and a rotating domain surrounding a DHKT model. Both regions were separated by a rotating cylindrical surface called an interface boundary where the continuity of absolute velocity was set to ensure the continuity of the flow field and the rotating region was allowed to rotate freely with different rotational speeds whereas the rest was kept as fixed [19],[23], [54],[55]. As seen in Figure 3.27(c) and (d), the model was located inside and at the center of a rotating cylinder which has the diameter and height of $1.33D_D$ and $1.3D_D$, respectively.

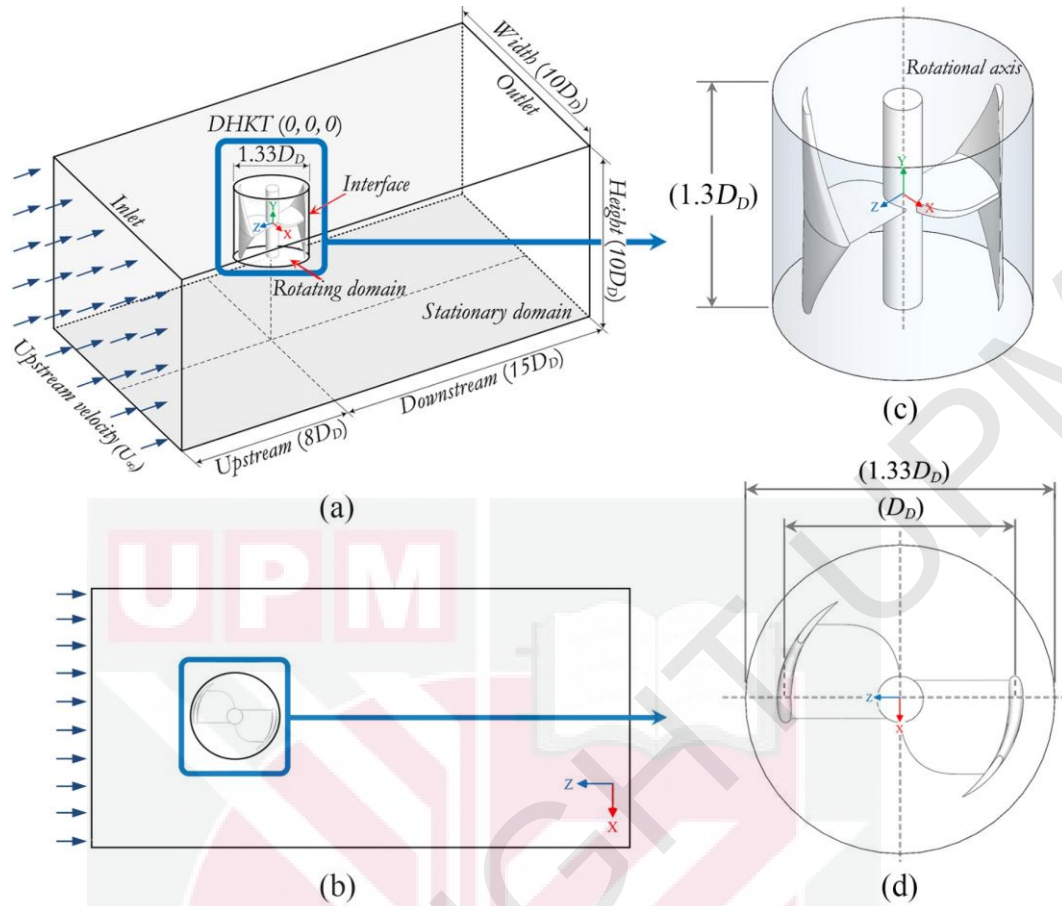


Figure 3.27 : Different views and geometrical parameters of (a) a rectangular computational domain, (b) a top view of a domain, (c) an isometric view of the rotating zone, and (d) a top view of the rotating zone

3.7.2.3 Boundary and Initial Conditions

The sliding mesh technique was employed for all simulations. For this technique, a model was forced to rotate constantly in the positive direction with different rotational speeds (ω_b) from 10 to 37.5 rad/s, under a constant upstream velocity (U_∞) of 1.5 m/s. The outlet total pressure was set as zero. The turbulence intensity (I) of 0.1% and turbulent viscosity ratio of 10 were imposed. For all surfaces of a model and flow channel, the standard wall roughness and non-slip condition were assumed. The boundary and initial conditions used for simulations of the DHKT models are represented in Table 3.8.

Table 3.8 : The boundary and initial conditions for the 3D DHKT model simulations

Location	Boundary and initial conditions	Condition	Value
Inlet	Velocity inlet	Uniform water flow, Normal to the boundary	1.5 m/s
Outlet	Pressure outlet	Total pressure	0 (gauge pressure)
Domain channel	No slip walls Stationary wall	Standard wall roughness	0.5
DHKT surfaces	No slip walls Stationary wall	Standard wall roughness Rotate at the setting speed	0.5
Flow condition	Turbulent intensity (I)		0.1%
	Turbulent viscosity ratio		10
Interface	The link between the rotating and stationary region		Coupled surface

For the DHKT simulations, the convergence criteria were imposed to be 10^{-5} for the residual of all equations including continuity and momentum equations. The maximum iterations were set as 100 per time step. The simulations were run until the results were converged. To consider the convergence, the repeated average power coefficient ($c_{p, avg}$) of each revolution was calculated and determined. The solver setting for all simulations of the designed DHKT model is presented in Table 3.9.

Table 3.9 : The solver setting for simulations of SDHKT models

Parameters	Values
Turbulence model	SST $k-\omega$
Pressure–velocity coupling	
-Scheme	SIMPLE
Spatial Discretized	
-Pressure equation,	Second order
-Momentum equation,	upwind
-Turbulent kinetic energy k equation,	
-Specific dissipation rate ω equation,	
-Transient formulation	
Maximum iterations per time step	100
Convergence criterion	$\varepsilon < 1\%$
-Continuity equation and the rest equation	10^{-5}

3.7.3 Mesh Configurations and Independency Study

Mesh was generated by using the ANSYS-Mesh module. As mentioned, there were two different regions of a flow domain. In practice, mesh generation with the unstructured tetrahedral elements was set with different densities in each region. In the rotating region, finer meshes compared to the stationary region were created. To capture velocity and pressure gradients, inflation layers were applied to model surfaces. To achieve a maximum y^+ value less than 5 over the near walls of a model, the first layer cell thickness of 0.5 mm was applied using a growth rate of 1.1 and total layers of 7.

The quality of the mesh influenced significantly the efficiency, accuracy, and precision of CFD simulation [31]. To optimize mesh size or mesh resolution, an independent study was conducted with 7 different mesh sizes which were varied from both regions. The mesh configuration details are presented in Table 3.10. The simulations were carried out until the results represented by the average c_p had already reached the convergence criterion as mentioned in equation (Eq. 3.4). The simulations of the mesh independency study were conducted by using the *MD3* model together with the sliding mesh method. The model was forced to rotate with $\omega_b = 37.5$ rad/s under a constant U_∞ of 1.5 m/s ($\lambda = 1.5$).

Table 3.10 : Setting of the mesh independency study

Mesh Parameters		M1	M2	M3	M4	M5	M6	M7
Element size (mm.)	Inner (MI)	50	50	30	30	30	20	15
	Outer (MO)	60	50	50	40	30	30	30
Number of Nods	Inner	185,549	185,481	185,721	185,548	185,191	185,492	185,524
	Total	225,522	253,094	253,334	316,075	489,978	490,279	490,311
Number of Elements	Inner	678,570	677,767	678,803	678,103	677,311	677,333	678,414
	Total	899,295	1,056,926	1,057,962	1,418,068	2,427,216	2,427,238	2,428,319
Orthogonal Quality	Maximum	0.9951	0.99561	0.99561	0.99606	0.99689	0.99689	0.99689
	average	0.74142	0.74924	0.74934	0.76024	0.77408	0.77404	0.7742
Aspect Ratio	Maximum	207.56	207.64	207.99	207.58	207.55	205.12	206.5
	average	3.638	3.363	3.3589	2.9653	2.476	2.4776	2.4768
Skewness	Minimum	1.26E-06	1.34E-06	1.74E-06	2.94E-07	6.16E-07	6.70E-07	3.01E-07
	Maximum	0.97281	0.97281	0.97281	0.97281	0.97281	0.97281	0.97281
	average	0.25702	0.24924	0.24915	0.23831	0.22455	0.22459	0.22443

Variations of instantaneous c_p with flow time illustrated in Figure 3.28(a) from all configurations were dramatically close. However, there were some differences, especially at the peaks. The peaks of M1 were a little bit higher than the rest. Besides, the results of M1-M7 showed that the average c_p of the 8th revolution already reached the convergence criterion. As seen in Figure 3.28(b), it was also found that the decreasing mesh size in the rotating region (MI) between M2 and M3, or M5 to M7 showed a smaller effect while a change of size in the stationary region (MO) from M3 to M5 showed a huge effect on the predicted results. In addition, the average c_p of M1 to M6 was compared with M7 (finest mesh resolution) as a reference value. It was found that the percentage differences (δ) of M4 - M6 were still close ($\delta < 1\%$) to M7. However, the average c_p of M3 was over the criterion value of 1%. Therefore, the M4 configuration was selected to be employed for all simulations. The mesh topology of the M4 configuration at different views using a 2-blade DHKT model (MD3) was presented in Figure 3.29.

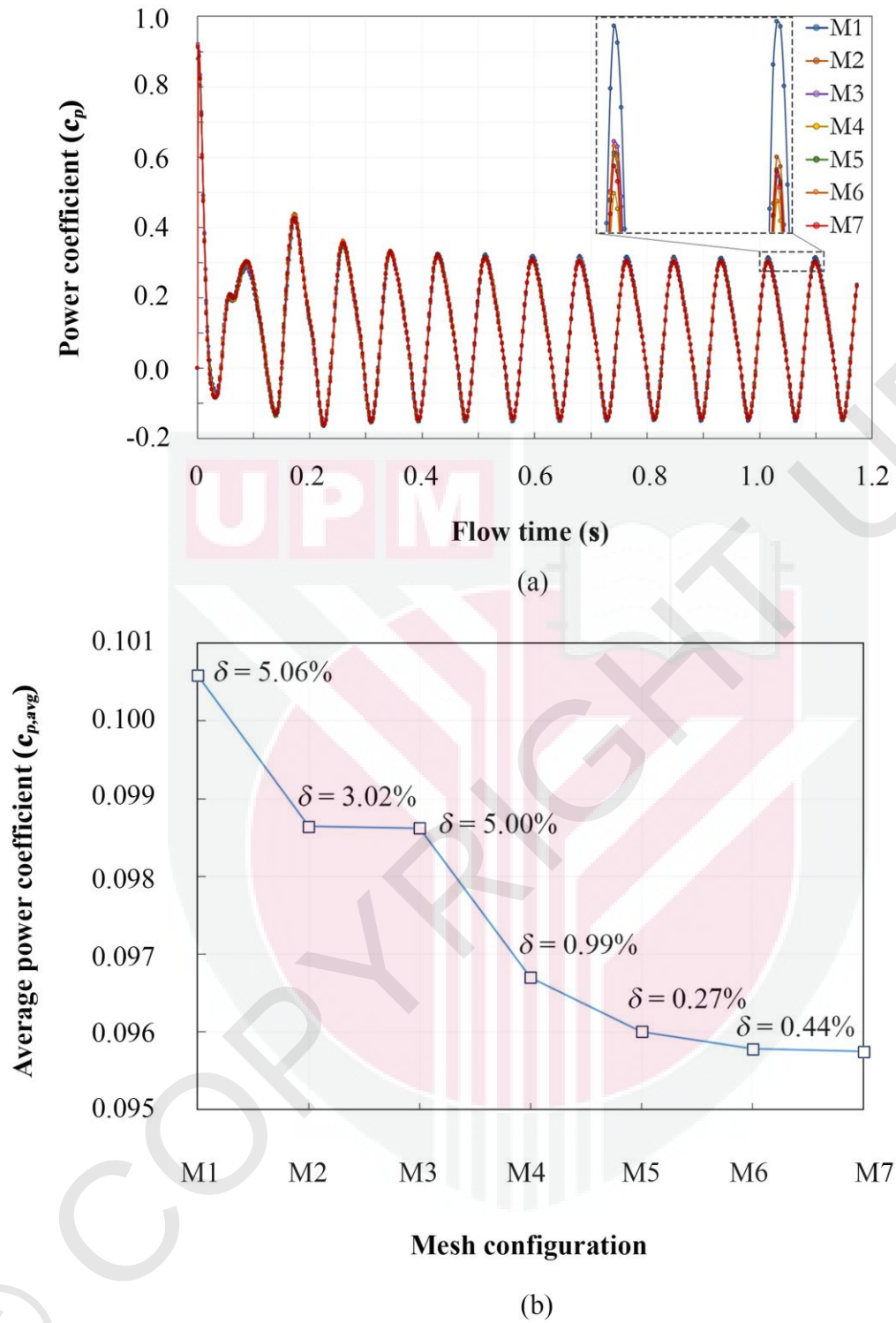


Figure 3.28 : (a) Variation of instantaneous c_p with flow time of several mesh configurations (M1-M7), and (b) The average c_p of the 8th revolution of each mesh configuration and its percentage differences (δ)

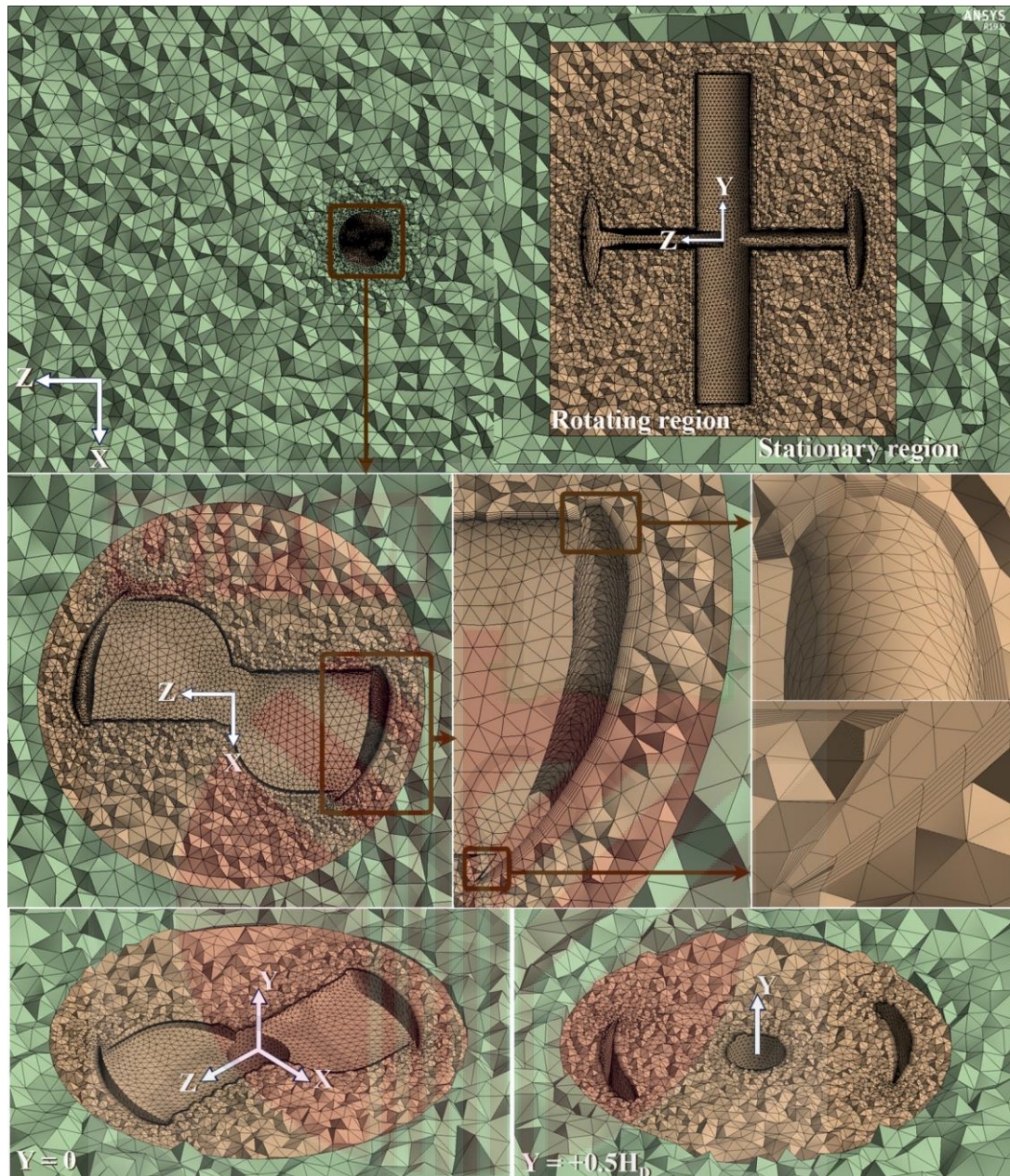


Figure 3.29 : Mesh configuration, M4 (MI = 30 mm., MO = 40 mm.) in simulations of the DHKT models

3.8 CFD Method of the Combined SDHKT Models

This section is going to present the simulation method on the three-dimensional (3D) combined Savonius-Darrieus hydrokinetic turbine (SDHKT) utilizing CFD ANSYS fluent simulations. The simulation method to optimize the designed Savonius and Darrieus hydrokinetic turbine models has already been presented in the previous sections. The optimized models of those were combined becoming the combined SDHKT models. Many parameters that influenced the performances of the combined SDHKT need to be considered. To study the effect of attachment angles (ϕ), and ratio of swept areas (R_{sa}), systematic optimization processes will be carried out. To begin with, model designs and configurations of those optimized SHKT, and DHKT models will be explained in the next section. CFD methodology including setting up methods, and analytical methods, are also presented in this section.

3.8.1 Model Designs and Configurations of a Combined SDHKT

Model designs and configurations of combined SDHKT models were considered by combining each optimized model including a Savonius hydrokinetic turbine (SHKT) model and Darrieus hydrokinetic turbine (DHKT) models investigated in the previous chapter. Based on the simulation results in chapter 4, it was found that a 2-stage SHKT model with a 45° attachment angle between each stage, and in chapter 4, the 2-blade DHKT models, *MD3* and *MD4* provide better results compared to the rest designs. Hence, those SHKT and DHKT models were selected to be a part of the combined SDHKT models to get benefits from the advantages of each, and to be able to operate in the wider range of tip speed ratio.

Based on the previous study of Wakui et al., 2005 [150], a Savonius turbine placed inside the middle of a Darrieus turbine sharing the same shaft was the most suitable configuration [150]. For this study, a 2-stage SHKT model therefore was placed inside a DHKT model on the same shaft as shown in Figure 3.30.

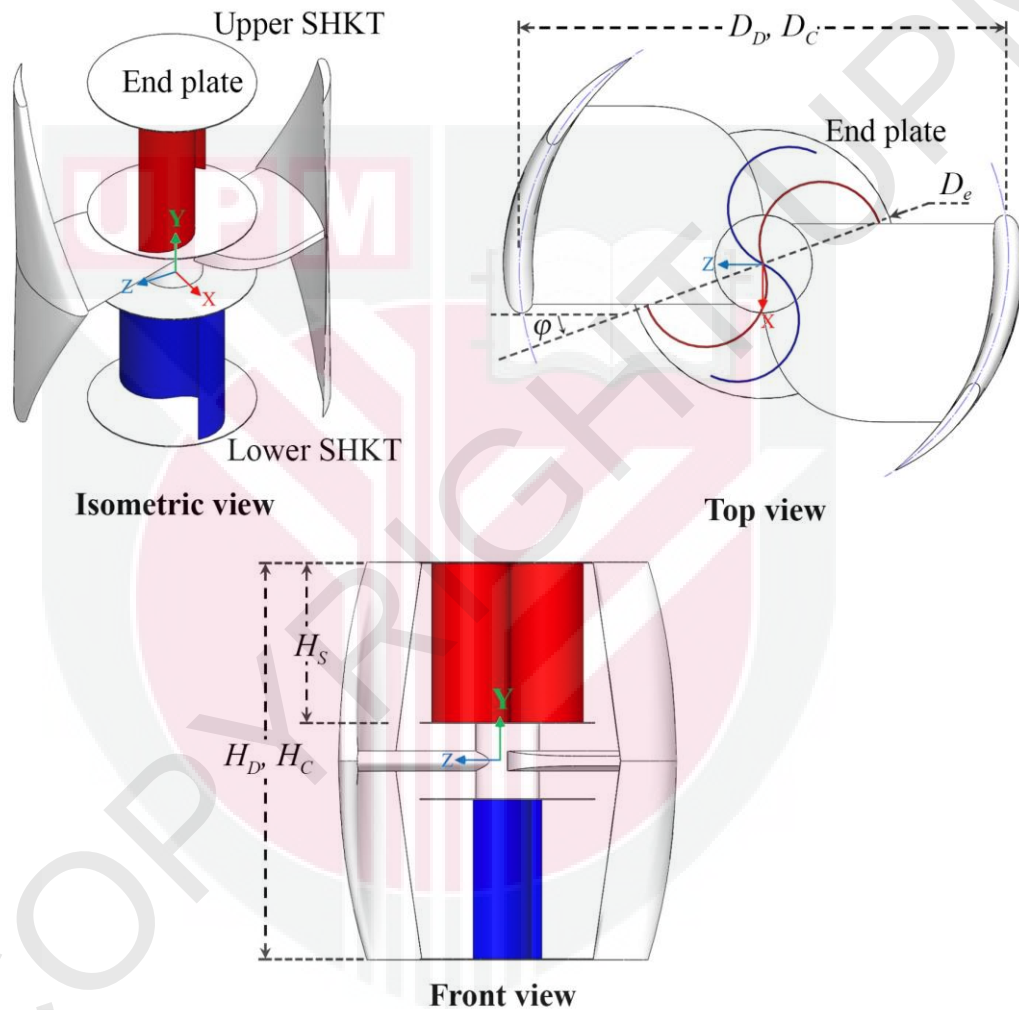


Figure 3.30 : Design of the combined SDHKT model

To get the optimum design and configuration of a combined SDHKT model, some parameters such as attachment angles (ϕ), and ratios of swept area (R_{sa}) were considered. For each design configuration, 3D SDHKT models with several factors were modeled by using the SolidWorks software. To study the effect of attachment angles (ϕ) defined as an angle between the bucket of an upper SHKT model and the blade of a DHKT model, five different angles including 0° (AAD0), 20° (AAD20), 30° (AAD30), 90° (AAD90), and 120° (AAD120) illustrated in Figure 3.31 were modeled. An angle difference between the upper and lower buckets of SHKT models was kept constant at 45° which was the best value from the study of Saad et al., 2020 [36].

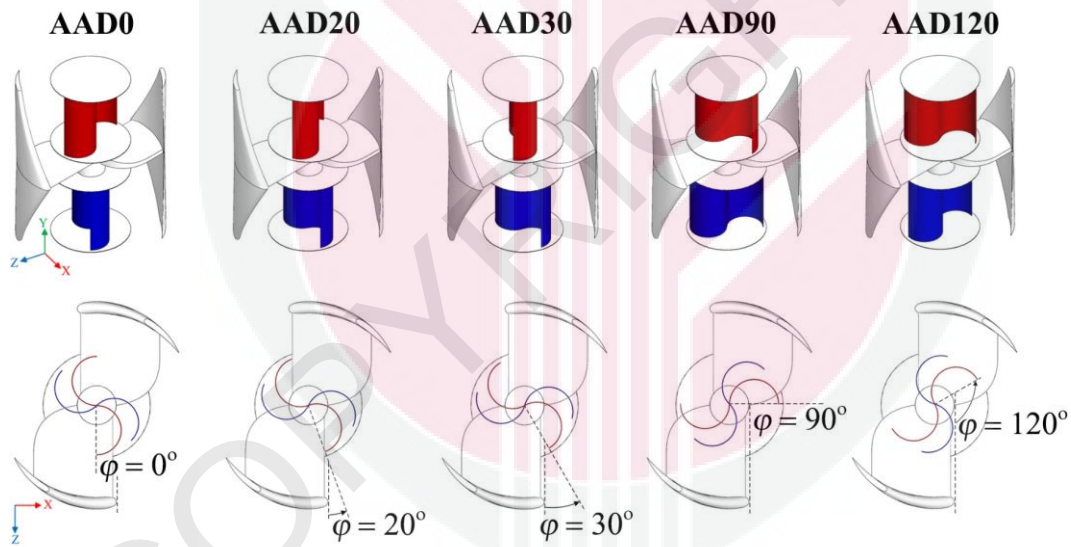


Figure 3.31 : Combined SDHKT models with several attachment angles (ϕ)

The main features of a combined SDHKT model in this study are presented in Table 3.11. The position and dimensions of SHKT and DHKT models were held constant while the bucket orientations of SHKT models were varied to obtain different attachment angles (φ).

Table 3.11 : Main features of a combined SDHKT model

Parameters	Results
Airfoil blade profile	NACA0012
Characteristic of DHKT blade	Delta-shaped blades
Models of DHKT	MD3, MD4
Chord length (c) at $Y = 0$	0.10 mm.
Number of blades of DHKT	2
Number of stages of SHKT models	2
Characteristic of SHKT buckets	2 Semi-circular buckets
Height of SHKT (H_S)	0.06 m.
Aspect ratio (H_S / D_S) of SHKT	1
Angle different between each stage of SHKT models	45°
Height of SDHKT models (H_C)	0.15 m.
Diameter of SDHKT models (D_C)	0.12 m.
Aspect ratio (H_C / D_C) of SDHKT	1.25
Attachment angles (φ)	0°, 20°, 30°, 90°, 120°
Shaft diameter (D_S)	0.024 m.
Material	Acrylonitrile Butadiene Styrene (ABS)
Density of material	1,020 kg/m ³

Next, the combined SDHKT model was modified once again in order to study the effect of the ratios of swept areas (R_{sa}) defined as a ratio of the swept areas of a 2-stage SHKT model (A_{sS}) to the swept area of a DHKT model (A_{sD}). For this study, dimensions including the diameter (D_D) and height (H_D) of a DHKT model were held constant whereas the diameter (D_s) and height (H_s) of each stage of SHKT models were varied to get different ratios of swept areas. Therefore, there were three different values of R_{sa} including $R_{sa1} = 0.4$, $R_{sa2} = 0.3$, and $R_{sa3} = 0.2$. Furthermore, an attachment angle (ϕ) equaled zero, aspect ratios of each model, as well as other parameters that influenced turbine performances, were kept constant to avoid any effect from those parameters. Figure 3.32 and Table 3.12 illustrate the model designs and details of model configurations.

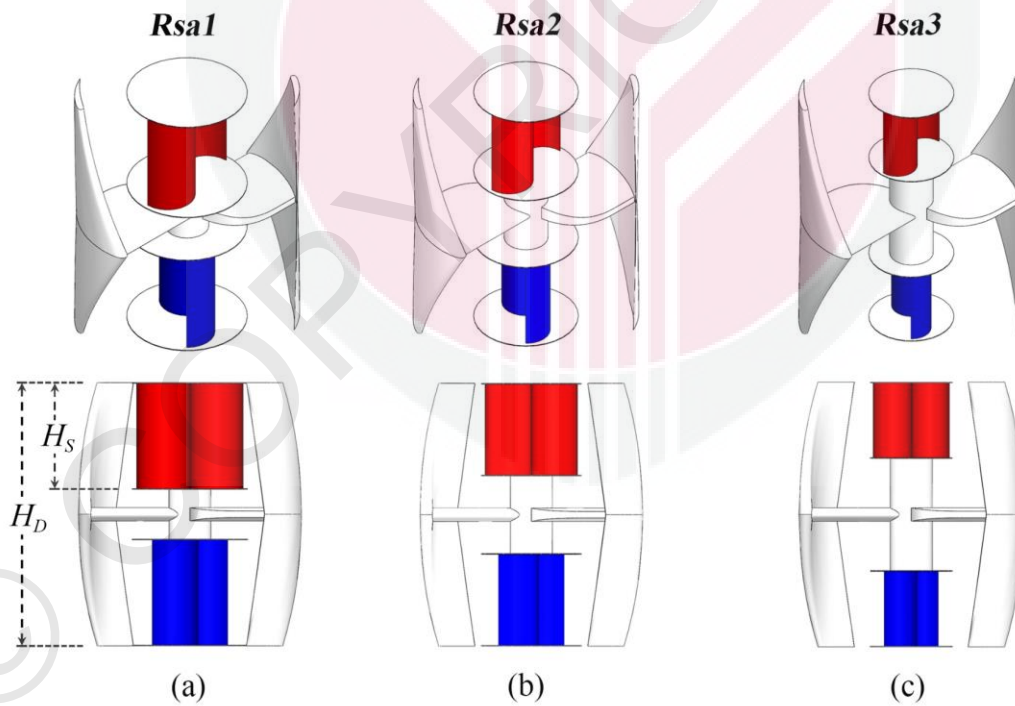


Figure 3.32 : Combined SDHKT models with three different values of R_{sa}

Table 3.12 : Design parameters for the models with various ratios of swept area

Parameters	R_{sa1}	R_{sa2}	R_{sa3}
Height of SHKT (H_S)	0.06	0.052	0.0424
Diameter of SHKT (D_S)	0.06	0.052	0.0424
Aspect ratio of SHKT (AR) = H_S/D_S	1	1	1
Swept area of 1 SHKT (A_{sS}) = $H_S \times D_S$	0.0036	0.0027	0.0018
Swept area of 2 SHKT (A_{sS})	0.0072	0.0054	0.0036
Height of DHKT (H_D)		0.15	
Diameter of DHKT (D_D)		0.12	
Aspect ratio of DHKT (AR) = H_D/D_D		1.25	
Swept area of DHKT (A_{sD}) = $H_D \times D_D$		0.018	
Ratio of swept area (R_{sa}) = A_{sS}/A_{sD}	0.4	0.3	0.2

3.8.2 Computational Methodology

3.8.2.1 Turbulence Model

Generally, there is no agreement on the most appropriate turbulence model adopted in CFD simulations. Nevertheless, according to previous studies on the combined Savonius-Darrieus turbine, the SST $k-\omega$ turbulence model which was a hybrid model consisting of the $k-\varepsilon$ and $k-\omega$ combination had abilities to capture flow structures in the boundary layer as well as the free stream regions [31]. It has been widely used in CFD

simulations of vertical axis turbines and showed good agreement with experimental measurements [31],[159]. Hence, the SST $k-\omega$ turbulence model was utilized for all simulations as an acceptable model to consider the turbine performances and flow characteristics. The transport equations for the SST $k-\omega$ were written as equation (Eq. 3.11) and (Eq. 3.12).

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k - Y_k \quad (\text{Eq. 3.11})$$

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_j}(\rho \omega u_j) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + G_\omega - Y_\omega \quad (\text{Eq. 3.12})$$

3.8.2.2 Computational Domain

A computational domain of the combined SDHKT model used for all simulations was created as a rectangular cube. The domain dimensions as depicted in Figure 3.33(a) were represented in the form of multiples of the model diameter (D_C) which were $10D_C$, $10D_C$, and $25D_C$ in X, Y, and Z directions, respectively. The SDHKT model was located at the origin of a Cartesian coordinate system (0, 0, 0) which was downward $5D_C$ from the inlet boundary. The model initial position or an azimuth angle (θ) where $\theta = 0^\circ$ is depicted in Figure 3.33(b).

The domain comprised two main regions which were a stationary domain and a rotating domain surrounded a combined SDHKT model. Both regions were separated by a rotating cylindrical surface called as an interface boundary where the continuity of absolute velocity was imposed to ensure the continuity of the flow field and the rotating region was allowed to rotate freely with different rotational speeds whereas the rest was kept as fixed [19],[23],[53],[54]. As seen in Figure 3.33(c) and (d), the model was located inside and at the center of a rotating cylinder which has the diameter and height of $1.25D_c$ and $1.5D_c$, respectively.

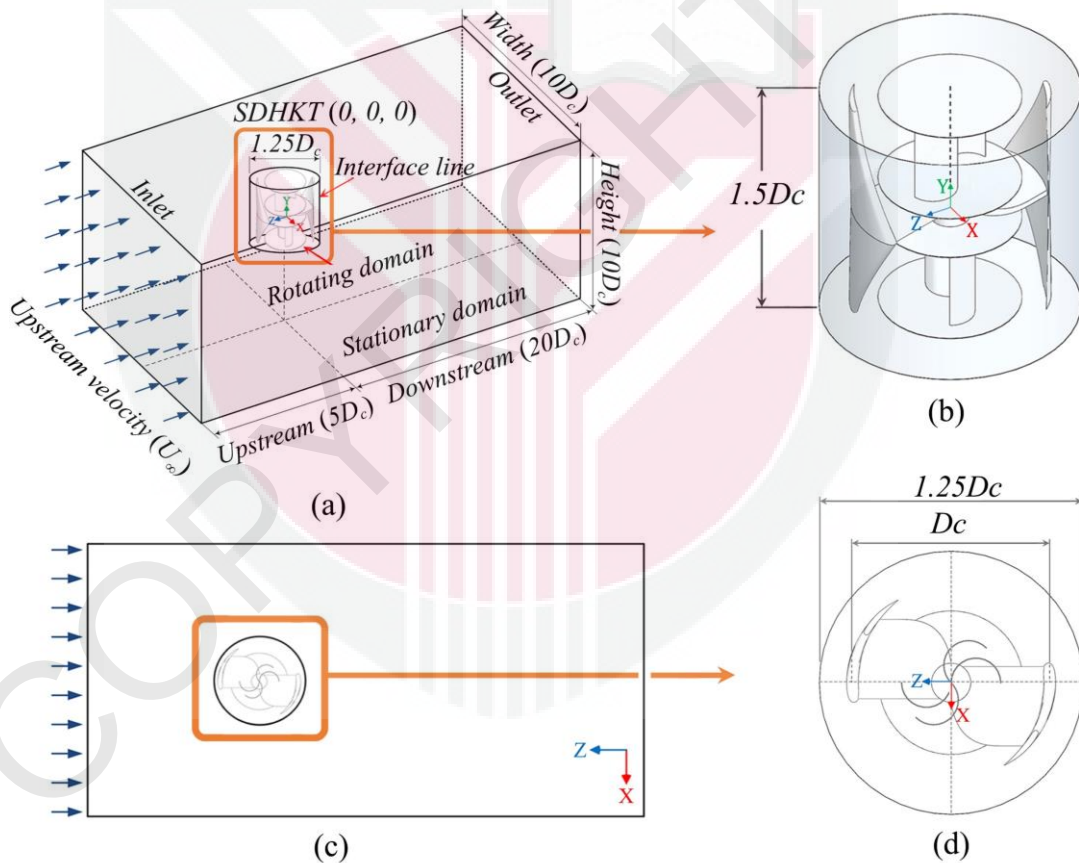


Figure 3.33 : Different views and geometric parameters of (a) the computational domain, (b) the top view, (c) the isometric view of the rotating zone, and (d) the top view of the rotating zone

3.8.2.3 Boundary and Initial Conditions

The sliding mesh technique was used in all SDHKT model simulations. For the sliding mesh simulation, a model was forced to rotate constantly in the positive direction with 6 different rotational speeds (ω_b) under a constant upstream velocity (U_∞) of 1.5 m/s. The outlet total pressure was set as zero. The turbulence intensity (I) of 0.1% and turbulent viscosity ratio of 10 were imposed. For all surfaces of a model and flow channel, the standard wall roughness and non-slip condition were assumed. The boundary and initial conditions using in the 3D simulations of the combined SDHKT models are represented in Table 3.13 and the summary of the current solver setting for all simulations of the designed combined SDHKT model was presented in Table 3.14.

Table 3.13 : The boundary and initial conditions for 3D simulations of combined SDHKT models

Location	Boundary and initial conditions	Condition	Value
Inlet	Velocity inlet	Uniform water flow, Normal to the boundary	1.5 m/s
Outlet	Pressure outlet	Total pressure	0 (gauge pressure)
Domain channel	No slip walls Stationary wall	Standard wall roughness	0.5
Model surfaces	No slip walls Stationary wall	Standard wall roughness Rotate at the setting speed	0.5
Flow condition	Turbulent intensity (I)		0.1%
	Turbulent viscosity ratio		10
Interface	The link of the rotating and stationary region		Coupled surface

Table 3.14 : The solver setting for simulations of SDHKT models

Parameters	Values
Turbulence model	SST $k-\omega$
Pressure-velocity coupling -Scheme	SIMPLE
Spatial Discretized -Pressure equation, -Momentum equation, -Turbulent kinetic energy k equation, -Specific dissipation rate ω equation, -Transient formulation	Second order upwind
Maximum iterations per time step	100
Convergence criterion -Continuity equation and the rest equation	$\varepsilon < 1\%$ 10^{-5}

3.8.2.4 Meshing Configuration

In CFD simulations of the combined SDHKT models, mesh size, and configurations were set as similar to the ones used in the simulations of DHKT mentioned in section 3.7.2. Meshes were generated by using the ANSYS-Mesh module. Because of two different regions, stationary and rotating regions of a flow domain. Hence, mesh generation with the unstructured tetrahedral elements was set with different resolutions in each region. Finer meshes compared to the stationary region were created in the rotating region. To capture pressure and velocity gradients, inflation layers were applied to the solid walls of all combined SDHKT models with a maximum thickness of 3.0 mm, a growth rate of 1.1, and a total of 10 layers.

Theoretically, the quality of mesh influences significantly the efficiency, accuracy, and precision in the CFD simulation [31]. Therefore, of mesh size optimization is required. The independent mesh, M4 (MI = 30mm., and MO = 40mm.) as depicted in Figure 3.34 that was investigated and presented in the Section 3.7.3 was utilized once again for all simulations of combined SDHKT models. The details of mesh configurations are presented in Table 3.15.

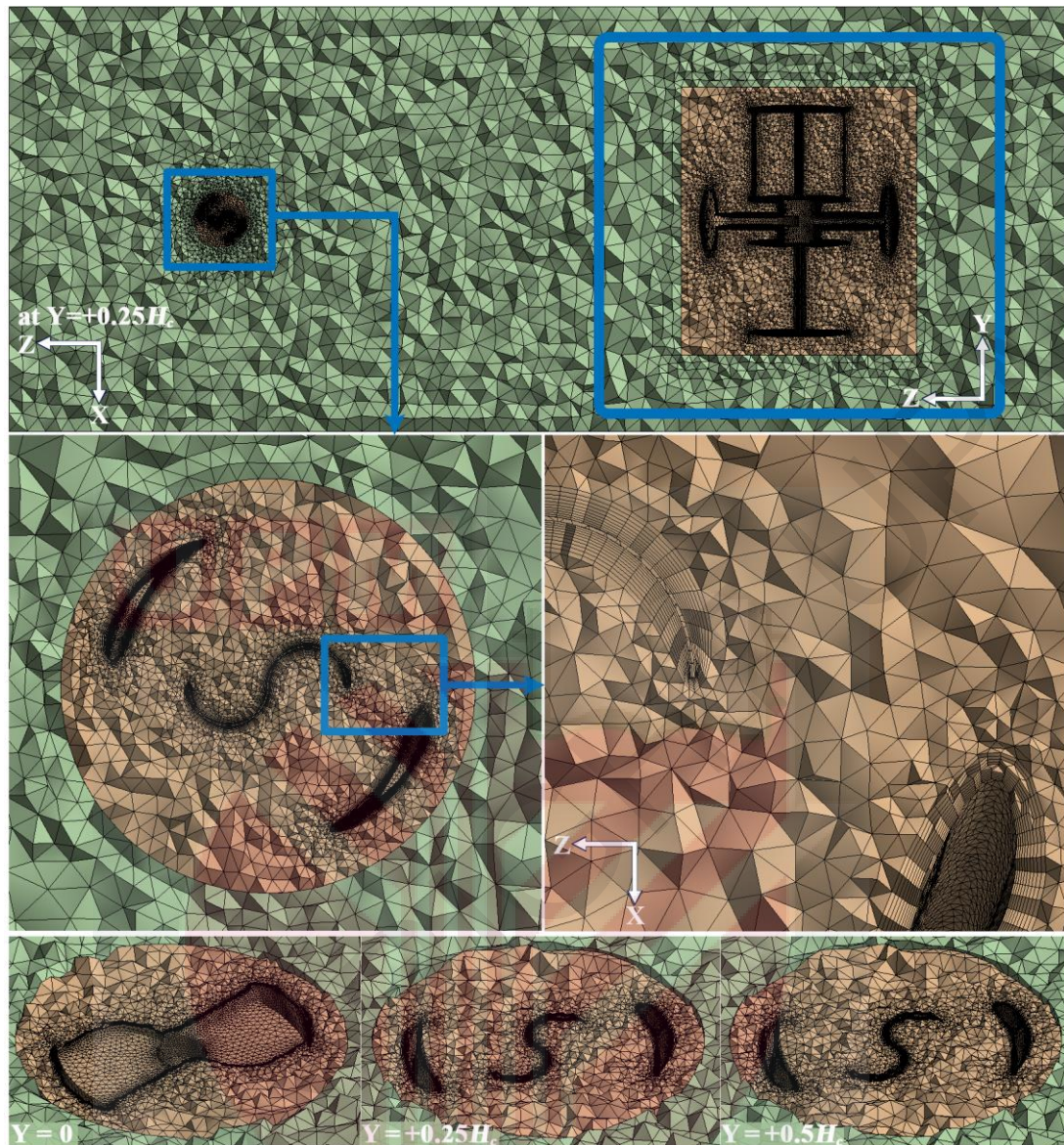


Figure 3.34 : Mesh configuration, M4 (MI = 30 mm., MO = 40 mm.) using in the simulations of the combined SDHKT models

Table 3.15 : Mesh detail and configurations

Mesh configurations	Values
<i>-Element size</i>	
; Rotating region	30 mm.
; Stationary region	40 mm.
<i>-Number of elements</i>	
; Rotating region	1.49 million cells
; Stationary region	0.69 million cells
; Total element	2.18 million cells
<i>-Inflation</i>	
Maximum thickness	3.0 mm.
Total layers	10
Growth rate	1.1

CHAPTER 4

RESULTS AND DISCUSSION

In the previous Chapter, CFD methodology including solver setting up methods, and analytical methods were described intensively. In this chapter, the CFD simulation results utilizing ANSYS fluent commercial software to investigate some geometric parameters that influenced the performances of the SHKT, DHKT, and combined SDHKT models will be presented. Validation of predicted results from this study was also carried out with previous experimental and numerical studies.

4.1 Predicted Simulation Results of SHKT Models

4.1.1 2D Simulations

Simulation results from SHKT models with 3 different central shaft designs in this study were compared with some previous studies including Saad et al., 2020 [36], M. A. Kamoji et al., 2009 [84], M A Kamoji et al., 2009 [85], Menet et al., 2004 [92] and torque was used to compare with the results of Yaakob et al., 2010 [91] . As shown in Figure 4.1, the variations of torque coefficient with overlap ratios were plotted. It found that the c_t from the current study was in the range of 0.1 to 0.4 which was not much different from the previous studies.

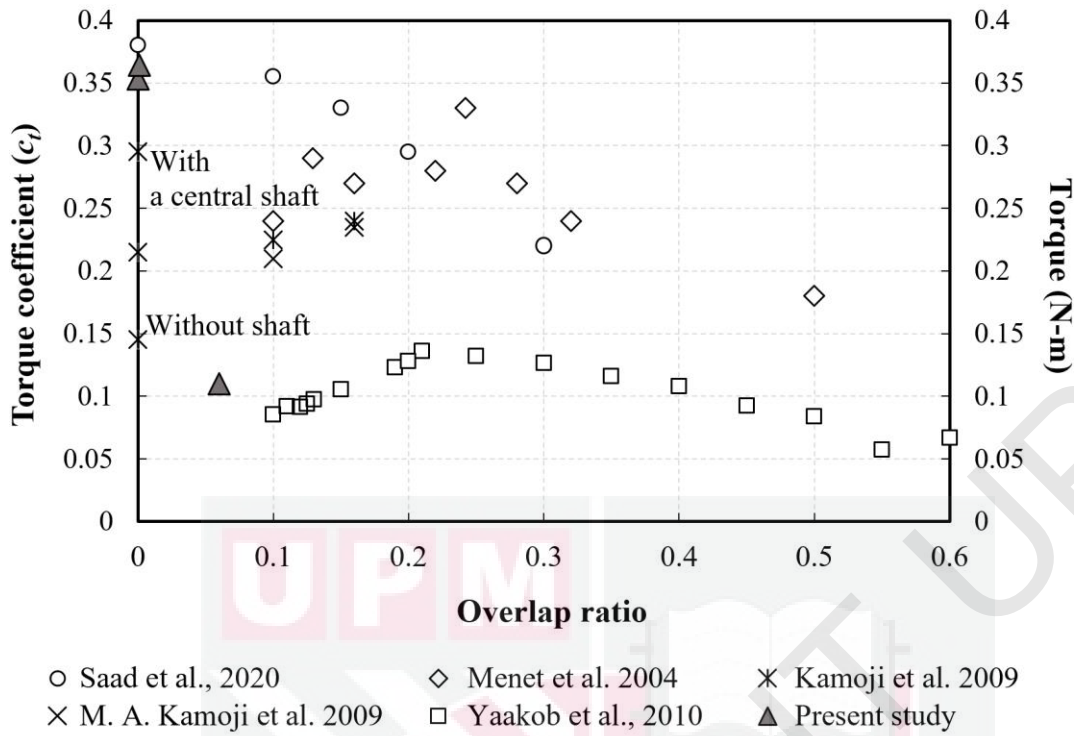


Figure 4.1 : Comparison of the results from this study with published literature
4.1.1.1 Effects of the Central Shaft

The starting torque ability represented by instantaneous torques in one revolution with four different U_∞ including 0.3, 0.5, 0.8, and 1.1 m/s is displayed in Figure 4.2. It could be observed that the model without a central shaft provided the lowest torque compared to the rests in every U_∞ . The instantaneous torques of designs I and II were almost similar. Besides, the amplitude of instantaneous torques also increases when U_∞ increases. Beyond $U_\infty = 0.8$ m/s, the amplitudes and average torque decreased. In addition, fluctuation or turbine vibration can be observed.

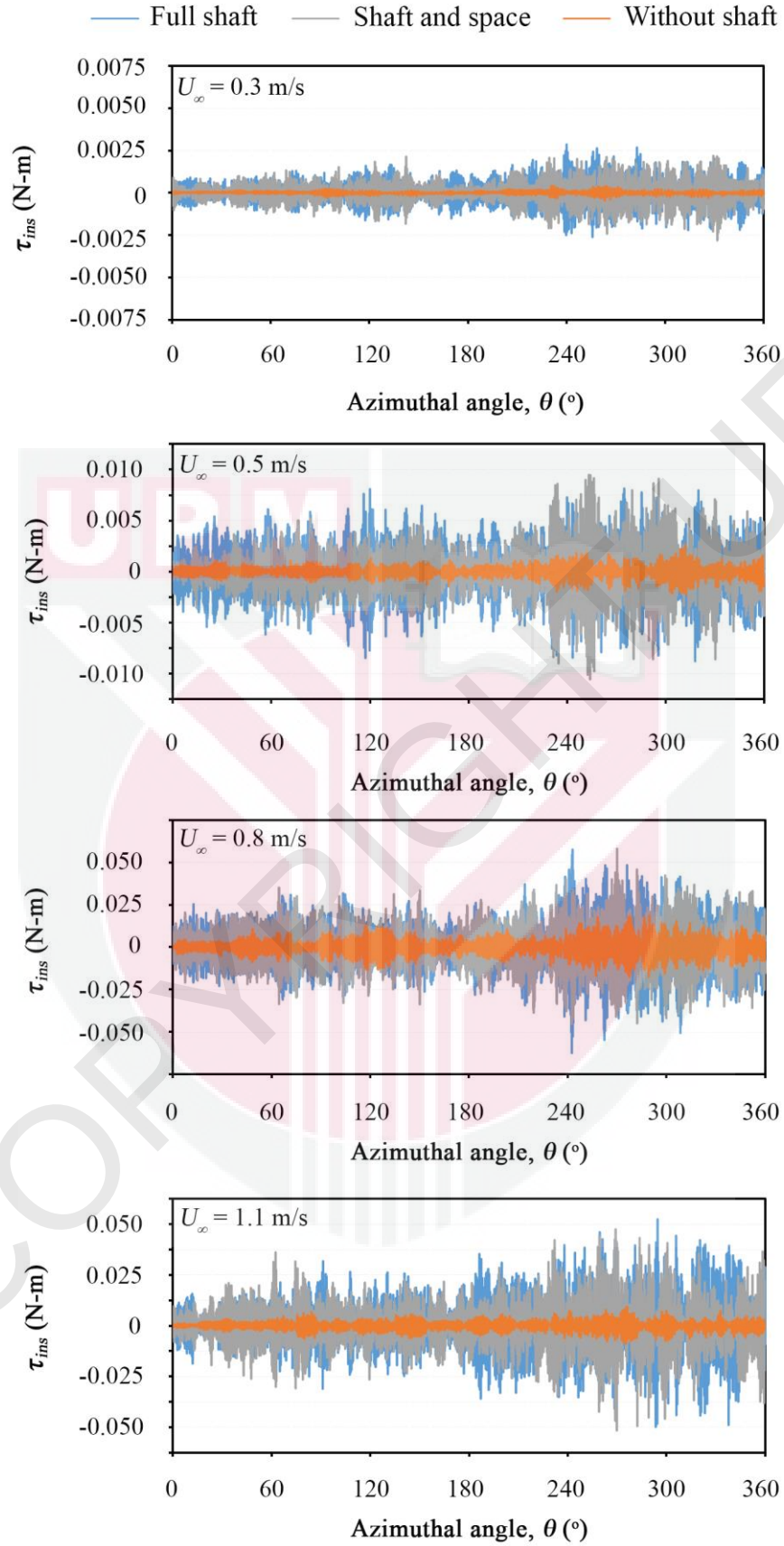


Figure 4.2 : Instantaneous torque with different shaft designs and upstream velocities U_{∞}

The variations of maximum power output ($P_{m,max}$) as a function of the upstream velocity of all designs were plotted as depicted in Figure 4.3. It showed that, when the upstream velocity was increased, $P_{m,max}$ was also increased. Additionally, $P_{m,max}$ of all cases was not significantly different when U_∞ equaled or lower than 0.5 m/s. However, $P_{m,max}$ of the model design II was slightly higher than the rests when $U_\infty = 0.8$ m/s. When $U_\infty = 1.1$ m/s, a significant difference could be observed.

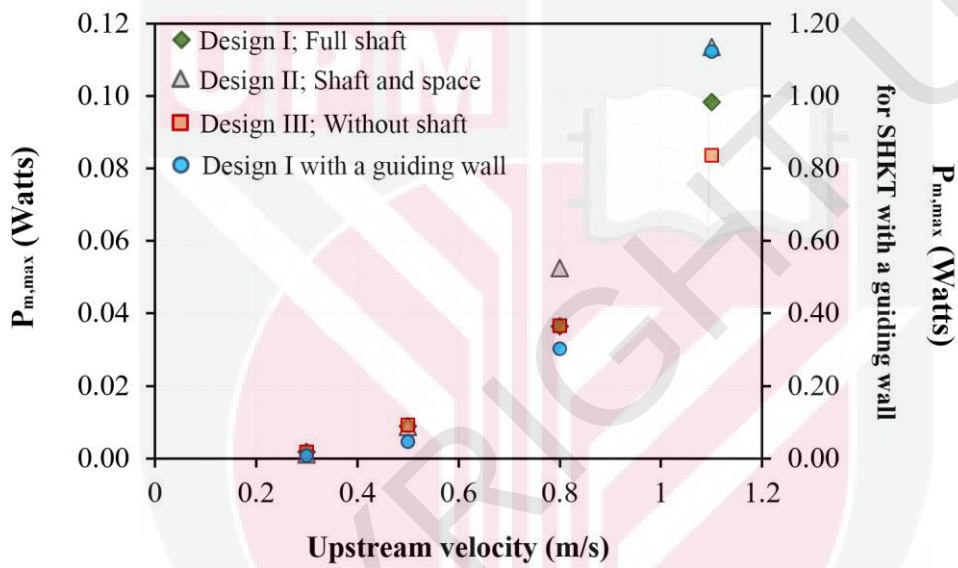


Figure 4.3 : Variation of $P_{m,max}$ (watts) as a function of U_∞ for all cases

Figure 4.4 shows velocity and pressure contours across the SHKT models of all design cases at fixed $U_\infty = 0.8$ m/s and $\theta = 45^\circ$. Flow structures across the models could be observed. The flow recirculation regions occurred on the convex sides, and counter rotating vortices occurred at a trailing edge of a returning bucket. These flow structures influenced directly on SHKT performances.

As showed in velocity contours of design III, water could flow passing through the space between turbine buckets. It contributed to the decreasing of water pressure indicated by the color contour inside a concave surface, and hence decreasing the pressure different between convex and concave surfaces of a bucket. In fact, pressure different was play an important role on torque generation. Hence, the generated torque as illustrated in Figure 4.2 from the design III was lower than the design I and II. Besides, the contours of design I and II were almost similar although there were small gaps between the turbine central shaft and buckets in design II. However, the gaps were too small compared to a rotor diameter or a shaft diameter.

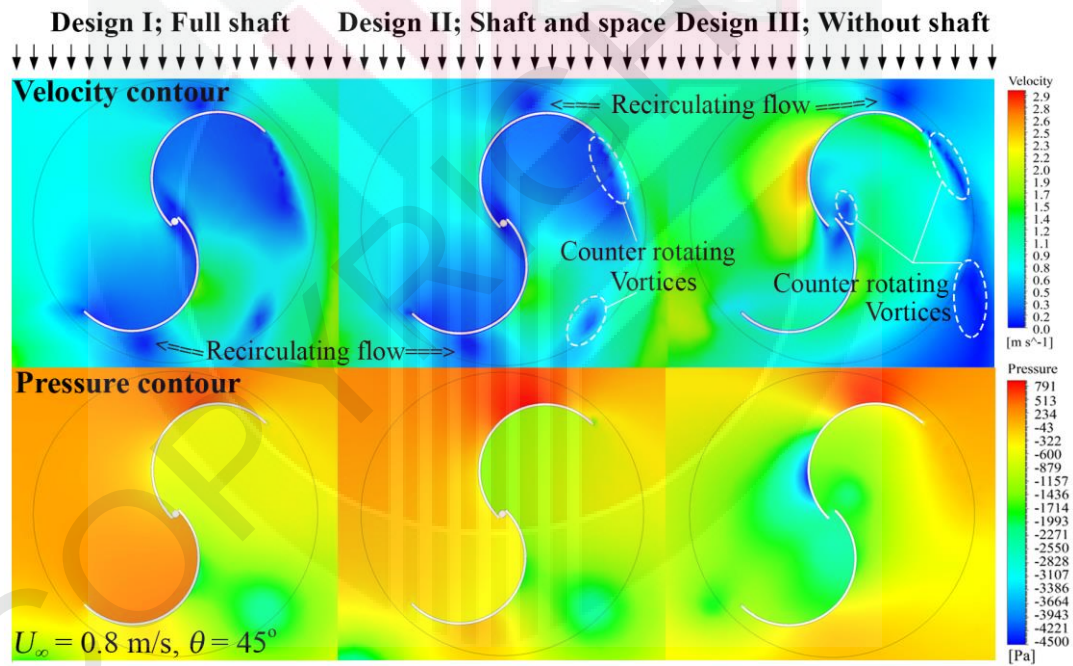


Figure 4.4 : Velocity and pressure contours across 2D SHKT models

4.1.1.2 Effects of a Guiding Wall

The optimum design from previous section has been found. A full shaft SHKT model (Design I) was selected to be modified by adding a guiding wall. The CFD simulations were carried out once again by using an upstream velocity (U_∞) of 0.5 m/s. As mentioned, $P_{m,max}$ of all designs was almost the same when $U_\infty = 0.5$ m/s. To investigate the influence of a guiding wall, a full shaft SHKT model with a guiding wall was simulated under fixed $U_\infty = 0.5$ m/s. The results of $P_{m,max}$ were compared with the models without a guiding wall. As shown in Figure 4.3, it was noticed that, $P_{m,max}$ of a model with a guiding wall was the highest, and higher than $P_{m,max}$ of that one without a guiding wall about 10-13 times.

The variation of c_p and instantaneous torque with azimuthal angle (θ) of a full shaft SHKT model with and without a guiding wall at fixed $U_\infty = 0.5$ m/s are illustrated in Figure 4.5. As seen in the figure, less fluctuation of torque and greater peak of c_p could be observed from the model with a guiding wall. Further, c_p of the model with a guiding wall was going to increase when θ was about 75° , 150° , 260° , and 340° whereas c_p of the model without a guiding wall was going to increase when θ was about 45° , 135° , 225° , and 315° . Having a 45° guiding wall changes the direction of upcoming water. Hence, phase of c_p was shifted around $25^\circ - 30^\circ$ behind the c_p of a model without a guiding wall. From the contours and graphs in Figure 4.5, it was noticed that, the c_p increasing ranges of angles occurred when the bucket A was forced and accelerated from water flowing through it. Whereas, the decreasing range occurred when the SHKT buckets lost their momentum due to the decreasing of hydrodynamic force, leading to the decreasing of rotational speed and correspond with the decreasing of c_p at that range of angular positions. In addition, velocity contours in Figure 4.5 showed

flow field across a full-shaft SHKT model without and with a 45° guiding wall with U_∞ of 0.5 m/s at $\theta = 45^\circ, 90^\circ, 180^\circ, 270^\circ$, and 325° . Flow structures which were counter rotating vortices, and recirculating flow areas could be observed from both cases. when $\theta = 45^\circ, 180^\circ$ and 325° , c_p of a full-shaft SHKT model with a guiding wall was higher because of smaller counter rotating vortices, and no have recirculating flow at a concave surface of a returning bucket. On the other hand, c_p of that case was lower when $\theta = 90^\circ$, and 270° , due to the occurrence of recirculating flow and counter rotating vortices.

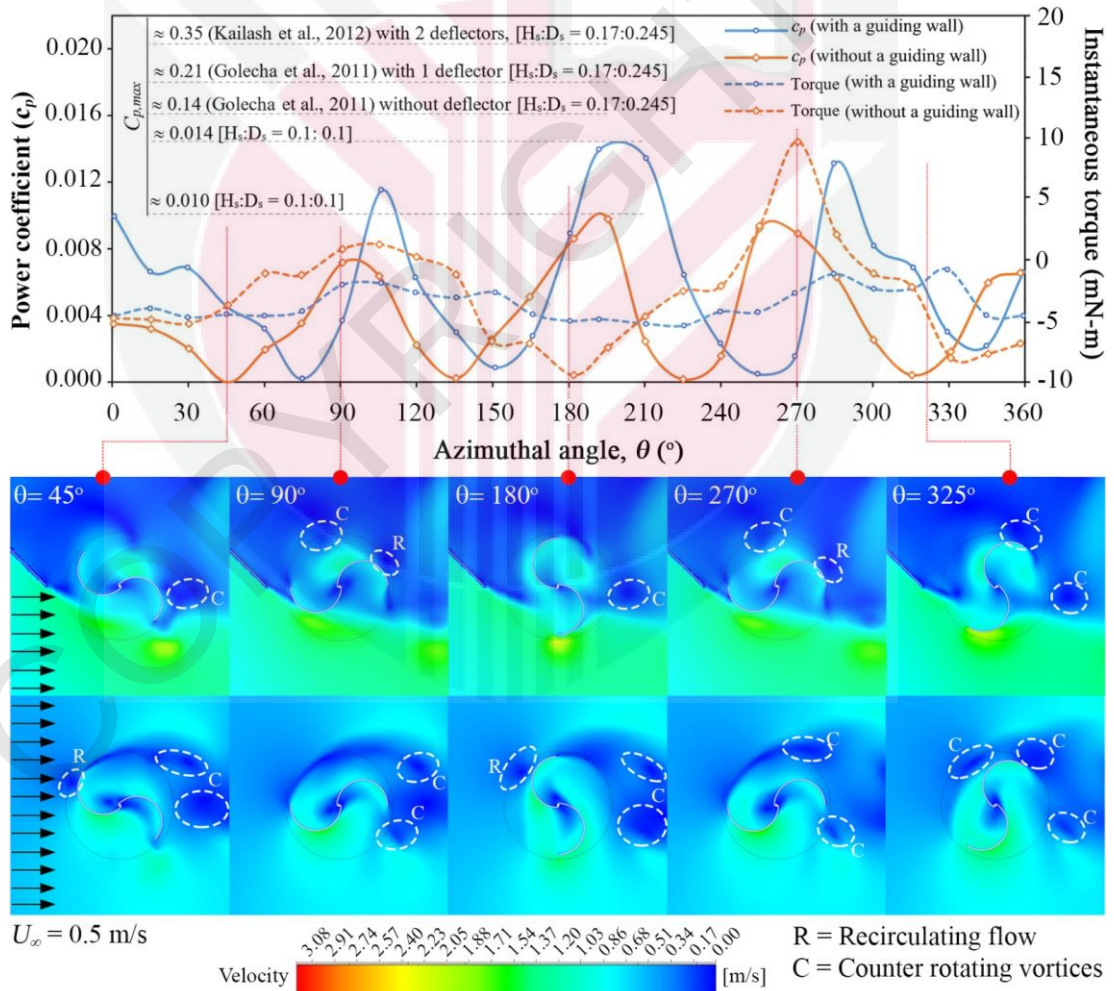


Figure 4.5 : The variation of c_p and instantaneous torque with θ , and the velocity contours of a full-shaft SHKT model with and without a guiding wall at fixed $U_\infty = 0.5$ m/s

Not only flow field characteristics across a SHKT model, pressure distributions and pressure differences between the convex and concave sides of buckets also influenced on the performances as shown in Figure 4.6. It was consistent with the c_p curves shown in Figure 4.5. When the bucket A changed from being an advancing bucket becoming a returning bucket ($\theta = 180^\circ$), the bucket A was suffering with drag force or high pressure from the incoming flow and hence generating the opposite torque direction. At that angular position, the pressure on a convex surface of a bucket A was equal to -42 Pa and 442 Pa for a SHKT model with and without a guiding wall respectively. Thus, the line of a guiding wall increased to surpass the without one. Furthermore, pressure along the concave surface was also important. When $\theta = 135^\circ$, the pressure inside the bucket A of the model with a guiding wall was greater than without a guiding wall, leading to c_p was still higher even though the pressure on the convex surface was higher. When $\theta = 225^\circ$, the model with a guiding wall had lower pressure on the convex surface and higher pressure on the concave surface of the bucket A contributing to higher c_p . However, that was a declining period because the momentum of the model was decreasing. Hence the c_p was also decreasing.

Finally, the comparison between the c_p of the model with a guiding wall from this study and previous studies was shown to prove and emphasize the necessity of the presence of a guiding wall that not only augmented turbine performances where $c_{p,max}$ increased 40% compared to the one without a guiding wall, but it also enhanced hydrodynamics behavior of water flowing through the model as well. It was consistent with the results from the experiment of Golecha et al., 2011 [161] and Kailash et al., 2012 [162] indicated in Figure 4.5.

It was seen that the $c_{p,max}$ from the studies was higher. On the other hand, the turbine size or an aspect ratio (H_s/D_r) also affected directly on turbine performances [38],[83]–[85]. Therefore, a fair comparison is needed to be carried out by the $c_{p,max}$ per projected area ($H_s \times D_r$) of the turbine where H_s was turbine height and D_r was rotor diameter. Based on the results of this study, it could be proved that a full shaft SHKT model with a guiding wall provided better $c_{p,max}$ per projected area than the previous study.

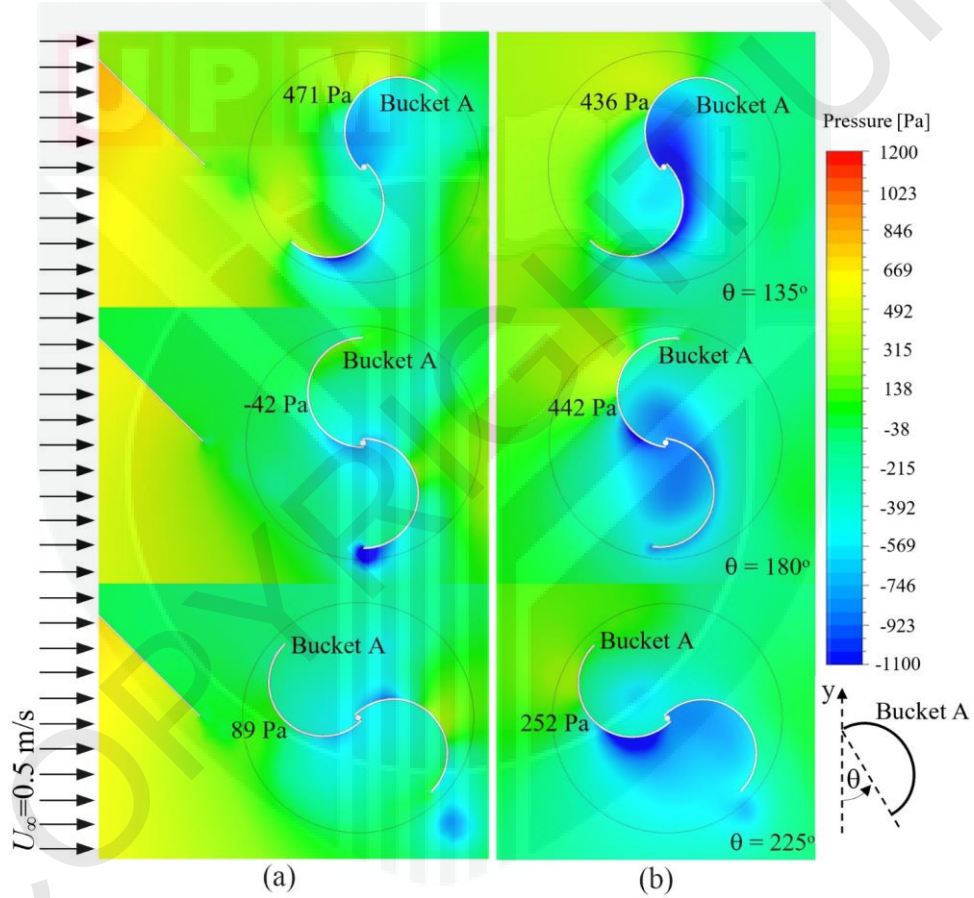


Figure 4.6 : Pressure contours of a full-shaft SHKT model in cases of (a) with and (b) without a guiding wall at $\theta = 135^\circ$, 180° and 225° for fixed $U_\infty = 0.5$ m/s

4.1.2 3D Simulations of the SHKT Models

To optimize the design configurations on the SHKT model in this study, the effect of stage number was studied using 3D CFD simulations. ANSYS Fluent commercial software was employed to investigate the performances including coefficient of power (c_p) and torque (c_t) of the SHKT models from all four cases as mentioned previously. The results of each are discussed in the following subsections.

4.1.2.1 Convergence of Simulation Results

Figure 4.7 shows the instantaneous power coefficient (c_p) and cycle-averaged power coefficient ($c_{p,avg}$) of each revolution over six cycles from the simulation of a SHKT model of Case 4 when $\lambda = 0.8$, $\omega = 30$ rad/s, and $U_\infty = 1.5$ m/s. The instability can be observed for the initial revolutions (revolution 1 to 4), and after that a periodic solution was obtained. The converged periodic results were observed after completing the fourth cycle. Therefore, the simulation was ended after the sixth cycle because the convergent criteria (ε) was less than 1% for two consecutive cycles (cycle 5 and 6).

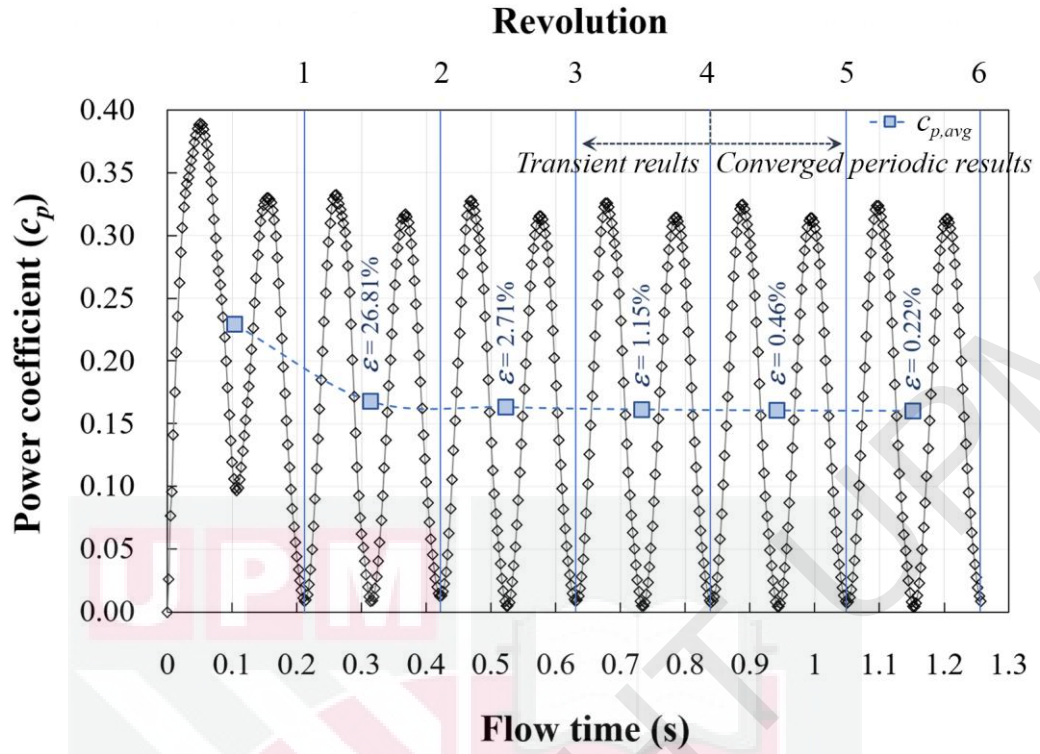


Figure 4.7 : Instantaneous power coefficient (c_p) and cycle-averaged power coefficient ($c_{p,avg}$) of each revolution from the simulation of a model in Case 4

4.1.2.2 Results of Case 1 and Validation with the Study of Saad et al., 2020

A 3D conventional SHKT model demonstrated in Figure 3.15 of case 1 was simulated numerically using air as a medium fluid with a constant upstream velocity of 6 m/s. The rotational speeds of the model were varied from 12 - 60 rad/s. Hence, the tip speed ratios (λ) were varied from 0.4 - 2. The physical parameters mentioned in the previous section of the model were set as similar to the study of Saad et al., 2020 [36].

The comparative studies were carried out to validate the CFD approaches and the simulated results from this study with the literature results of Saad et al., 2020 [36]. The performance curves including c_p and c_t as a function of λ of a SHKT model of case 1 were plotted and compared with those literature results as shown in Figure 4.8.

It was found that there was a good agreement in the simulated results from both results. The maximum power coefficient ($c_{p,max}$) of the model in case 1 was 0.165 occurred at $\lambda = 0.8$ while, $c_{p,max}$ from the study of Saad et al., 2020 was 0.162 occurring at $\lambda = 0.8$. There was only 1.85% difference of $c_{p,max}$ from both studies. Although, both results were correlated, however, a bit difference occurred due to the differences of model designs especially the material of model, and the CFD mesh setting.

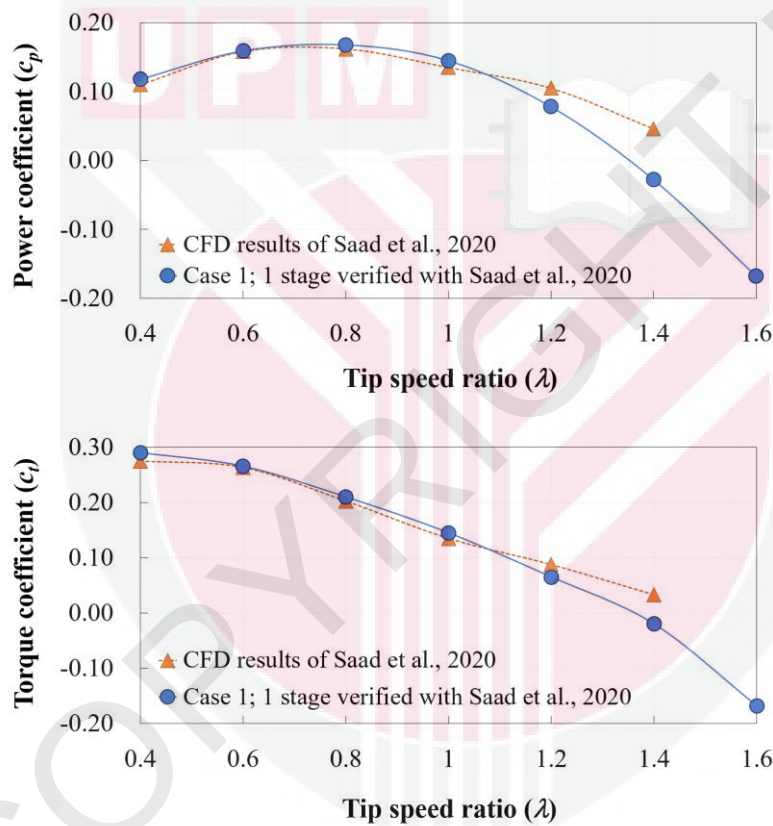


Figure 4.8 : Power and torque coefficient of a conventional SHKT model verified with the CFD results of Saad et al., 2020

Figure 4.9 shows the contour of pressure distribution along with streamlines of the case 1 at different azimuthal angles including 40° , 80° , 120° , and 160° at the section A-A, and the contour from the study of Saad et al., 2020 [36] at azimuthal angles of 0° , and 45° . It can be observed that the pressure distribution between both studies was almost the same. Besides, flow separation at the end of the advancing blade due to adverse pressure gradients was also noticed.

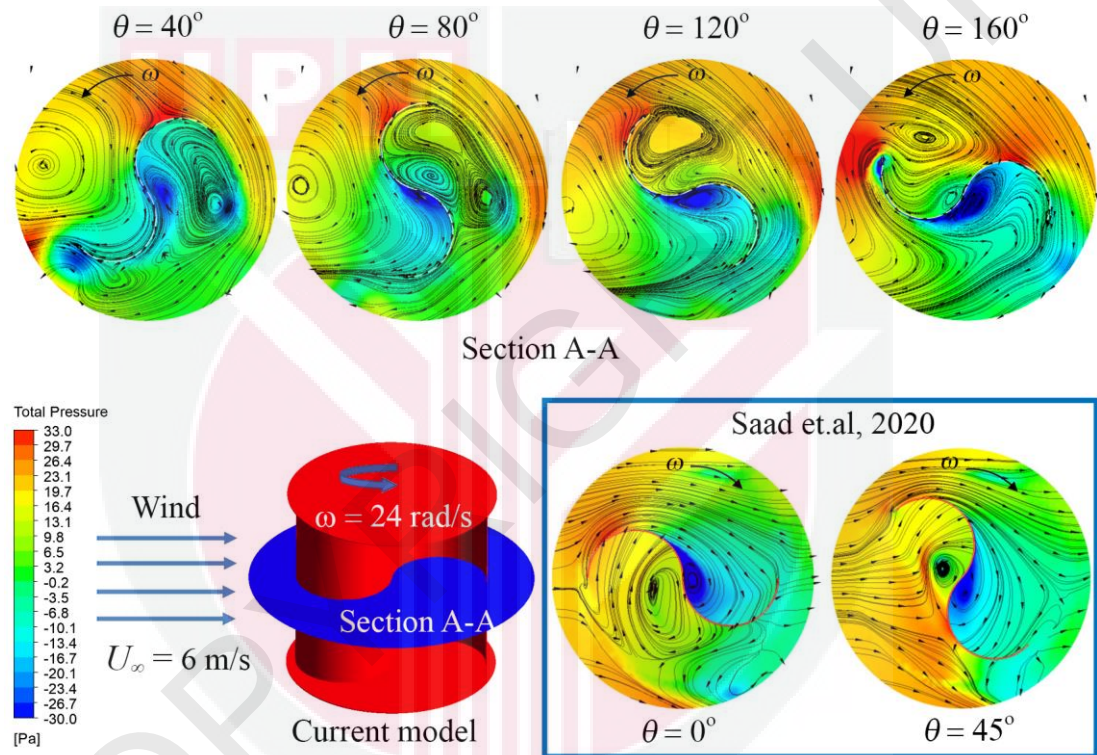


Figure 4.9 : Contour of pressure distribution along with streamlines of the case 1 from this study and the study of Saad et al., 2020

4.1.2.3 Effect of Stage Numbers

CFD simulations of case 2 to case 4 were conducted with the same CFD setting. The medium fluid was changed from air as used in the study of case 1 to water. The simulation results including the power and torque coefficient of the case 1-4 and the results of Saad et al., 2020 are illustrated in Figure 4.10. All predicted results provided similar curves however, there are some differences due to the differences in model

geometry, and CFD setting. Among the case 2-4, the 2-stage SHKT models with constant stage aspect ratio of case 4 performed better, because the c_p and c_t curves are higher compared with the curves of the case 2 and 3.

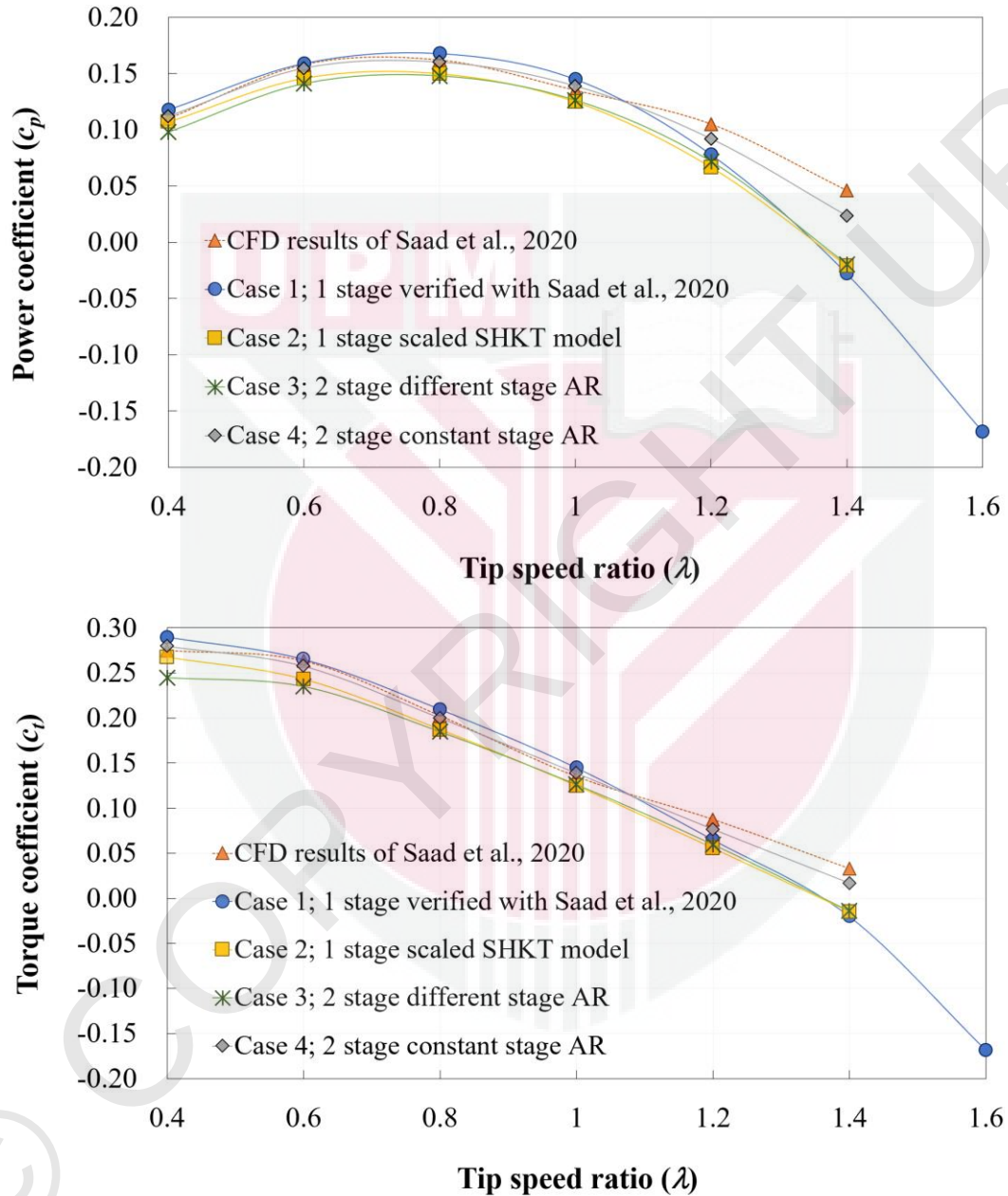


Figure 4.10 : Power and torque coefficient of SHKT models of the case 1 – 4

Figure 4.11 shows the instantaneous power coefficient corresponding with azimuthal angles of the SHKT models of case 2 – 4 when $\omega = 30$ rad/s and $\lambda = 0.8$. It can be seen that, the model of case 2 provided a higher peak of c_p while at the same time, negative torque can also be observed. On the other hand, the instantaneous c_p of the case 3 and case 4 models were all positive. In addition, c_p peaks of the SHKT models of case 4 ($c_{p,max} = 0.324$) were clearly higher than that one of case 3 ($c_{p,max} = 0.294$) around 10%.

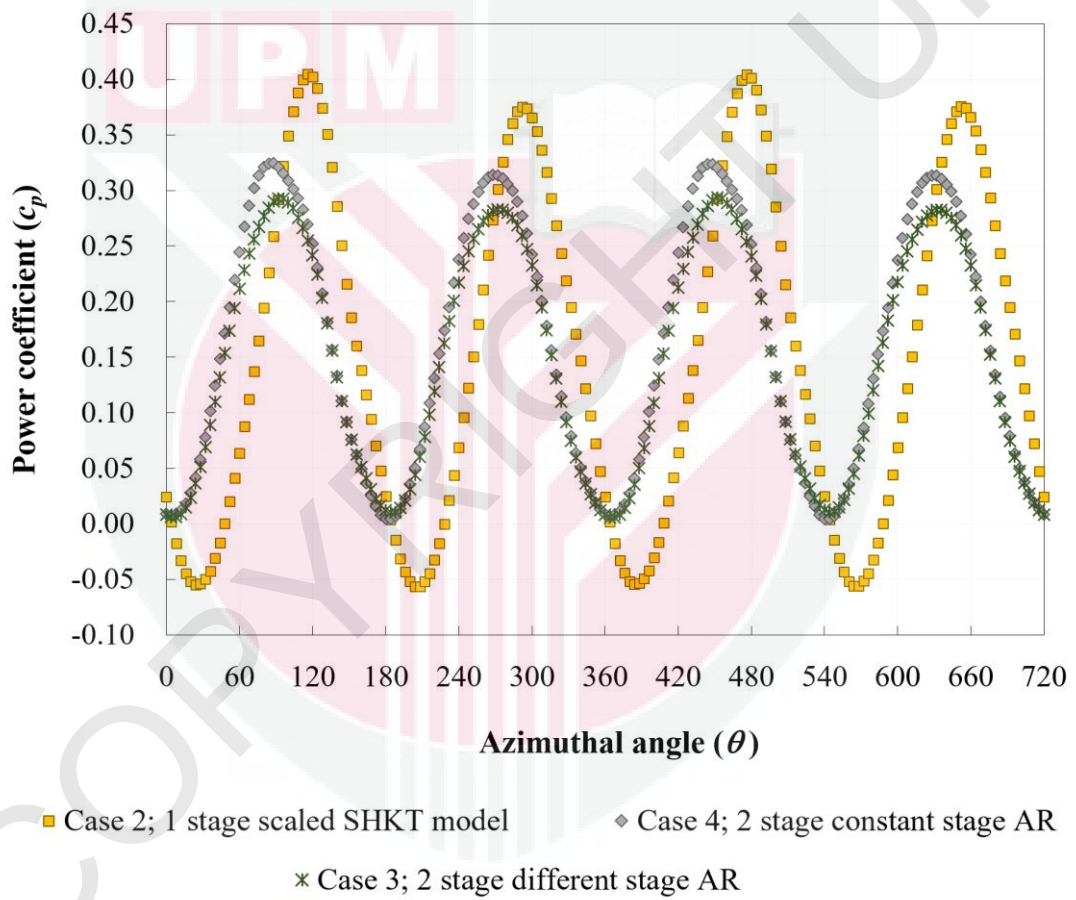


Figure 4.11 : c_p versus θ of the case 2 – 4 when $\omega = 30$ rad/s and $\lambda = 0.8$

4.2 Simulation Results of DHKT Models

4.2.1 Validation of Simulation Results

To assess reliability and accuracy in CFD simulations, computational approaches including the turbulence model (the SST $k-\omega$ turbulence model), boundary and initial conditions, mesh resolution, and domain size have been validated with published experimental results of Doan et al., 2020 [124]. The CAD model was created as similar to the tested model used in the study of Doan et al., 2020 [124] as presented in Table 4.1 (the same geometry, material, and characteristics). The upstream velocity of 0.316 m/s and flow condition were also imposed similarly.

Table 4.1 : Detail specifications of the verification DHKT models using NACA0012 profile

Parameters	Yagmur et al., 2021 [31]	Tian et al., 2013 [40]	Doan et al., 2020 [124]	Present study
Method	2D Simulation	2D Simulation	Experiment	3D Simulation
Turbine	H-type blade	Straight blade	Straight blade	Delta-shaped blade
Characteristic	3 blades	3 blades	3 blades	2, and 3 blades
Turbine size (H_D , D_D , c) (mm.)	(-, 250, 100)	(-, 2000, 150)	(100, 62.8, 25.4)	at Y=0 (150,120,100) At Y=75 mm. (150,120,50)
Upstream velocity	0.46 m/s	0.5 m/s	0.316 m/s	1.5 m/s
Range of TSR	[0.9,1.4]	[1.0,2.5]	(0.6,1.3)	[0.4,1.5]
Maximum c_p	0.182 ($\lambda=1.0$)	Not mentioned	0.108 ($\lambda=0.9$)	0.106 ($\lambda=1.3$)

The validation can be accomplished by comparing the curves of power coefficient (c_p) versus the tip speed ratio (λ) between the simulations and experiments. As seen in Figure 4.12, the curves of the predicted c_p versus λ and the experimental result of Doan et al., 2020 [124] show a good agreement with an average deviation of about 1.25%.

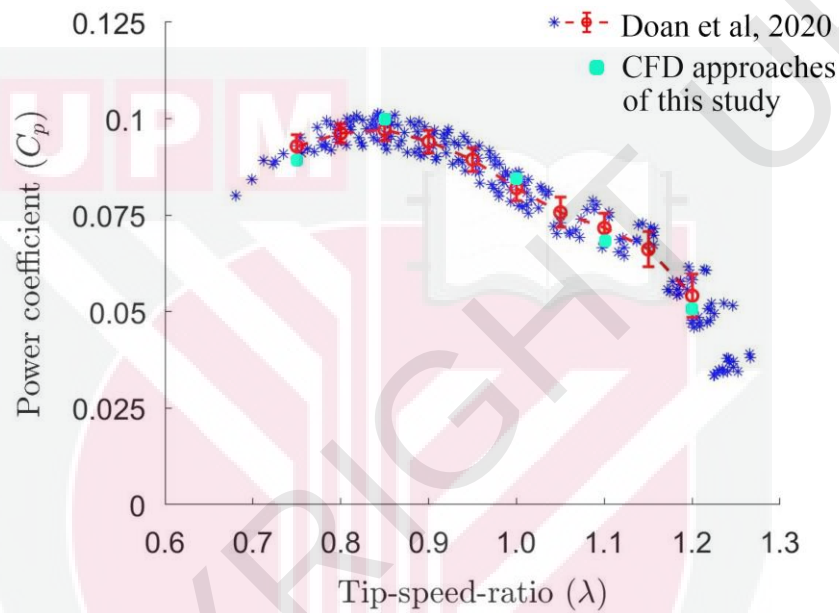


Figure 4.12 : Validation of the current CFD work with experimental data of Doan et al., 2020 [124]

The calculated c_p as a function of λ of *MD3* and *MD8* were compared with the previous computational studies presented in [31],[40] and experimental measurements [124] as shown in Figure 4.13. The previous results were investigated by utilizing the Darrieus hydro turbines with NACA0012 hydrofoils. The geometries of those models were listed in Table 4.1. The experimental and numerical results from previous and present studies showed similar downward parabolic shapes. Each curve had a maximum c_p point. However, there were some differences in the curves as well due to different tested model designs, materials and sizes, tested approaches, and flow conditions.

From Figure 4.13, the simulation result of Yagmur et al., 2021 [31] was slightly higher than others because of using 2D simulation. As mentioned, 2D simulation did not consider the effects of the flow phenomenon at the end of turbine blades as well as the effects of shear flow at the top and bottom walls. Therefore, the c_p of a 2D simulation was overpredicted in contrast with the 3D simulation and experimental results [31]. Moreover, the operational range of λ from Tian et al., 2013 [40] was wider due to the longer diameter of the tested turbine.

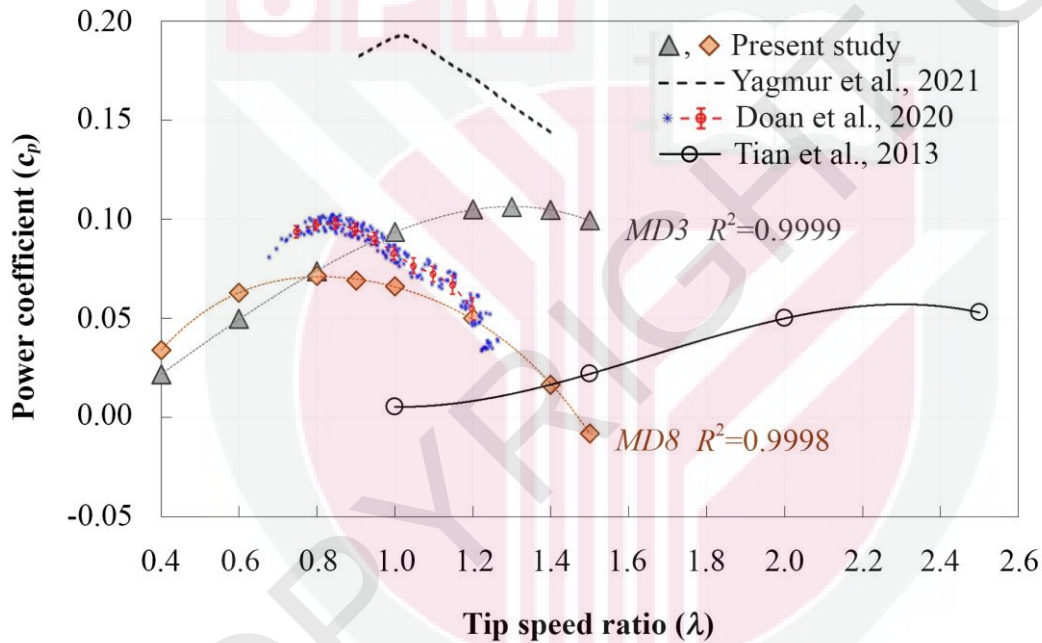


Figure 4.13 : Comparison of simulation results

4.2.2 Effect of the Model Designs

Under the convergence criterion, variations of average c_p and c_t with λ for four different turbine designs, MD1 - MD4 at a constant U_∞ of 1.5 m/s were depicted in Figure 4.14. As illustrated in Figure 4.14(a), c_p and c_t of MD1 and MD2 decreased as λ increased. Both designs always generated the negative torque direction over a complete cycle in every rotational position of a turbine. Hence c_p and c_t were also

negative for all range of λ . On the other hand, c_p and c_t shown in Figure 4.14(b) of *MD3* and *MD4* increased with an increase of λ up to the maximum value of each, and then the c_p and c_t of both were down. The maximum c_p for *MD3* and *MD4* were about 0.1064 at $\lambda = 1.3$, and 0.1012 at $\lambda = 1.2$, respectively. Besides, c_p and c_t of *MD4* were slightly higher than *MD3* when λ was lower than 1.0. Beyond $\lambda = 1.0$, c_p and c_t of *MD3* were clearly higher.

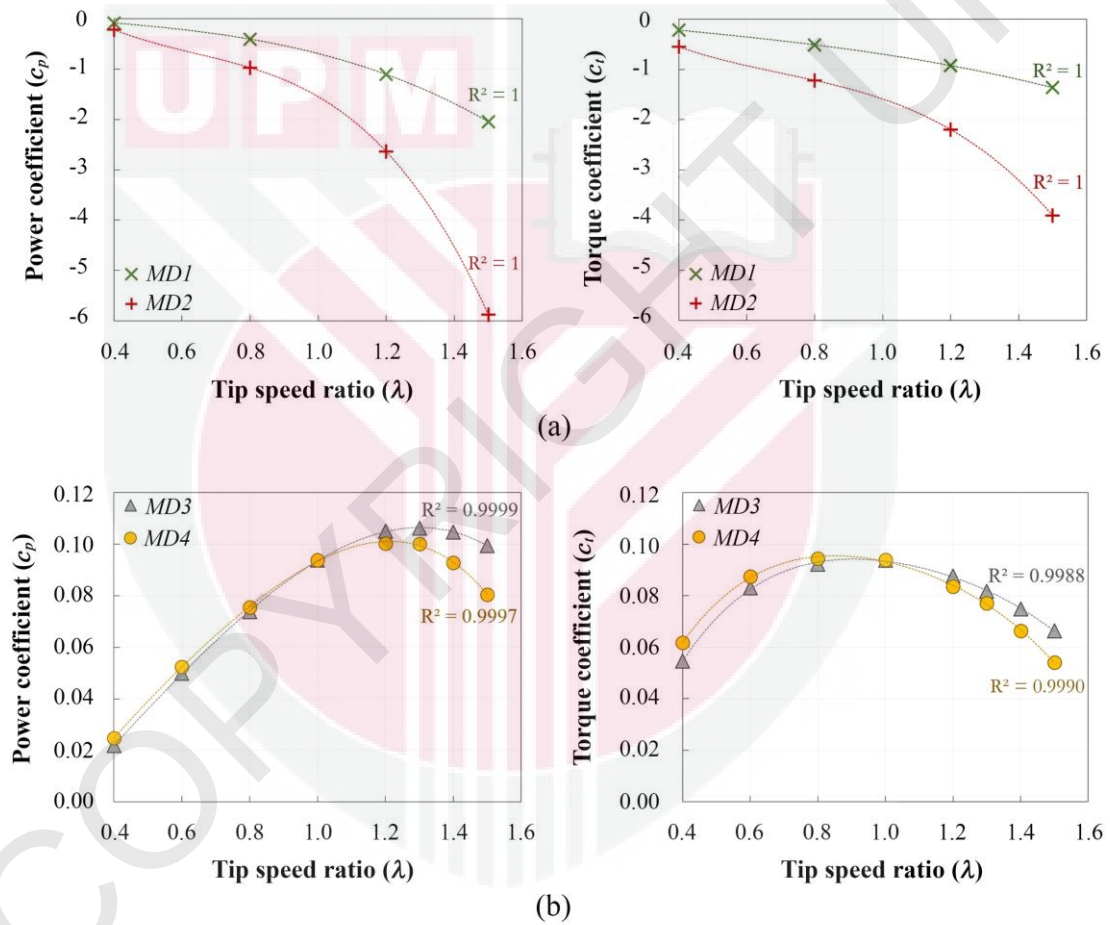


Figure 4.14 : Variation of the average c_p and c_t versus λ of (a) *MD1* and *MD2*, and (b) *MD3* and *MD4* designs

Figure 4.15 illustrates isometric views of velocity streamlines inside the inner domain of MD1 - MD4 designs, when $\theta = 360^\circ$, $\omega_0 = 10$ rad/s, and $U_\infty = 1.5$ m/s. The sections a-a and b-b illustrated in Figure 4.15 at $Y = 0$, and $Y = +0.0375$ m., respectively were used to present flow characteristics on the sections.

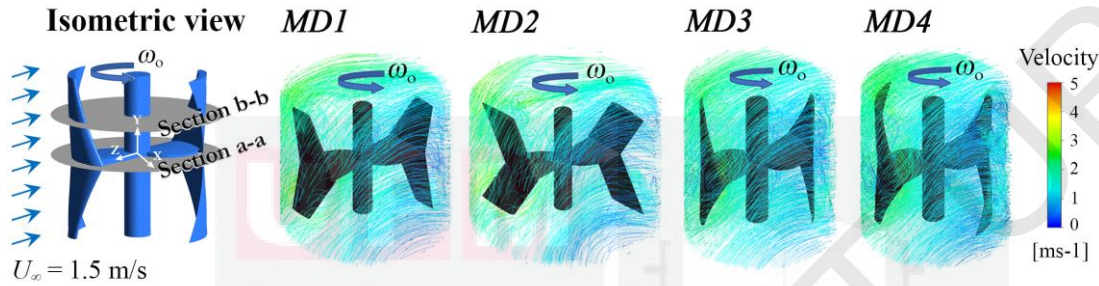
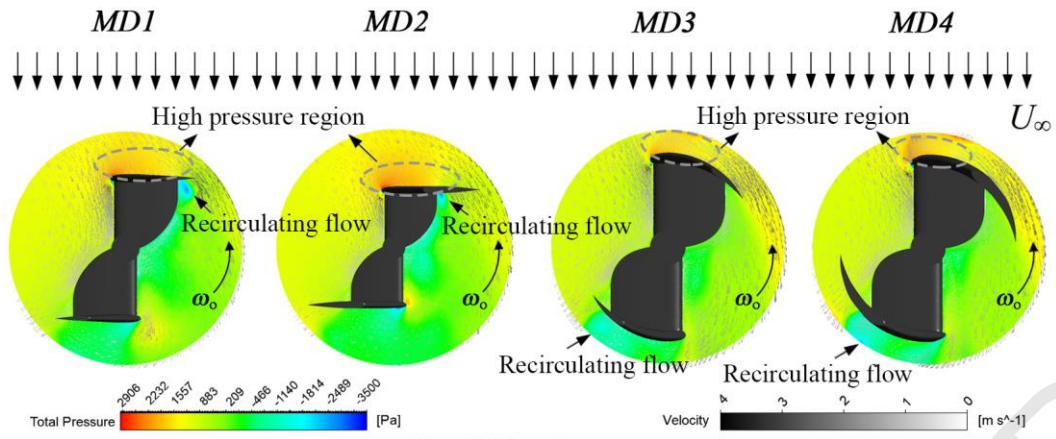


Figure 4.15 : Isometric views of velocity streamlines inside the inner domain of the model designs, MD1 - MD4 when $\theta = 360^\circ$, $\omega_0 = 10$ rad/s, and $U_\infty = 1.5$ m/s

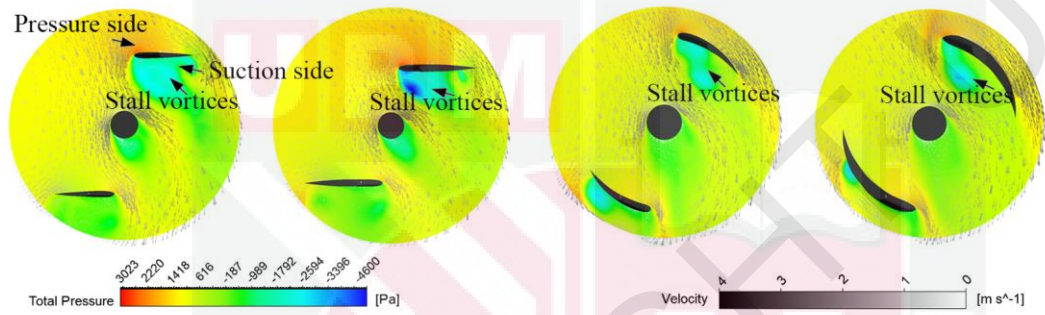
Figure 4.16 shows the sectional total pressure contours along with velocity vectors of flow fields inside the inner domain at two different sections (section a-a, and section b-b) when $\theta = 0^\circ$ and 120° , $\omega_0 = 10$ rad/s, and $U_\infty = 1.5$ m/s. As seen in Figure 4.16, the higher-pressure regions could be observed at the leading edge and pressure side surface where positive pressure was generated from an advancing blade for all designs and the pressure decreased after becoming a returning blade. At $\theta = 0^\circ$, an advancing blade of MD1, and MD2 was hit directly on the pressure side surface by upcoming water flow while water flow could travel along the blade surface of MD3, and MD4. It caused more adverse pressure gradients, and hence dynamic stalls were generated at the trailing edges of MD1, and MD2. Vortices detached from the leading and trailing edge could be observed clearly at section b-b and it was bigger and wider for MD1, and MD2.

When $\theta = 120^\circ$, the higher-pressure zone was seen for *MD1*, and *MD2* at both sections. Besides, thin hydrofoil stalls and recirculating flow also could be observed for *MD1*, and *MD2* due to an angle of attack of blades. An advancing blade of *MD1*, and *MD2* was acted by water pressure and shear stress on its surface at the pressure side surface. Therefore, the negative torque value was generated and then negative c_p and c_t curves as shown in Figure 4.14(a) were produced for all rotational speeds. For the twisted blade designs (*MD3* and *MD4*), pressure distribution and flow behavior on both sections when $\theta = 0^\circ$, and 120° were almost similar.

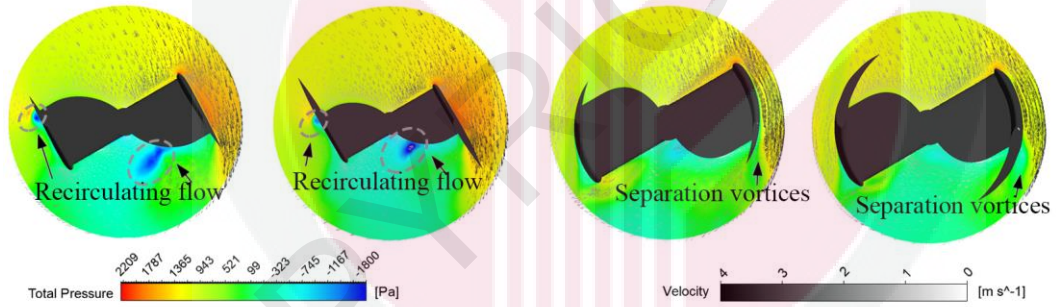
Flow separations at the trailing edge on the pressure side surface could be observed when $\theta = 120^\circ$ on both sections. Vortices detached from leading and trailing were observed on section b-b when $\theta = 0^\circ$, however, it was considerably smaller compared to the in-plane blade designs. Besides, vortex shedding was also noticed behind the cylinder shaft of all models. Based on c_p and c_t curves and flow characteristics, *MD3* and *MD4* were selected to continuously consider on the effect of the number of blades.



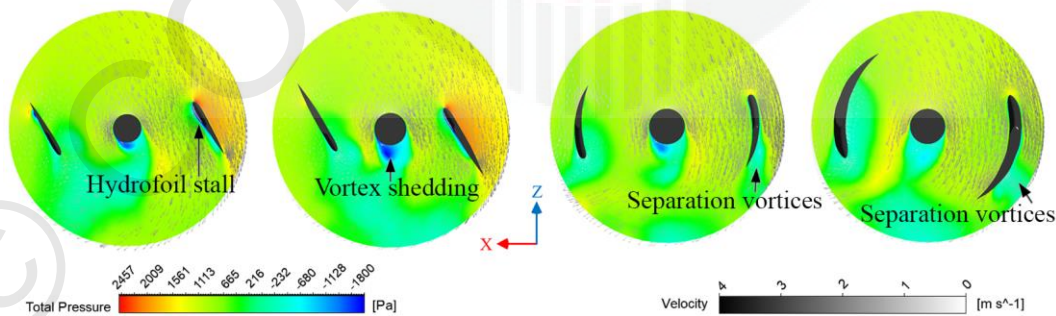
$\theta = 0^\circ$ Section a-a



$\theta = 0^\circ$ Section b-b



$\theta = 120^\circ$ Section a-a



$\theta = 120^\circ$ Section b-b

Figure 4.16 : Sectional total pressure contours and velocity vector around the blades of the DHKT model, MD1 – MD4 at $\theta = 0^\circ$, and 120° , $\omega_o = 10$ rad/s and $U_\infty = 1.5$ m/s

4.2.3 Effect of the Number of Blades

By using similar blade designs and model configurations, 3-blade models that were *MD7* and *MD8* were selected. The *MD5* and *MD6* designs which used in-plane blades were eliminated because of negative torque generation. Under similar simulation approaches, and constant U_∞ of 1.5 m/s, the converged average c_p and c_t with respect to λ of 3-blade models for each simulation were calculated and plotted together with the 2-blade models as illustrated in Figure 4.17(a) and (b).

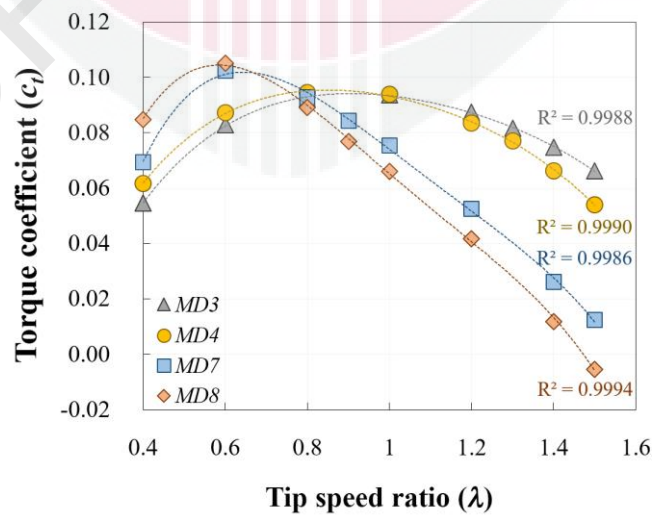
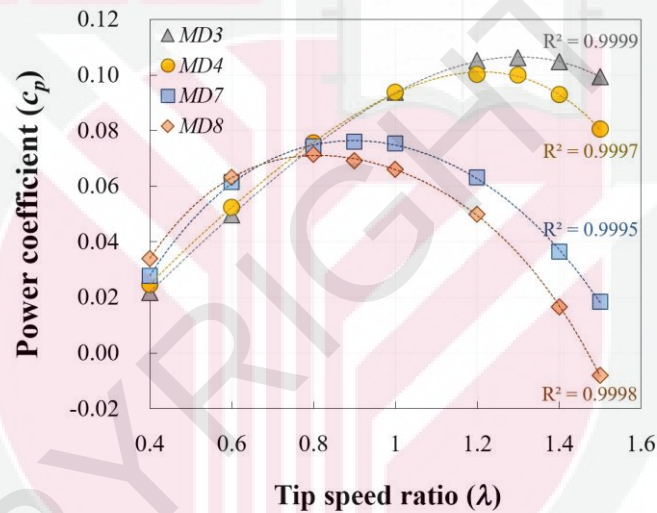


Figure 4.17 : Variation of the average (a) c_p and (b) c_t with λ of 2-blade (*MD3*, *MD4*), and 3-blade (*MD7*, *MD8*) models

The average c_p and c_t curves for the 2-blade models (*MD3*, and *MD4*) and the 3-blade models (*MD7*, and *MD8*) were slightly different. As illustrated in Figure 4.17, the c_p and c_t of *MD7*, and *MD8* were clearly higher than the ones of *MD3*, and *MD4* when λ was lower than 0.8 approximately. Further increasing of λ beyond 0.8, *MD3*, and *MD4* provided higher c_p and c_t . Besides the number of blades, the cross-sectional areas of blades also affected turbine performance. During a low range of λ , the models with constant cross-sectional areas, *MD4* and *MD8* showed higher c_p and c_t while the reduced cross-sectional area blade models, *MD3* and *MD7* provided higher c_p and c_t during a high range of λ . For 2-blade models, the maximum c_p was 0.10623 at $\lambda = 1.3$ for *MD3* and 0.10027 at $\lambda = 1.2$ for *MD4*. While, the maximum c_p of 3-blade models was found to be 0.07594 at $\lambda = 0.9$ for the *MD7*, and 0.07138 at $\lambda = 0.8$ for the *MD8*. Furthermore, the maximum c_t of *MD7* and *MD8* occurred at $\lambda = 0.6$ were around 0.10241, and 0.10519, respectively while maximum c_t points of *MD3* and *MD4* were of 0.09380, and 0.09448 at $\lambda = 1$, and 0.8, respectively.

It could be observed that the 3-blade models and models with constant cross-sectional area blades performed better during a low range of λ whereas the 2-blade models and models with reduced cross-sectional area blades showed higher c_p , and c_t when λ was higher. Flow visualization represented by isometric views of velocity streamlines, velocity, and pressure contours inside the inner domain of the models at the maximum operational condition when $\theta = 360^\circ$, and $U_\infty = 1.5$ m/s were presented in Figure 4.18, and Figure 4.19, respectively.

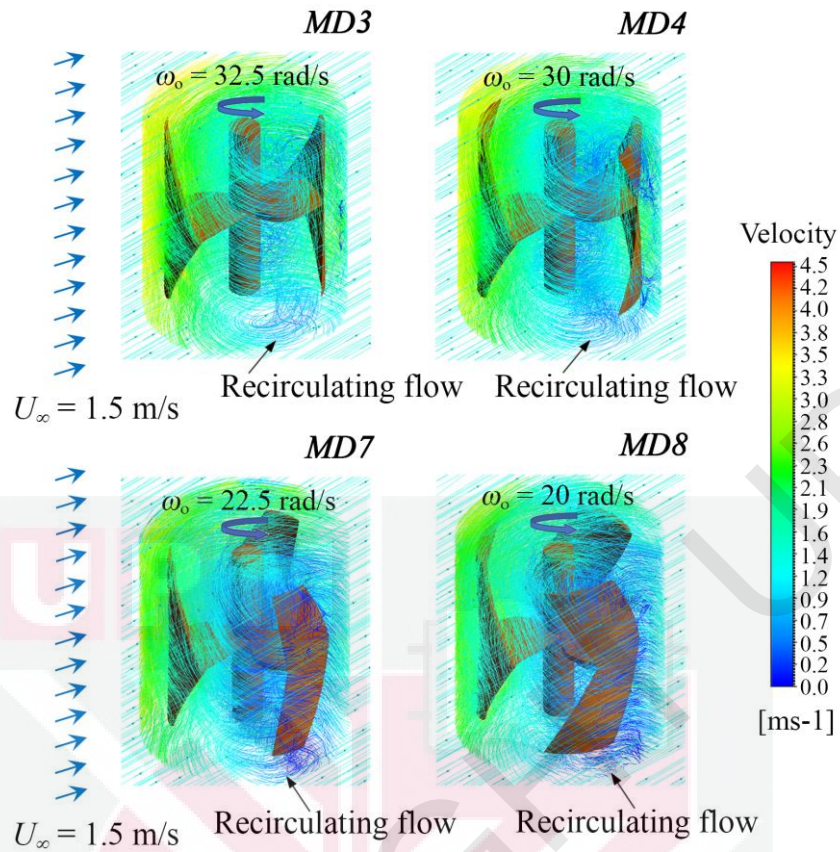


Figure 4.18 : Isometric views inside the inner domain surrounded by velocity streamlines of MD3, MD4, MD7, and MD8 at the maximum operational condition

As seen in Figure 4.19, it was found that flow recirculation areas were observed bigger and wilder for the 3-blade models. The low velocity of water flow inside the 3-blade models was seen clearly. When the 3-blade models rotated, it acted as an obstruction where water could not flow through its blades. It was a reason why the maximum c_p of the 3-blade models was lower than 2-blade designs.

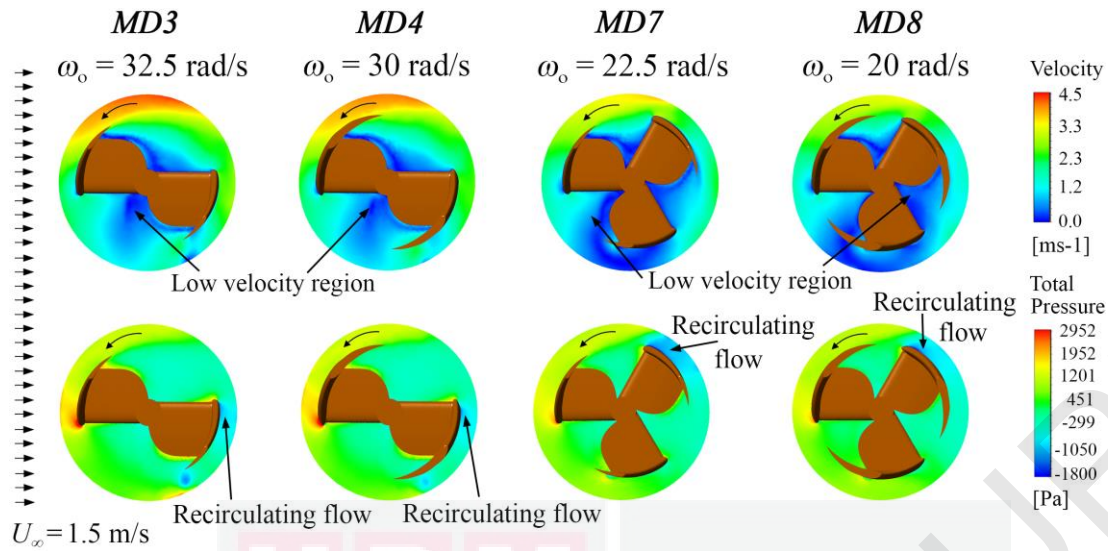


Figure 4.19 : Sectional velocity and total pressure contours of MD3, MD4, MD7, and MD8 at the maximum operational condition

4.3 Result of the Combined SDHKT Models

In this section, simulation results of combined SDHKT models from the effect of the attachment angles and ratios of swept area are presented. The turbine performances including power coefficient (c_p) and torque coefficient (c_t) were calculated and plotted. The contour plots of pressure and velocity at various rotational angles were illustrated to present flow characteristics around the proposed combined turbine. To ensure the accuracy of CFD results from approaches used in this simulation, validation processes also need to take place.

4.3.1 On-Scale Model Validation

To assess accuracy and reliability in computational simulations, computational approaches including the turbulence model (the SST $k-\omega$ turbulence model), boundary and initial conditions, mesh resolution, and domain size have been validated with published experimental results of Sahim et al., 2013 [154]. The CAD model as illustrated in Figure 4.20(a) was created as similar to the tested model depicted in Figure 4.20(b) that was used in the study of Sahim et al., 2013 (the same geometry, material, and characteristics). Flow condition was also imposed similarly.

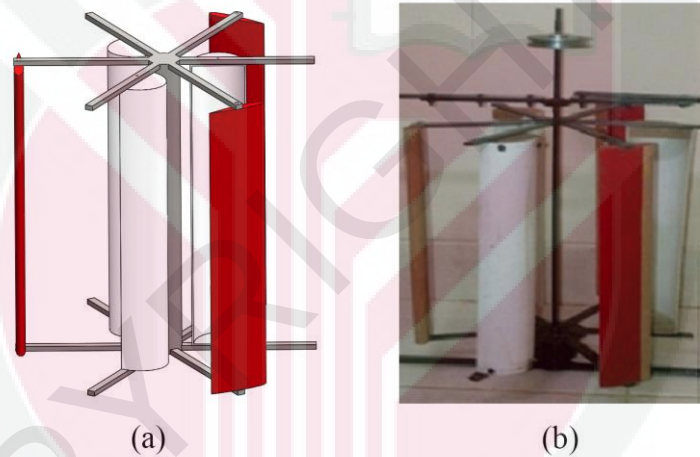


Figure 4.20 : CFD validation of (a) the CAD model in this study and (b) the model from the experiment of Sahim et al., 2013 [154]

The validation can be accomplished by comparing the curves of power coefficient (c_p) with respect to the tip speed ratio (λ) between the simulations and experiments. As seen in Figure 4.21, the variation of the predicted c_p versus λ and the experimental result of Sahim et al., 2013 [154] shows a good agreement with a maximum deviation of about 1.5%. However, the present simulation was slightly higher than the experimental work.

It is proved that the chosen CFD approaches in the current study are reasonably reliable and provide an accurate estimation of the performance. It is also capable of capturing the flow physics of the combined Savonius-Darrieus hydrokinetic turbine.

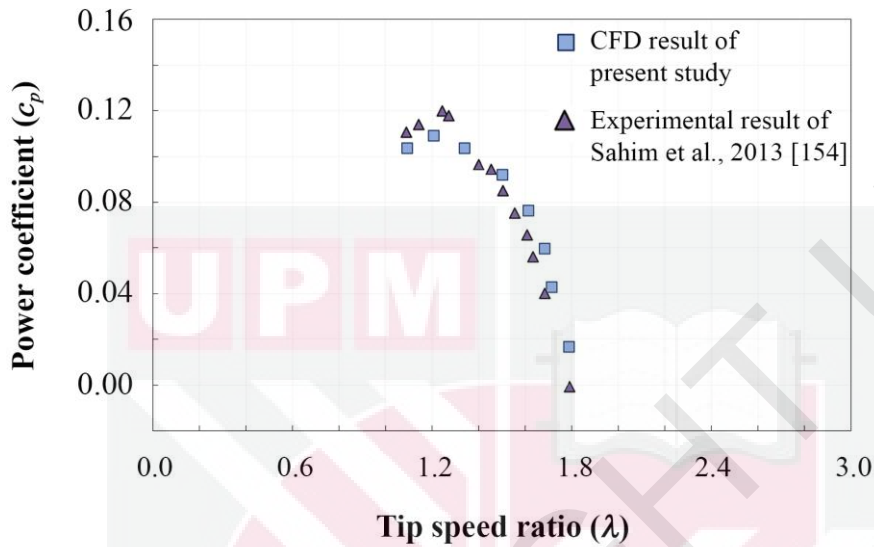


Figure 4.21 : Validation of the current CFD work with experimental data of Sahim et al., 2013 [154]

4.3.2 Comparison of Simulation Results

To compare the predicted results from CFD approaches employed in this study, power coefficient (c_p) curves as a function of tip speed ratios from this study were compared with the curves from the available numerical and experimental results. The four previous works including Saini and Saini, 2020 [46], Saini and Saini, 2020 [153], Sahim et al. 2013 [154], Sahim et al. 2014 [163] which conducted the investigation on the combined Savonius and Darrieus hydrokinetic turbines were selected. The turbine details and tested conditions used in the previous and current studies as well as some results are presented in Table 4.2.

Table 4.2 : Details of combined Savonius-Darrieus turbines used for the validation in this study

Parameters	Present study	Sahim et al. 2013 [154]	Sahim et al. 2014 [163]	Saini and Saini, 2020 [46]	Saini and Saini, 2020 [153]
Method	3D CFD	Experimental	Experimental	3D Numerical	3D Numerical & Experimental
Darrieus turbine	Delta-shaped	Straight blade	Straight blade	Straight blade	Straight blade
Blade profile	NACA0012	NACA0015	NACA0015	S-1046	S-1046
Blade numbers (n)	2	3	2	3	3
Height (H_D)	0.15 m.	0.3 m.	0.6 m.	0.160 m.	0.160 m.
Diameter (D_D)	0.12 m.	0.3 m.	0.4 m.	0.175 m.	0.175 m.
c (m.)	0.1 to 0.05	0.063	0.125	0.055	0.055
Savonius turbine					
Bucket shape	semi-circular	semi-elliptic,	semi-circular	semi-circular	semi-circular
Buckets numbers	2	3	2	2	2
Height (H_S)	0.06 m.	0.3 m.	0.2 m.	0.160 m.	0.160 m.
Diameter (D_S)	0.06 m.	major = 0.132 m. minor = 0.106 m.	0.04 & 0.063 m.	0.105* m.	0.105* m.
Attachment angles ϕ ($^\circ$)	0°	$0^\circ, 60^\circ$	90°	90°	90°
Radius ratio	0.4	0.36, 0.79	2.5, 1.58	0.6	0.3
U_∞ (m/s)	1.5	0.61	0.71	0.5	1.0
Range of λ	0.4-1.5	≈ 0.6 -3.0	≈ 0.8 -2.2	0.2, 0.5, 0.8, 1.1	0.2, 0.5, 0.8, 1.1
$C_{p,max}$	0.094 at $\lambda = 1$	0.12	0.134	0.109 at $\lambda = 0.703$	0.08 at $\lambda = 0.81$

As depicted in Figure 4.22, the experimental and numerical results from previous and present studies show similarly downward parabolic shapes. Each curve has a maximum c_p point. However, there are some differences in c_p curves as well due to different tested model designs, materials, and sizes, tested approaches, and flow conditions.

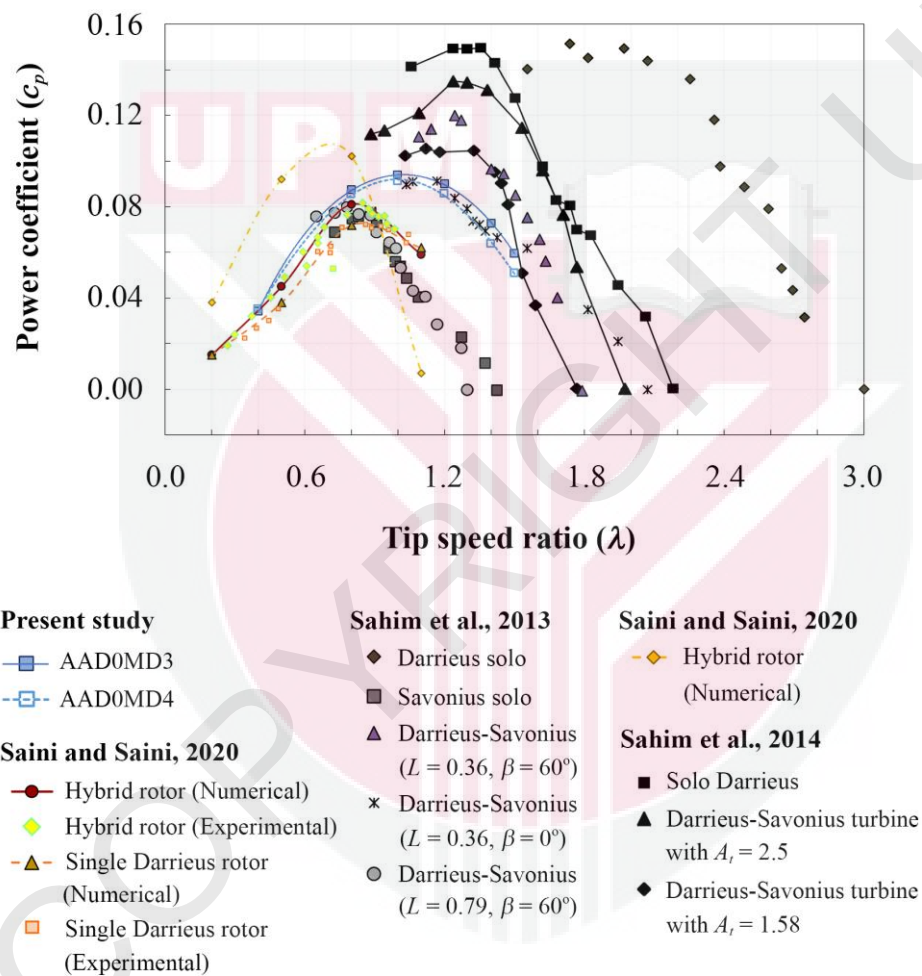


Figure 4.22 : Comparison of the current CFD results and previous published experimental and numerical results

When λ is equalled or lower than 0.8, the c_p curves from this study are almost similar to the curves of a combined Darrieus-Savonius turbine ($L = 0.79$, $\beta = 60^\circ$) from the experiment of Sahim et al. 2013 [154] and a hybrid rotor from the experimental and numerical testing of Saini and Saini, 2020 [153]. After $\lambda = 1$ onward, the curves of this study are definitely close to the curve of the combined Darrieus-Savonius turbine ($L = 0.36$, $\beta = 0^\circ$) from the experiment of Sahim et al. 2013 [154].

4.3.3 Effect of Attachment Angles (ϕ)

An attachment angle of the combined SDHKT models was investigated. The both DHKT models including *MD3* and *MD4* designs were combined with a 2-stage SHKT model. The configurations of the combined SDHKT system were varied with five different values of attachment angles such as AAD0, AAD20, AAD30, AAD90, and AAD120. Therefore, there was a total of 10 combinations that were simulated using the ANSYS fluent and sliding mesh method with six tip speed ratios (λ) for example 0.4, 0.8, 1, 1.2, 1.4, and 1.6 under a constant upstream water velocity (U_∞) of 1.5 m/s. The average power coefficient (c_p) and torque coefficient (c_t) of a converged revolution of each simulation were calculated. The performance curves of the average c_p and c_t versus λ as shown in Figure 4.23 were plotted. The results showed that $c_{p,max}$ of the combined model with *MD3* of the DHKT model and $\phi = 0^\circ$ was 9.36% at $\lambda = 1$ and higher than $c_{p,max}$ of the models with $\phi = 20^\circ$ and 30° around 0.02% and 0.88%, respectively. This trend was similar to the results of the combined SDHKT model composed of the *MD4* design.

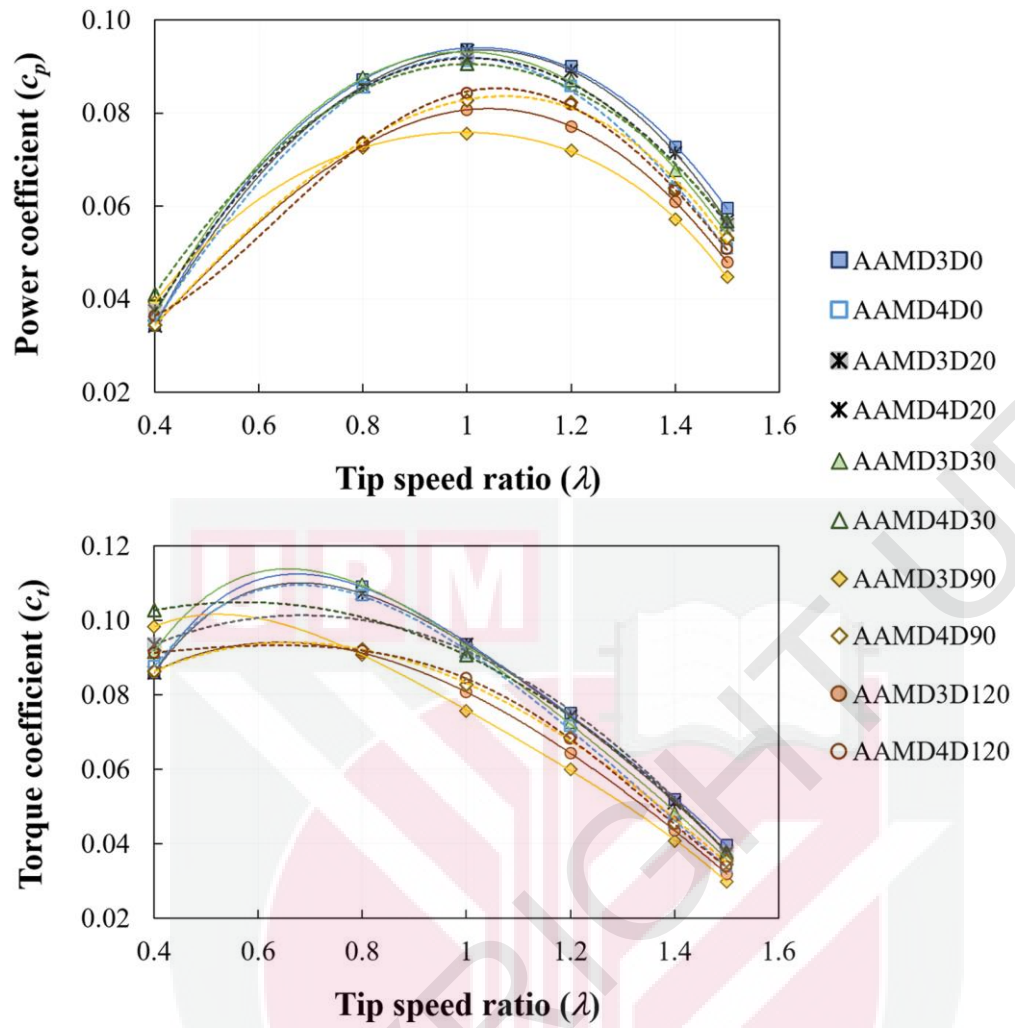


Figure 4.23 : Variation of the average (a) c_p and (b) c_t with tip speed ratios for various attachment angles (φ)

Figure 4.24 shows the average c_p and c_t of a converged revolution with tip speed ratios for five attachment angle designs of the combined SDHKT models combined with MD3 and MD4 of DHKT designs under a constant upstream water velocity of 1.5 m/s.

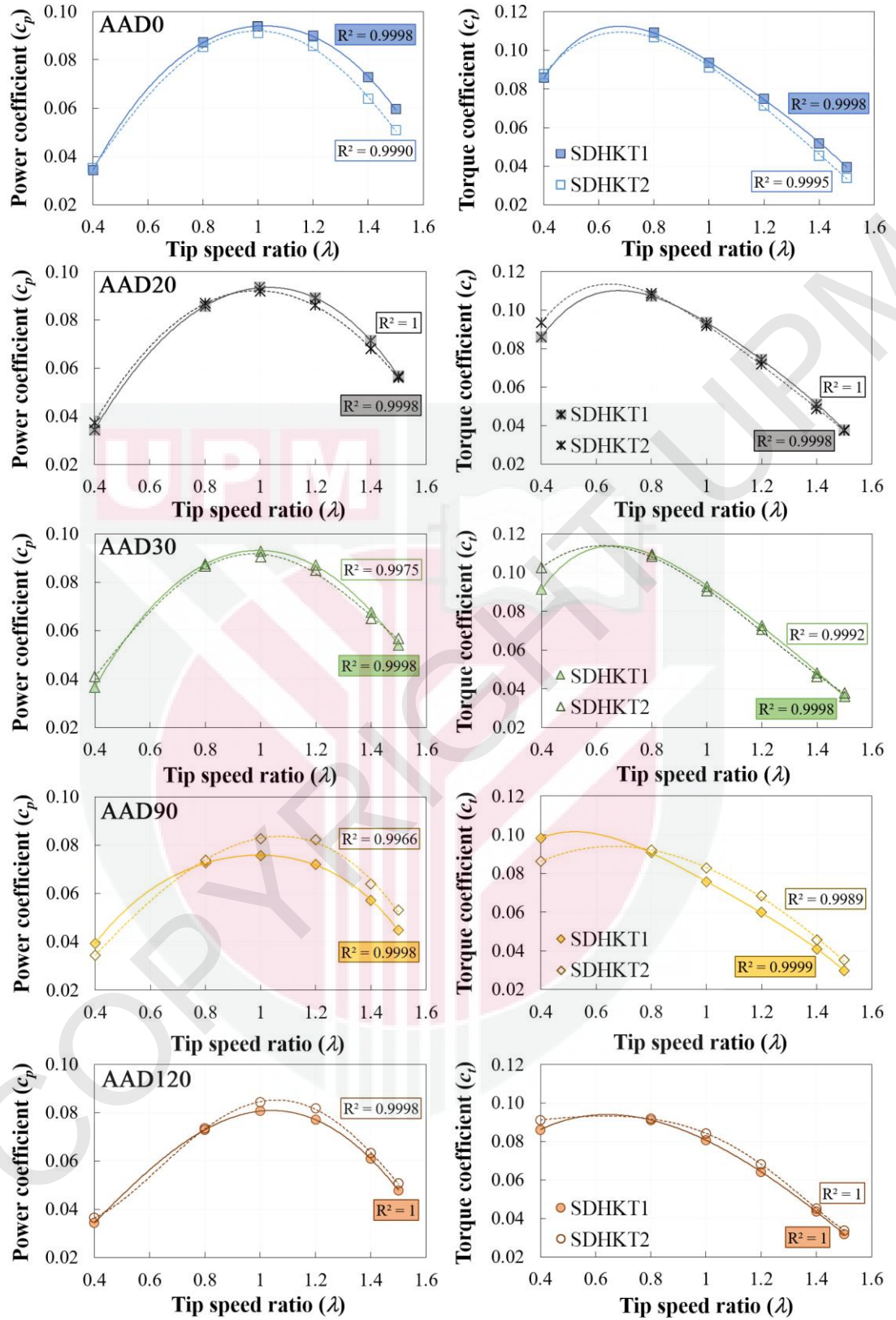


Figure 4.24 : Average c_p and c_t with tip speed ratios for each attachment angle (ϕ)

It was observed that the c_p and c_t curves of combined SDHKT models composed of *MD3* showed a bit higher than that one with *MD4* when attachment angles were 0° , 20° , and 30° , and curves of both for each angle were almost similar. On the other hand, the c_p and c_t curves of the models comprising *MD4* were clearly higher when attachment angles were 90° , 120° .

Figure 4.25 represents the maximum average power coefficient of the combined SDHKT models consisting of the DHKT models with *MD3* and *MD4* from five different attachment angles. It can be seen clearly that, the combined SDHKT models consisting of the DHKT models with *MD3* showed higher maximum average power coefficient when attachment angles were 0° , 20° , and 30° . Whereas, the models combined with *MD4* of the DHKT model provided a higher maximum average power coefficient when attachment angles were 90° and 120° .

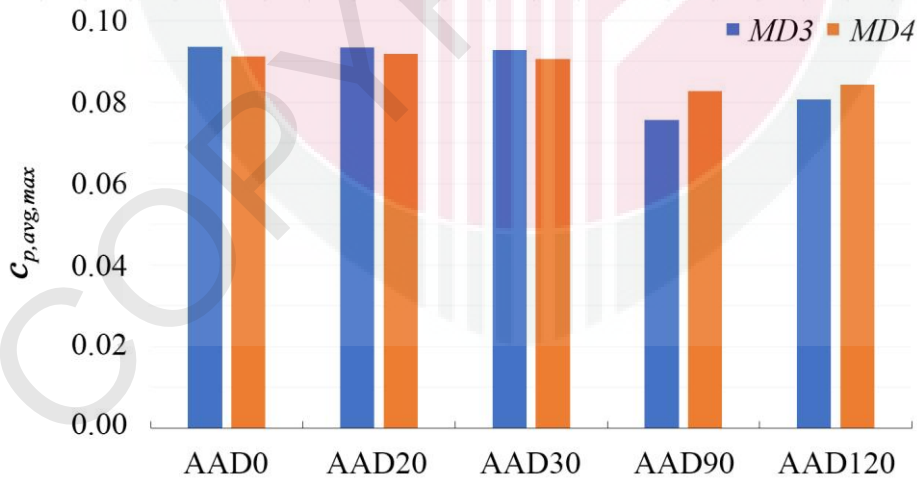


Figure 4.25 : Maximum average power coefficient of SDHKT combined with *MD3* and *MD4* at different attachment angle

4.3.4 Effect of Ratios of Swept Area (R_{sa})

To investigate the effect of ratios of swept area, the optimized combined model that was a combined SDHKT model composed of the 2-stage SHKT model and *MD3* design of a DHKT model with an attachment angle of zero ($\varphi = 0$) from previous studies have been continuously simulated. The average power coefficient (c_p) and torque coefficient (c_t) of a converged revolution from all simulations were calculated and plotted with respect to tip speed ratios (λ) for three different values of ratios of swept areas (R_{sa}) under a constant upstream water velocity (U_∞) of 1.5 m/s and similar CFD setting as mentioned earlier.

According to the trendline correlation function with the polynomial fit of degree 4, as shown in Figure 4.26, it can be seen that the c_p and c_t of R_{sa1} were higher than the rests when λ was lower than 0.9. However, a smaller portion of the SHKT part in the combined system as R_{sa3} showed better results when λ was higher than 0.9. In addition, the highest and lowest maximum c_p that equaled 0.10471 and 0.09364 were R_{sa3} and R_{sa1} , respectively. There were about 11.8% differences between both values. Furthermore, R_{sa1} provided the highest maximum $c_t = 0.10909$ and 7.7% higher than the lowest minimum $c_t = 0.10127$ occurred in the R_{sa3} design.

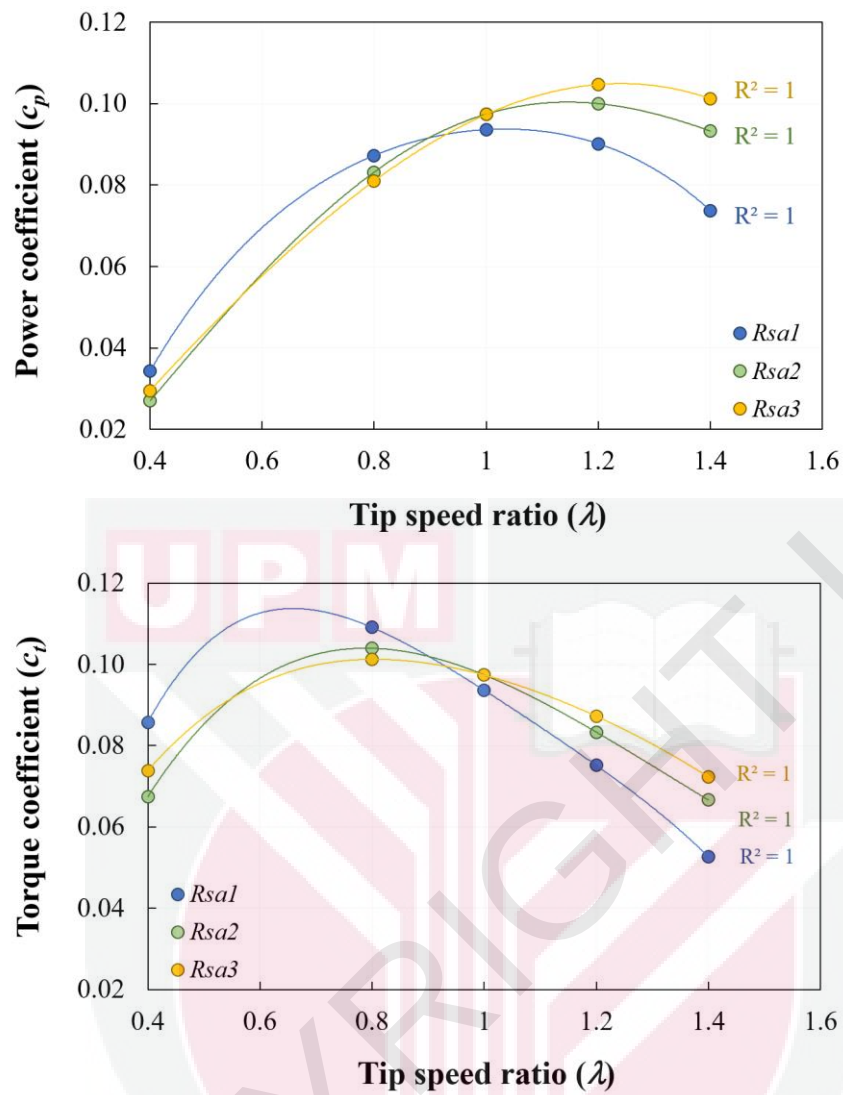


Figure 4.26 : Variation of average c_p and c_t with tip speed ratios at different R_{sa}

Figure 4.27 shows isometric, top, and front views of velocity streamlines and velocity vectors of water flows inside the inner domain around the model (R_{sa3}) when $\theta = 2^\circ$, 58° , 146° , 268° , and 338° under constant upstream velocity (U_∞) of 1.5 m/s. Flow separations were observed at the areas behind the model in every azimuthal angle.

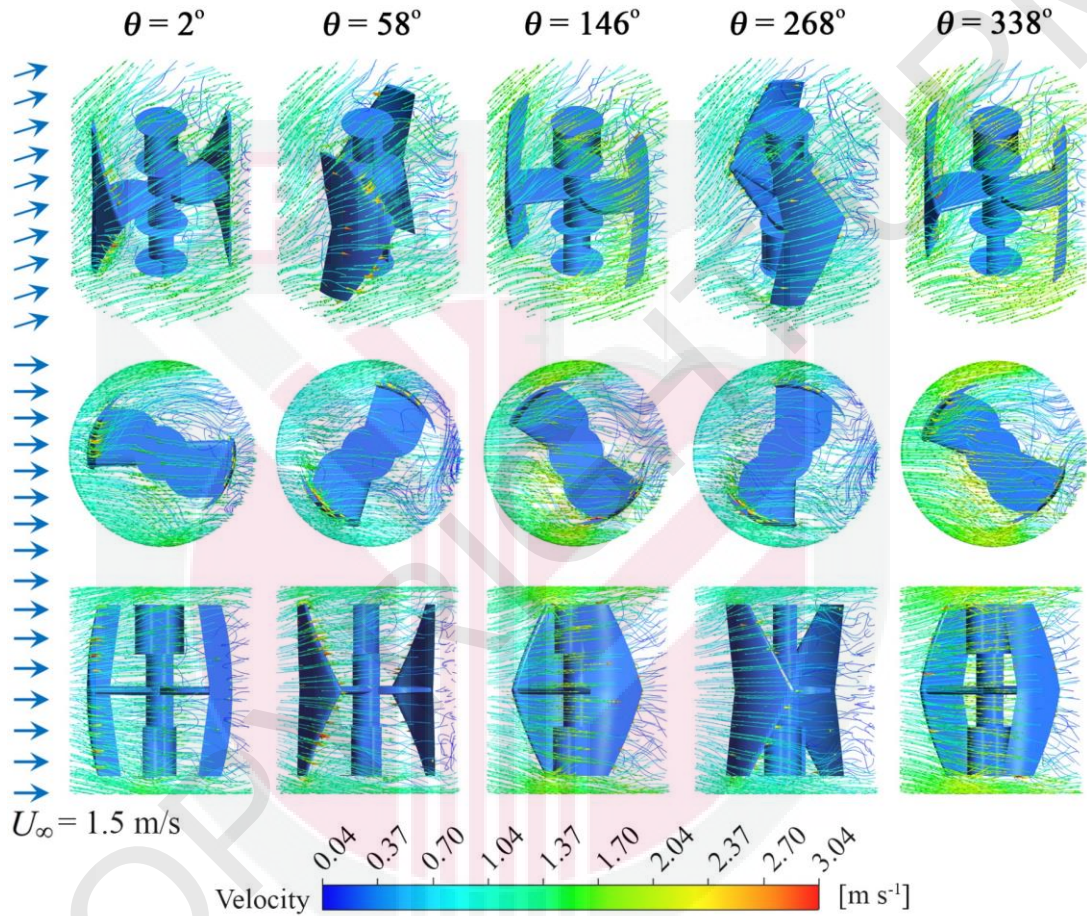


Figure 4.27 : Velocity streamlines and vectors of a combined SDHKT model (R_{sa3})

Velocity contours, streamlines, and vectors of water flow around the combined SDHKT model, and *MD3* of DHKT model under a constant upcoming water velocity of 1.5 m/s are illustrated in Figure 4.28. Those results were also validated with the CFD results of Saini & Saini, 2020 [153]. From the contour plots of the present study, it can be noticed that the water velocity inside either the combined SDHKT or *MD3* models is almost equal to zero. This behavior can be explained by using and analyzing the ANSYS CFD animations. From the animations, the turbine rotors rotated too fast under $U_{\infty} = 1.5$ m/s because of the high tip speed ratio. The rotating blades of those models behaved as a solid obstruction blocking the flow. Therefore, water could not flow through the area inside the models.

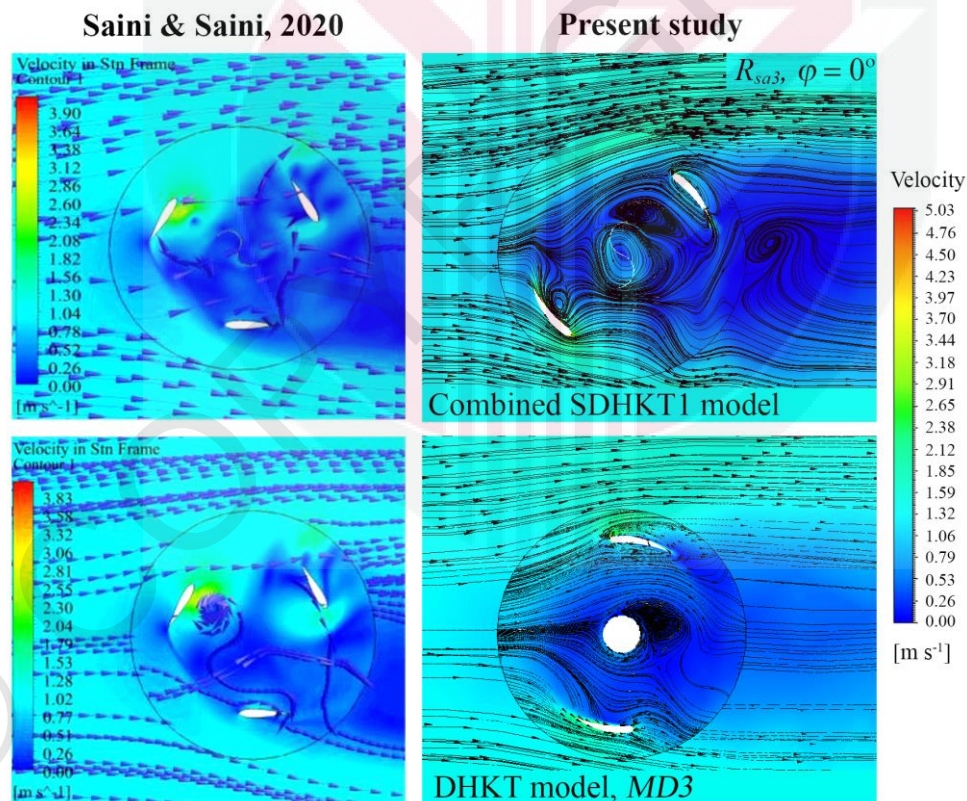


Figure 4.28 : Velocity contours, streamlines, and vectors around the combined SDHKT and the *MD3* of DHKT models compared with results of Saini & Saini, 2020 [153]

CHAPTER 5

CONCLUSION AND RECOMMENDATIONS FOR FUTURE RESEARCH

5.1 Summary and Conclusion

Two-, and three-dimensional simulations were carried out to investigate and optimize the design parameters of the Savonius, Darrieus, and combined Savonius- Darrieus hydrokinetic turbines used in low-speed fluid flow. The investigation processes in ANSYS Fluent software were used to obtain the optimum design parameters for the individual SHKT and DHKT models. To investigate and optimize the design parameters and configurations including the effects either with (i.e., a full central shaft or a central shaft with spacing) or without the central shaft and also the effects either with or without the guiding walls, and the effect of stage numbers of the SHKT models, two-, and three-dimensional transient CFD simulations were conducted. Validation was performed to validate the computational approaches used in the current study with the numerical study of Saad et al., 2020. A good agreement can be observed from both results. The simulated results showed that the starting torque and c_p of a full shaft SHKT with a guiding wall were significantly higher, and torque was less fluctuated as compared to the one without a guiding wall. The maximum power output of the one with a guiding wall was able to be improved by 10 to 13 times and the maximum c_p was increased by 40% as compared to the one without a guiding wall. Besides, the c_p of a two-stage model with a constant stage aspect ratio was significantly higher than a single-stage model and a two-stage model with a different stage aspect ratio.

Similarly, an ANSYS Fluent solver was employed to evaluate and optimize the new designs of the delta-shaped blade Darrieus hydrokinetic turbine (DHKT) models employing NACA0012 hydrofoils. The 2-bladed models with four different designs (*MD1-MD4*) of varying blade characteristics (In-plane, and twisted blades) and cross-sectional areas (reduced, and constant areas) were simulated. The models were positioned fully submerged inside a water flow domain and were forced to rotate with different rotational speeds by utilizing the sliding mesh technique under a constant upstream velocity of 1.5 m/s. Computations in this study were validated versus previous experimental and numerical works in terms of the turbine efficiency curve showing good agreement between numerical and experimental values. The optimum model designs were shown to be the models with twisted blades and reduced and constant cross-sectional areas (*MD3* and *MD4*). The 3-bladed models with similar blade characteristics (*MD7* and *MD8*) were continuously tested and compared with the 2-bladed models. The 2-bladed models performed better during the higher range of tip speed ratio (λ), whereas the 3-bladed models were outstanding at the lower range. Based on the work using CFD approaches, the *MD4* model was shown to be the most appropriate design to operate under the specified conditions.

To design and optimize the combined Savonius-Darrieus hydrokinetic turbine (SDHKT) models, the individual design optimum of SHKT and DHKT models were taken as the fundamental basis. A 2-stage SHKT model was placed at the center and inside *MD3* and *MD4* of DHKT models. The attachment angles (φ) and ratios of swept areas (R_{sa}) of the combined SDHKT configuration were investigated using an ANSYS Fluent solver.

Similar computational approaches were conducted as in the earlier DHKT simulations. The results indicated that $c_{p,max}$ of $\varphi = 0^\circ$ was the highest when λ equaled to 1 and higher by 0.02% and 0.88% when $c_{p,max}$ of $\varphi = 20^\circ$ and 30° . Apart from that, the combined SDHKT models with an attachment angle of 0° and three different ratios of swept areas of $R_{sa1} = 0.4$, $R_{sa2} = 0.3$, and $R_{sa3} = 0.2$ were simulated. The CFD results showed the model with R_{sa1} performed better when λ was lower than 0.9. Beyond 0.9, the performances of the model with R_{sa3} were higher than the rests. These findings confirmed that the SHKT role was more dominant when the ratio of swept areas was higher in the combined SDHKT model. Therefore, the performance curve of the combined SDHKT model with a higher ratio of swept areas was similar to the performance curve of the SHKT model.

Generally, based on the conducted study, the proposed combined Savonius-Darrieus hydrokinetic turbine as depicted in Figure 5.1 should have the following design characteristics as presented in Table 5.1.

1. The two-stage model with a constant stage aspect ratio and without the space between turbine buckets of the Savonius hydrokinetic turbine is placed inside the combined model.
2. The delta-shaped-bladed Darrieus hydrokinetic turbine with the constant bladed cross-sectional area.
3. The configurations of the combined model should have the attachment angle of zero and higher ratios of swept area for use in low-tip speed ratio.

Table 5.1 : Main features of an optimum combined SDHKT model

Parameters	Results
Airfoil blade profile	NACA0012
Characteristic of DHKT blade	Delta-shaped blades
Models of DHKT	MD3 (Reduce cross-sectional area)
Chord length (c) at $Y = 0$	0.10 mm.
Number of blades of DHKT	2
Number of stages of SHKT models	2
Shape of SHKT buckets	2 Semi-circular buckets
Height of SHKT (H_S)	0.06 m.
Aspect ratio (H_S / D_S) of SHKT	1
Angle different between each stage of SHKT models	45°
Height of SDHKT models (H_C)	0.15 m.
Diameter of SDHKT models (D_C)	0.12 m.
Aspect ratio (H_C / D_C) of SDHKT	1.25
Attachment angles (φ)	0°
Shaft diameter (D_s)	0.024 m.
Swept area of 2 SHKT (A_{sS})	0.0072
Swept area of DHKT (A_{sD}) = $H_D \times D_D$	0.018
Ratio of swept area (R_{sa}) = A_{sS} / A_{sD}	0.4

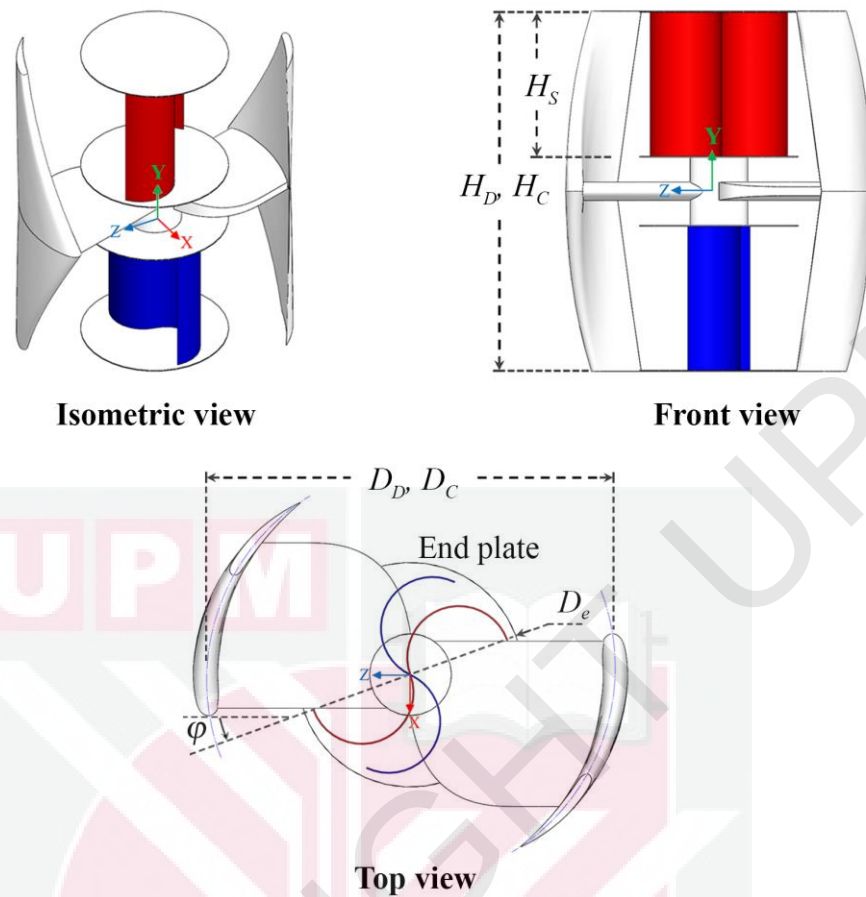


Figure 5.1 : Characteristics of the proposed combined Savonius-Darrieus hydrokinetic turbine

5.2 Recommendations for Future Research

The objectives of this study were initially to optimize separately the design configurations of the Savonius hydrokinetic turbine (SHKT) and the Darrieus hydrokinetic turbine (DHKT) and finally based on this obtained initial knowledge to propose a combined Savonius-Darrieus hydrokinetic turbine (SDHKT) design and optimize its design configuration. The CFD results in these studies provide a perspective for future research regarding the model designs and configuration to evaluate the suitable design. It still has a lot of design parameters of the turbines that can play around to enhance turbine performance.

Additional future work should include the fabrication, an installation of the final model design, and a test at the actual irrigation canal. The performance of the combined Savonius-Darrieus hydrokinetic turbine as well as the electricity generation should be collected for its long-term field performance evaluations.



REFERENCE

- [1] P. K. Yadav, A. Kumar, and S. Jaiswal, "A critical review of technologies for harnessing the power from flowing water using a hydrokinetic turbine to fulfill the energy need," *Energy Reports*, vol. 9, pp. 2102–2117, Dec. 2023,
- [2] M. S. Guney, "Evaluation and measures to increase performance coefficient of hydrokinetic turbines," *Renewable and Sustainable Energy Reviews*, vol. 15, no. 8, pp. 3669–3675, Oct. 2011,
- [3] M. Sood and S. K. Singal, "Development of hydrokinetic energy technology: A review," *International Journal of Energy Research*, vol. 43, no. 11, pp. 5552–5571, 2019,
- [4] W. I. Ibrahim, M. R. Mohamed, R. M. T. R. Ismail, P. K. Leung, W. W. Xing, and A. A. Shah, "Hydrokinetic energy harnessing technologies: A review," *Energy Reports*, vol. 7, pp. 2021–2042, Nov. 2021,
- [5] M. El-Sawy, A. S. Shehata, A. . A. Elbatran, and A. Tawfiq, "Numerical simulation of flow in hydrokinetic turbine channel to improve its efficiency by using first and second-law efficiency analysis," *Ocean Engineering*, vol. 244, p. 110400, Jan. 2022,
- [6] O. Paish, "Small hydro power: technology and current status," *Renewable and Sustainable Energy Reviews*, vol. 6, pp. 537–556, 2002, [Online]. Available: www.elsevier.com/locate/rser
- [7] A. Ali, J. Yuan, H. Javed, *et al.*, "Small hydropower generation using pump as turbine; a smart solution for the development of Pakistan's energy," *Heliyon*, vol. 9, no. 4, p. e14993, Apr. 2023,
- [8] A. Kumar and R. P. Saini, "Performance parameters of Savonius type hydrokinetic turbine – A Review," *Renewable and Sustainable Energy Reviews*, vol. 64, pp. 289–310, Oct. 2016,
- [9] M. I. Yuce and A. Muratoglu, "Hydrokinetic energy conversion systems: A technology status review," *Renewable and Sustainable Energy Reviews*, vol. 43. Elsevier Ltd, pp. 72–82, Mar. 01, 2015.
- [10] K. Tantichukiad, A. Yahya, A. Mohd Mustafah, A. S. Mohd Rafie, and A. S. Mat Su, "Design evaluation reviews on the savonius, darrieus, and combined savonius-darrieus turbines," *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*, vol. 237, no. 6, pp. 1348–1366, Mar. 2023,
- [11] M. F. Basar, A. Ahmad, N. Hasim, and K. Sopian, "Introduction to the pico hydro power and the status of implementation in Malaysia," in *2011 IEEE Student Conference on Research and Development*, Cyberjaya, Malaysia: IEEE, 2011, pp. 283–288.

- [12] C. S. Kaunda, C. Z. Kimambo, and T. K. Nielsen, "A technical discussion on microhydropower technology and its turbines," *Renewable and Sustainable Energy Reviews*, vol. 35. Elsevier Ltd, pp. 445–459, Jul. 01, 2014.
- [13] A. M. A. Haidar, M. F. M. Senan, A. Noman, and T. Radman, "Utilization of pico hydro generation in domestic and commercial loads," *Renewable and Sustainable Energy Reviews*, vol. 16, no. 1, pp. 518–524, Jan. 2012,
- [14] M. J. Khan, G. Bhuyan, M. T. Iqbal, and J. E. Quaicoe, "Hydrokinetic energy conversion systems and assessment of horizontal and vertical axis turbines for river and tidal applications: A technology status review," *Applied Energy*, vol. 86, no. 10. Elsevier Ltd, pp. 1823–1835, Oct. 01, 2009.
- [15] A. Janon, "Torque coefficient analysis of a novel direct-drive parallel-stream counter-rotating darrieus turbine system," *Renewable Energy*, vol. 147, pp. 110–117, Mar. 2020,
- [16] J. L. Menet, "A double-step Savonius rotor for local production of electricity: A design study," *Renewable Energy*, vol. 29, no. 11, pp. 1843–1862, Sep. 2004,
- [17] J. V. Akwa, H. A. Vielmo, and A. P. Petry, "A review on the performance of Savonius wind turbines," *Renewable and Sustainable Energy Reviews*, vol. 16, no. 5. Pergamon, pp. 3054–3064, Jun. 01, 2012.
- [18] M. NAKAJIMA, S. IIO, and T. IKEDA, "Performance of Savonius Rotor for Environmentally Friendly Hydraulic Turbine," *Journal of Fluid Science and Technology*, vol. 3, no. 3, pp. 420–429, 2008,
- [19] S. Roy and U. K. Saha, "Review on the numerical investigations into the design and development of Savonius wind rotors," *Renewable and Sustainable Energy Reviews*, vol. 24. Pergamon, pp. 73–83, Aug. 01, 2013.
- [20] F. Behrouzi, M. Nakisa, A. Maimun, and Y. M. Ahmed, "Global renewable energy and its potential in Malaysia: A review of Hydrokinetic turbine technology," *Renewable and Sustainable Energy Reviews*, vol. 62. Elsevier Ltd, pp. 1270–1281, Sep. 01, 2016.
- [21] A. Pallotta, D. Pietrogiacomini, and G. P. Romano, "HYBRI – A combined Savonius-Darrieus wind turbine: Performances and flow fields," *Energy*, vol. 191, p. 116433, Jan. 2020,
- [22] M. Ghasemian, Z. N. Ashrafi, and A. Sedaghat, "A review on computational fluid dynamic simulation techniques for Darrieus vertical axis wind turbines," *Energy Conversion and Management*, vol. 149, pp. 87–100, Oct. 2017,
- [23] J. Alam and T. Iqbal, "A LOW CUT-IN SPEED MARINE CURRENT TURBINE," *The Journal of Ocean Technology*, vol. 5, no. 4, pp. 49–61, 2010, Accessed: Oct. 29, 2020. [Online]. Available: www.mun.ca

- [24] A. A. Lahimer, M. A. Alghoul, K. Sopian, N. Amin, N. Asim, and M. I. Fadhel, "Research and development aspects of pico-hydro power," *Renewable and Sustainable Energy Reviews*, vol. 16, no. 8, pp. 5861–5878, 2012,
- [25] A. A. Williams and R. Simpson, "Pico hydro - Reducing technical risks for rural electrification," *Renewable Energy*, vol. 34, no. 8, pp. 1986–1991, 2009,
- [26] S. J. Williamson, B. H. Stark, and J. D. Booker, "Low head pico hydro turbine selection using a multi-criteria analysis," *Renewable Energy*, vol. 61, pp. 43–50, 2014,
- [27] K. V. Alexander and E. P. Giddens, "Microhydro: Cost-effective, modular systems for low heads," *Renewable Energy*, vol. 33, no. 6, pp. 1379–1391, Jun. 2008,
- [28] H. J. Vermaak, K. Kusakana, and S. P. Koko, "Status of micro-hydrokinetic river technology in rural applications: A review of literature," *Renewable and Sustainable Energy Reviews*, vol. 29, pp. 625–633, Jan. 2014,
- [29] N. Guillaud, G. Balarac, E. Goncalves, and J. Zanette, "Large Eddy Simulations on Vertical Axis Hydrokinetic Turbines - Power coefficient analysis for various solidities," *Renewable Energy*, vol. 147, pp. 473–486, Mar. 2020,
- [30] K. M. Almohammadi, D. B. Ingham, L. Ma, and M. Pourkashan, "Computational fluid dynamics (CFD) mesh independency techniques for a straight blade vertical axis wind turbine," *Energy*, vol. 58, pp. 483–493, Sep. 2013,
- [31] S. Yagmur and F. Kose, "Numerical evolution of unsteady wake characteristics of H-type Darrieus Hydrokinetic Turbine for a hydro farm arrangement," *Applied Ocean Research*, vol. 110, p. 102582, May 2021,
- [32] Q. Li, T. Maeda, Y. Kamada, *et al.*, "Effect of rotor aspect ratio and solidity on a straight-bladed vertical axis wind turbine in three-dimensional analysis by the panel method," *Energy*, vol. 121, pp. 1–9, Feb. 2017,
- [33] K. H. Wong, W. T. Chong, S. C. Poh, Y. C. Shiah, N. L. Sukiman, and C. T. Wang, "3D CFD simulation and parametric study of a flat plate deflector for vertical axis wind turbine," *Renewable Energy*, vol. 129, pp. 32–55, Dec. 2018,
- [34] H. F. Lam and H. Y. Peng, "Study of wake characteristics of a vertical axis wind turbine by two- and three-dimensional computational fluid dynamics simulations," *Renewable Energy*, vol. 90, pp. 386–398, May 2016,
- [35] M. Lajnef, M. Mosbahi, Y. Chouaibi, and Z. Driss, "Performance Improvement in a Helical Savonius Wind Rotor," *Arabian Journal for Science and Engineering*, vol. 45, no. 11, pp. 9305–9323, Nov. 2020,
- [36] A. S. Saad, I. I. El-Sharkawy, S. Ookawara, and M. Ahmed, "Performance enhancement of twisted-bladed Savonius vertical axis wind turbines," *Energy Conversion and Management*, vol. 209, p. 112673, Apr. 2020,

- [37] L. B. Kothe, S. V. Möller, and A. P. Petry, "Numerical and experimental study of a helical Savonius wind turbine and a comparison with a two-stage Savonius turbine," *Renewable Energy*, vol. 148, pp. 627–638, Apr. 2020,
- [38] Z. Zhao, Y. Zheng, X. Xu, W. Liu, and G. Hu, "Research on the improvement of the performance of savonius rotor based on numerical study," in *2009 International Conference on Sustainable Power Generation and Supply*, Nanjing, China: IEEE, 2009.
- [39] A. Ghosh, A. Biswas, K. K. Sharma, and R. Gupta, "Computational analysis of flow physics of a combined three bladed Darrieus Savonius wind rotor," *Journal of the Energy Institute*, vol. 88, no. 4, pp. 425–437, Nov. 2015,
- [40] W. Tian, B. Song, and Z. Mao, "Conceptual design and numerical simulations of a vertical axis water turbine used for underwater mooring platforms," *International Journal of Naval Architecture and Ocean Engineering*, vol. 5, no. 4, pp. 625–634, Dec. 2013,
- [41] S. Varga, J. Soares, R. Lima, and A. C. Oliveira, "On the selection of a turbulence model for the simulation of steam ejectors using CFD," *International Journal of Low-Carbon Technologies*, vol. 12, no. 3, pp. 233–243, Sep. 2017,
- [42] G. Saini and R. P. Saini, "A numerical analysis to study the effect of radius ratio and attachment angle on hybrid hydrokinetic turbine performance," *Energy for Sustainable Development*, vol. 47, pp. 94–106, Dec. 2018,
- [43] X. Liang, S. Fu, B. Ou, C. Wu, C. Y. H. Chao, and K. Pi, "A computational study of the effects of the radius ratio and attachment angle on the performance of a Darrieus-Savonius combined wind turbine," *Renewable Energy*, vol. 113, pp. 329–334, Dec. 2017,
- [44] R. Gupta and K. K. Sharma, "Flow physics of a combined darrieus-savonius rotor using computational fluid dynamics (CFD)," *International Research Journal of Engineering Science, Technology and Innovation*, vol. 1, no. 1, pp. 1–13, 2012.
- [45] M. H. Mohamed, "Performance investigation of H-rotor Darrieus turbine with new airfoil shapes," *Energy*, vol. 47, no. 1, pp. 522–530, Nov. 2012,
- [46] G. Saini and R. P. Saini, "A computational investigation to analyze the effects of different rotor parameters on hybrid hydrokinetic turbine performance," *Ocean Engineering*, vol. 199, Mar. 2020,
- [47] J. Jacob and D. Chatterjee, "Design methodology of hybrid turbine towards better extraction of wind energy," *Renewable Energy*, vol. 131, pp. 625–643, Feb. 2019,
- [48] S. ed-Dîn Fertahi, T. Bouhal, O. Rajad, *et al.*, "CFD performance enhancement of a low cut-in speed current Vertical Tidal Turbine through the nested hybridization of Savonius and Darrieus," *Energy Conversion and Management*, vol. 169, pp. 266–278, Aug. 2018,

- [49] F. Balduzzi, A. Bianchini, R. Maleci, G. Ferrara, and L. Ferrari, "Critical issues in the CFD simulation of Darrieus wind turbines," *Renewable Energy*, vol. 85, pp. 419–435, Jan. 2016,
- [50] T. Maître, E. Amet, and C. Pellone, "Modeling of the flow in a Darrieus water turbine: Wall grid refinement analysis and comparison with experiments," *Renewable Energy*, vol. 51, pp. 497–512, Mar. 2013,
- [51] A. Arab, M. Javadi, M. Anbarsooz, and M. Moghiman, "A numerical study on the aerodynamic performance and the self-starting characteristics of a Darrieus wind turbine considering its moment of inertia," *Renewable Energy*, vol. 107, pp. 298–311, Jul. 2017,
- [52] M. H. Mohamed, A. M. Ali, and A. A. Hafiz, "CFD analysis for H-rotor Darrieus turbine as a low speed wind energy converter," *Engineering Science and Technology, an International Journal*, vol. 18, no. 1, pp. 1–13, Mar. 2015,
- [53] M. T. Asr, E. Z. Nezhad, F. Mustapha, and S. Wiriadidjaja, "Study on start-up characteristics of H-Darrieus vertical axis wind turbines comprising NACA 4-digit series blade airfoils," *Energy*, vol. 112, pp. 528–537, Oct. 2016,
- [54] A. Kumar and R. P. Saini, "Performance analysis of a single stage modified Savonius hydrokinetic turbine having twisted blades," *Renewable Energy*, vol. 113, pp. 461–478, Dec. 2017,
- [55] J. V. Akwa, G. Alves Da Silva Júnior, and A. P. Petry, "Discussion on the verification of the overlap ratio influence on performance coefficients of a Savonius wind rotor using computational fluid dynamics," *Renewable Energy*, vol. 38, no. 1, pp. 141–149, Feb. 2012,
- [56] F. Guo, B. Song, Z. Mao, and W. Tian, "Experimental and numerical validation of the influence on Savonius turbine caused by rear deflector," *Energy*, vol. 196, Apr. 2020,
- [57] R. Hassanzadeh and M. Mohammadnejad, "Effects of inward and outward overlap ratios on the two-blade Savonius type of vertical axis wind turbine performance," *International Journal of Green Energy*, vol. 16, no. 15, pp. 1485–1496, Dec. 2019,
- [58] P. K. Talukdar, A. Sardar, V. Kulkarni, and U. K. Saha, "Parametric analysis of model Savonius hydrokinetic turbines through experimental and computational investigations," *Energy Conversion and Management*, vol. 158, pp. 36–49, Feb. 2018,
- [59] D. Satrio, I. Ketut Aria Pria Utama, and Mukhtasor, "The influence of time step setting on the CFD simulation result of vertical axis tidal current turbine," *Journal of Mechanical Engineering and Sciences*, vol. 12, no. 1, pp. 3399–3409, Mar. 2018,

- [60] A. Rezaeiha, H. Montazeri, and B. Blocken, "Towards optimal aerodynamic design of vertical axis wind turbines: Impact of solidity and number of blades," *Energy*, vol. 165, pp. 1129–1148, Dec. 2018,
- [61] A. Rezaeiha, I. Kalkman, and B. Blocken, "Effect of pitch angle on power performance and aerodynamics of a vertical axis wind turbine," *Applied Energy*, vol. 197, pp. 132–150, Jul. 2017,
- [62] S. Chhin and V. S. Djanali, "Numerical Simulation on Hybrid Savonius Turbine with NACA-Airfoils as H-rotor Blades," in *2020 International Seminar on Intelligent Technology and Its Applications (ISITIA)*, Surabaya, Indonesia: IEEE, Jul. 2020, pp. 123–128.
- [63] M. Asadi and R. Hassanzadeh, "Effects of internal rotor parameters on the performance of a two bladed Darrieus-two bladed Savonius hybrid wind turbine," *Energy Conversion and Management*, vol. 238, Jun. 2021,
- [64] A. Hosseini and N. Goudarzi, "Design and CFD study of a hybrid vertical-axis wind turbine by employing a combined Bach-type and H-Darrieus rotor systems," *Energy Conversion and Management*, vol. 189, pp. 49–59, Jun. 2019,
- [65] M. H. Mohamed, A. Dessoky, and F. Alqurashi, "Blade shape effect on the behavior of the H-rotor Darrieus wind turbine: Performance investigation and force analysis," *Energy*, vol. 179, pp. 1217–1234, Jul. 2019,
- [66] I. Hashem and M. H. Mohamed, "Aerodynamic performance enhancements of H-rotor Darrieus wind turbine," *Energy*, vol. 142, pp. 531–545, Jan. 2018,
- [67] Ł. Marzec, Z. Buliński, and T. Krysiński, "Fluid structure interaction analysis of the operating Savonius wind turbine," *Renewable Energy*, vol. 164, pp. 272–284, Feb. 2021,
- [68] V. Mendoza, E. Katsidoniotaki, and H. Bernhoff, "Numerical study of a novel concept for manufacturing Savonius turbines with twisted blades," *Energies*, vol. 13, no. 8, Apr. 2020,
- [69] A. Rossetti and G. Pavesi, "Comparison of different numerical approaches to the study of the H-Darrieus turbines start-up," *Renewable Energy*, vol. 50, pp. 7–19, Feb. 2013,
- [70] S. Yagmur, F. Kose, and S. Dogan, "A study on performance and flow characteristics of single and double H-type Darrieus turbine for a hydro farm application," *Energy Conversion and Management*, vol. 245, p. 114599, Oct. 2021,
- [71] N. Ma, H. Lei, Z. Han, *et al.*, "Airfoil optimization to improve power performance of a high-solidity vertical axis wind turbine at a moderate tip speed ratio," *Energy*, vol. 150, pp. 236–252, May 2018,

- [72] P. Marsh, D. Ranmuthugala, I. Penesis, and G. Thomas, "Three-dimensional numerical simulations of straight-bladed vertical axis tidal turbines investigating power output, torque ripple and mounting forces," *Renewable Energy*, vol. 83, pp. 67–77, Nov. 2015,
- [73] S. M. H. Karimian and A. Abdolahifar, "Performance investigation of a new Darrieus Vertical Axis Wind Turbine," *Energy*, vol. 191, p. 116551, Jan. 2020,
- [74] A. M. Abdelsalam, M. A. Kotb, K. Yousef, and I. M. Sakr, "Performance study on a modified hybrid wind turbine with twisted Savonius blades," *Energy Conversion and Management*, vol. 241, p. 114317, Aug. 2021,
- [75] Y. Chen, J. Su, Z. Han, *et al.*, "A shape optimization of ϕ -shape Darrieus wind turbine under a given range of inlet wind speed," *Renewable Energy*, vol. 159, pp. 286–299, Oct. 2020,
- [76] A. Bianchini, F. Balduzzi, P. Bachant, G. Ferrara, and L. Ferrari, "Effectiveness of two-dimensional CFD simulations for Darrieus VAWTs: a combined numerical and experimental assessment," *Energy Conversion and Management*, vol. 136, pp. 318–328, 2017,
- [77] R. Nobile, M. Vahdati, J. F. Barlow, and A. Mewburn-Crook, "Unsteady flow simulation of a vertical axis augmented wind turbine: A two-dimensional study," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 125, pp. 168–179, Feb. 2014,
- [78] M. Elkhoury, T. Kiwata, and E. Aoun, "Experimental and numerical investigation of a three-dimensional vertical-axis wind turbine with variable-pitch," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 139, pp. 111–123, Apr. 2015,
- [79] N. K. Sarma, A. Biswas, and R. D. Misra, "Experimental and computational evaluation of Savonius hydrokinetic turbine for low velocity condition with comparison to Savonius wind turbine at the same input power," *Energy Conversion and Management*, vol. 83, pp. 88–98, Jul. 2014,
- [80] A. Orlandi, M. Collu, S. Zanforlin, and A. Shires, "3D URANS analysis of a vertical axis wind turbine in skewed flows," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 147, pp. 77–84, Dec. 2015,
- [81] K. Kacprzak, G. Liskiewicz, and K. Sobczak, "Numerical investigation of conventional and modified Savonius wind turbines," *Renewable Energy*, vol. 60, pp. 578–585, Dec. 2013,
- [82] K. S. Jeon, J. I. Jeong, J. K. Pan, and K. W. Ryu, "Effects of end plates with various shapes and sizes on helical Savonius wind turbines," *Renewable Energy*, vol. 79, no. 1, pp. 167–176, Jul. 2015,
- [83] N. H. Mahmoud, A. A. El-Haroun, E. Wahba, and M. H. Nasef, "An experimental study on improvement of Savonius rotor performance," *Alexandria Engineering Journal*, vol. 51, no. 1, pp. 19–25, Mar. 2012,

- [84] M. A. Kamoji, S. B. Kedare, and S. V. Prabhu, "Experimental investigations on single stage modified Savonius rotor," *Applied Energy*, vol. 86, no. 7–8, pp. 1064–1073, 2009,
- [85] M. A. Kamoji, S. B. Kedare, and S. V. Prabhu, "Performance tests on helical Savonius rotors," *Renewable Energy*, vol. 34, no. 3, pp. 521–529, 2009,
- [86] A. Damak, Z. Driss, and M. S. Abid, "Experimental investigation of helical Savonius rotor with a twist of 180° ," *Renewable Energy*, vol. 52, pp. 136–142, Apr. 2013,
- [87] M. Jahangir Alam and M. T. Iqbal, "Design and development of hybrid vertical axis turbine," *Canadian Conference on Electrical and Computer Engineering*, pp. 1178–1183, 2009,
- [88] S. Roy and U. K. Saha, "Computational Study to Assess the Influence of Overlap Ratio on Static Torque Characteristics of a Vertical Axis Wind Turbine," *Procedia Engineering*, vol. 51, pp. 694–702, Jan. 2013,
- [89] I. Mabrouki, Z. Driss, and M. S. Abid, "Performance Analysis of a Water Savonius Rotor: Effect of the Internal Overlap," *Sustainable Energy*, vol. 2, no. 4, pp. 121–125, 2014,
- [90] A. Kianifar and M. Anbarsooz, "Blade curve influences on the performance of Savonius rotors: experimental and numerical," *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*, vol. 225, no. 3, pp. 343–350, 2011,
- [91] O. Bin Yaakob, K. B. Tawi, and D. T. S. Sunanto, "Computer simulation studies on the effect overlap ratio for savonius type vertical axis marine current turbine," *International Journal of Engineering, Transactions A: Basics*, vol. 23, no. 1, pp. 79–88, 2010, Accessed: Oct. 21, 2020. [Online]. Available: http://www.ije.ir/article_71836.html
- [92] J.-L. J. Menet, N. Bourabaa, and N. Bouraba, "Increase in the Savonius rotors efficiency via a parametric investigation," *EWEA - 2004 European Wind Energy Conference*, no. April, 2004.
- [93] U. K. Saha, S. Thotla, and D. Maity, "Optimum design configuration of Savonius rotor through wind tunnel experiments," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 96, no. 8–9, pp. 1359–1375, Aug. 2008,
- [94] T. Hayashi, Y. Li, and Y. Hara, "Wind tunnel tests on a different phase three-stage Savonius rotor," *JSME International Journal, Series B: Fluids and Thermal Engineering*, vol. 48, no. 1, pp. 9–16, Feb. 2005,
- [95] M. A. Kamoji, S. B. Kedare, and S. V. Prabhu, "Experimental investigations on single stage, two stage and three stage conventional Savonius rotor," *International Journal of Energy Research*, vol. 32, no. 10, pp. 877–895, 2008,

- [96] W. Tjiu, T. Marnoto, S. Mat, M. H. Ruslan, and K. Sopian, "Darrieus vertical axis wind turbine for power generation I: Assessment of Darrieus VAWT configurations," *Renewable Energy*, vol. 75, pp. 50–67, Mar. 2015,
- [97] I. S. Hwang, Y. H. Lee, and S. J. Kim, "Optimization of cycloidal water turbine and the performance improvement by individual blade control," *Applied Energy*, vol. 86, no. 9, pp. 1532–1540, Sep. 2009,
- [98] S. Jain and U. K. Saha, "On the influence of blade thickness-to-chord ratio on dynamic stall phenomenon in H-type Darrieus wind rotors," *Energy Conversion and Management*, vol. 218, p. 113024, Aug. 2020,
- [99] V. Patel, T. I. Eldho, and S. V. Prabhu, "Experimental investigations on Darrieus straight blade turbine for tidal current application and parametric optimization for hydro farm arrangement," *International Journal of Marine Energy*, vol. 17, pp. 110–135, Apr. 2017,
- [100] A. Subramanian, S. A. Yogesh, H. Sivanandan, *et al.*, "Effect of airfoil and solidity on performance of small scale vertical axis wind turbine using three dimensional CFD model," *Energy*, vol. 133, pp. 179–190, Aug. 2017,
- [101] M. Asadbeigi, F. Ghafoorian, M. Mehrpooya, S. Chegini, and A. Jarrahan, "A 3D Study of the Darrieus Wind Turbine with Auxiliary Blades and Economic Analysis Based on an Optimal Design from a Parametric Investigation," *Sustainability*, vol. 15, no. 5, p. 4684, Mar. 2023,
- [102] S. Armstrong, A. Fiedler, and S. Tullis, "Flow separation on a high Reynolds number, high solidity vertical axis wind turbine with straight and canted blades and canted blades with fences," *Renewable Energy*, vol. 41, pp. 13–22, May 2012,
- [103] Y. T. Lee and H. C. Lim, "Numerical study of the aerodynamic performance of a 500W Darrieus-type vertical-axis wind turbine," *Renewable Energy*, vol. 83, pp. 407–415, Nov. 2015,
- [104] A. R. Sengupta, A. Biswas, and R. Gupta, "Studies of some high solidity symmetrical and unsymmetrical blade H-Darrieus rotors with respect to starting characteristics, dynamic performances and flow physics in low wind streams," *Renewable Energy*, vol. 93, pp. 536–547, Aug. 2016,
- [105] E. Singh, S. Roy, K. S. Yam, and M. C. Law, "Numerical analysis of H-Darrieus vertical axis wind turbines with varying aspect ratios for exhaust energy extractions," *Energy*, vol. 277, p. 127739, Aug. 2023,
- [106] R. Kumar and S. Sarkar, "Effect of design parameters on the performance of helical Darrieus hydrokinetic turbines," *Renewable and Sustainable Energy Reviews*, vol. 162, p. 112431, Jul. 2022,
- [107] M. H. Mohamed, "Criticism study of J-Shaped darrieus wind turbine: Performance evaluation and noise generation assessment," *Energy*, vol. 177, pp. 367–385, Jun. 2019,

- [108] R. Farzadi and M. Bazargan, "3D numerical simulation of the Darrieus vertical axis wind turbine with J-type and straight blades under various operating conditions including self-starting mode," *Energy*, vol. 278, p. 128040, Sep. 2023,
- [109] A. García Auyanet, R. E. Santoso, H. Mohan, S. S. Rathore, D. Chakraborty, and P. G. Verdin, "CFD-Based J-Shaped Blade Design Improvement for Vertical Axis Wind Turbines," *Sustainability (Switzerland)*, vol. 14, no. 22, 2022,
- [110] B. Hand and A. Cashman, "A review on the historical development of the lift-type vertical axis wind turbine: From onshore to offshore floating application," *Sustainable Energy Technologies and Assessments*, vol. 38, p. 100646, Apr. 2020,
- [111] O. S. Mohamed, A. A. Ibrahim, A. K. Etman, A. A. Abdelfatah, and A. M. R. Elbaz, "Numerical investigation of Darrieus wind turbine with slotted airfoil blades," *Energy Conversion and Management: X*, vol. 5, Jan. 2020,
- [112] A. Abdolahifar and S. M. H. Karimian, "A comprehensive three-dimensional study on Darrieus vertical axis wind turbine with slotted blade to reduce flow separation," *Energy*, vol. 248, p. 123632, Jun. 2022,
- [113] C. Attie, A. ElCheikh, J. Nader, and M. Elkhoury, "Performance enhancement of a vertical axis wind turbine using a slotted deflective flap at the trailing edge," *Energy Conversion and Management*, vol. 273, p. 116388, Dec. 2022,
- [114] W. Hao, M. Bashir, C. Li, and C. Sun, "Flow control for high-solidity vertical axis wind turbine based on adaptive flap," *Energy Conversion and Management*, vol. 249, p. 114845, Dec. 2021,
- [115] M. Yousefi Roshan, J. Khaleghinia, M. Eshagh Nimvari, and H. Salarian, "Performance improvement of Darrieus wind turbine using different cavity layouts," *Energy Conversion and Management*, vol. 246, Oct. 2021,
- [116] A. A. Ibrahim, A. M. R. Elbaz, P. F. Melani, O. S. Mohamed, and A. Bianchini, "Power augmentation of Darrieus wind turbine blades using trapped vortex cavity," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 223, p. 104949, Apr. 2022,
- [117] H. Sedighi, P. Akbarzadeh, and A. Salavatipour, "Aerodynamic performance enhancement of horizontal axis wind turbines by dimples on blades: Numerical investigation," *Energy*, vol. 195, p. 117056, Mar. 2020,
- [118] A. Eltayesh, F. Castellani, F. Natili, M. Burlando, and A. Khedr, "Aerodynamic upgrades of a Darrieus vertical axis small wind turbine," *Energy for Sustainable Development*, vol. 73, pp. 126–143, Apr. 2023,

- [119] Y. Yan, E. Avital, J. Williams, and J. Cui, "Aerodynamic performance improvements of a vertical axis wind turbine by leading-edge protuberance," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 211, p. 104535, Apr. 2021,
- [120] A. Bianchini, F. Balduzzi, D. Di Rosa, and G. Ferrara, "On the use of Gurney Flaps for the aerodynamic performance augmentation of Darrieus wind turbines," *Energy Conversion and Management*, vol. 184, pp. 402–415, Mar. 2019,
- [121] L. Ni, W. Miao, C. Li, and Q. Liu, "Impacts of Gurney flap and solidity on the aerodynamic performance of vertical axis wind turbines in array configurations," *Energy*, vol. 215, p. 118915, Jan. 2021,
- [122] F. Scheurich, T. M. Fletcher, and R. E. Brown, "The Influence of Blade Curvature and Helical Blade Twist on the Performance of a Vertical-Axis Wind Turbine," in *29th ASME Wind Energy Symposium*, Orlando, Florida: the American Institute of Aeronautics and Astronautics, Inc., 2010, pp. 1–16. Accessed: Jan. 07, 2010. [Online]. Available: <http://strathprints.strath.ac.uk/27341/>
- [123] S. Laín, P. Cortés, and O. D. López, "Numerical simulation of the flow around a straight blade Darrieus water turbine," *Energies*, vol. 13, no. 5, p. 1137, Mar. 2020,
- [124] M. N. Doan, Y. Kai, and S. Obi, "Twin marine hydrokinetic cross-flow turbines in counter rotating configurations: A laboratory-scaled apparatus for power measurement," *Journal of Marine Science and Engineering*, vol. 8, no. 11, pp. 1–16, 2020,
- [125] A. Dessoky, T. Lutz, and E. Krämer, "Aerodynamic and aeroacoustic performance investigations on modified H-rotor Darrieus wind turbine," *Wind Engineering*, vol. 46, no. 1, pp. 69–88, Feb. 2022,
- [126] R. Ansaf, H. S. Abdelhameed, I. Hashem, and Z. Harun, "Efficiency-based design optimization of the H-type Darrieus wind turbine with fixed guiding-walls," *Energy Reports*, vol. 9, pp. 3576–3592, Dec. 2023,
- [127] I. F. Zidane, H. M. Ali, G. Swadener, Y. A. Eldrainy, and A. I. Shehata, "Effect of upstream deflector utilization on H-Darrieus wind turbine performance: An optimization study," *Alexandria Engineering Journal*, vol. 63, pp. 175–189, Jan. 2023,
- [128] F. Arpino, M. Scungio, and G. Cortellessa, "Numerical performance assessment of an innovative Darrieus-style vertical axis wind turbine with auxiliary straight blades," *Energy Conversion and Management*, vol. 171, pp. 769–777, Sep. 2018,

- [129] F. Arpino, G. Cortellessa, M. Scungio, G. Fresilli, A. Facci, and A. Frattolillo, "PIV measurements over a double bladed Darrieus-type vertical axis wind turbine: A validation benchmark," *Flow Measurement and Instrumentation*, vol. 82, p. 102064, Dec. 2021,
- [130] M. Ahmad, A. Shahzad, F. Akram, F. Ahmad, and S. I. A. Shah, "Design optimization of Double-Darrieus hybrid vertical axis wind turbine," *Ocean Engineering*, vol. 254, p. 111171, Jun. 2022,
- [131] Z. Shen, S. Gong, G. Xie, H. Lu, and W. Guo, "Investigation of the effect of critical structural parameters on the aerodynamic performance of the double darrieus vertical axis wind turbine," *Energy*, vol. 290, p. 130156, Mar. 2024,
- [132] X. Jin, G. Zhao, K. Gao, and W. Ju, "Darrieus vertical axis wind turbine: Basic research methods," *Renewable and Sustainable Energy Reviews*, vol. 42, pp. 212–225, Feb. 2015,
- [133] P. Sabaeifard, H. Razzaghi, and A. Forouzandeh, "Determination of Vertical Axis Wind Turbines Optimal Configuration through CFD Simulations," in *2012 International Conference on Future Environment and Energy*, Singapore: IACSIT Press, 2012, pp. 109–113.
- [134] Q. Li, T. Maeda, Y. Kamada, J. Murata, K. Furukawa, and M. Yamamoto, "Effect of number of blades on aerodynamic forces on a straight-bladed Vertical Axis Wind Turbine," *Energy*, vol. 90, pp. 784–795, Oct. 2015,
- [135] O. Eboibi, L. A. M. Danao, and R. J. Howell, "Experimental investigation of the influence of solidity on the performance and flow field aerodynamics of vertical axis wind turbines at low Reynolds numbers," *Renewable Energy*, vol. 92, pp. 474–483, Jul. 2016,
- [136] S. Joo, H. Choi, and J. Lee, "Aerodynamic characteristics of two-bladed H-Darrieus at various solidities and rotating speeds," *Energy*, vol. 90, pp. 439–451, Oct. 2015,
- [137] D. Gang and W. C. Kau, "Unsteady Flow Numerical Simulation of Vertical Axis Wind Turbine," in *Procedia Engineering*, Elsevier Ltd, Jan. 2015, pp. 734–740.
- [138] D. Saeidi, A. Sedaghat, P. Alamdari, and A. A. Alemrajabi, "Aerodynamic design and economical evaluation of site specific small vertical axis wind turbines," *Applied Energy*, vol. 101, pp. 765–775, Jan. 2013,
- [139] M. A. Singh, A. Biswas, and R. D. Misra, "Investigation of self-starting and high rotor solidity on the performance of a three S1210 blade H-type Darrieus rotor," *Renewable Energy*, vol. 76, pp. 381–387, Apr. 2015,
- [140] C. Simão Ferreira and F. Scheurich, "Demonstrating that power and instantaneous loads are decoupled in a vertical-axis wind turbine," *Wind Energy*, vol. 17, no. 3, pp. 385–396, Mar. 2014,

- [141] W. Chen and C. Y. Zhou, "Application of numerical simulation to obtain the optimization pitch angle for VAWT," in *2009 World Non-Grid-Connected Wind Power and Energy Conference*, Nanjing, China: IEEE, 2009, pp. 110–114.
- [142] R. B. Sumantraa, S. Chandramouli, T. P. Premsai, P. Prithviraj, M. Vivek, and V. R. Kishore, "Numerical analysis of effect of pitch angle on a small scale vertical axis wind turbine," *International Journal of Renewable Energy Research*, vol. 4, no. 4, pp. 929–935, Dec. 2014,
- [143] E. Dyachuk, M. Rossander, A. Goude, and H. Bernhoff, "Measurements of the Aerodynamic Normal Forces on a 12-kW Straight-Bladed Vertical Axis Wind Turbine," *Energies*, vol. 8, no. 8, pp. 8482–8496, Aug. 2015,
- [144] G. Abdalrahman, W. Melek, and F. S. Lien, "Pitch angle control for a small-scale Darrieus vertical axis wind turbine with straight blades (H-Type VAWT)," *Renewable Energy*, vol. 114, pp. 1353–1362, Dec. 2017,
- [145] A. Bianchini, G. Ferrara, and L. Ferrari, "Pitch optimization in small-size darrieus wind turbines," in *Energy Procedia*, Elsevier Ltd, Dec. 2015, pp. 122–132.
- [146] P. Jain and A. Abhishek, "Performance prediction and fundamental understanding of small scale vertical axis wind turbine with variable amplitude blade pitching," *Renewable Energy*, vol. 97, pp. 97–113, Nov. 2016,
- [147] S. Bhuyan and A. Biswas, "Investigations on self-starting and performance characteristics of simple H and hybrid H-Savonius vertical axis wind rotors," *Energy Conversion and Management*, vol. 87, pp. 859–867, Nov. 2014,
- [148] P. Mohan Kumar, M. R. Surya, K. Sivalingam, T.-C. Lim, S. Ramakrishna, and H. Wei, "Computational Optimization of Adaptive Hybrid Darrieus Turbine: Part 1," *Fluids*, vol. 4, no. 2, p. 90, May 2019,
- [149] Y. Kyojuka, "An experimental study on the Darrieus-Savonius turbine for the tidal current power generation," in *Journal of Fluid Science and Technology*, The Japan Society of Mechanical Engineers, 2008, pp. 439–449.
- [150] T. Wakui, Y. Tanzawa, T. Hashizume, and T. Nagao, "Hybrid configuration of darrieus and savonius rotors for stand-alone wind turbine-generator systems," *Electrical Engineering in Japan (English translation of Denki Gakkai Ronbunshi)*, vol. 150, no. 4. John Wiley & Sons, Ltd, pp. 13–22, Mar. 01, 2005.
- [151] R. Gupta, A. Biswas, and K. K. Sharma, "Comparative study of a three-bucket Savonius rotor with a combined three-bucket Savonius-three-bladed Darrieus rotor," *Renewable Energy*, vol. 33, no. 9, pp. 1974–1981, Sep. 2008,

- [152] W. Kou, X. Shi, B. Yuan, and L. Fan, "Modeling analysis and experimental research on a combined-type vertical axis wind turbine," in *2011 International Conference on Electronics, Communications and Control, ICECC 2011 - Proceedings*, Ningbo, China: Institute of Electrical and Electronics Engineers, Inc., 2011, pp. 1537–1541.
- [153] G. Saini and R. P. Saini, "Comparative investigations for performance and self-starting characteristics of hybrid and single Darrieus hydrokinetic turbine," in *Energy Reports*, Elsevier Ltd, Feb. 2020, pp. 96–100.
- [154] K. Sahim, D. Santoso, and A. Radentan, "Performance of combined water turbine with semielliptic section of the savonius rotor," *International Journal of Rotating Machinery*, vol. 2013, 2013,
- [155] S. Sivasegaram, "Secondary parameters affecting the performance of resistance-type vertical-axis wind rotors," *Wind Engineering*, vol. 2, no. 1, pp. 49–58, 1978.
- [156] N. Fujisawa, "On the torque mechanism of Savonius rotors," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 40, no. 3, pp. 277–292, Jan. 1992,
- [157] A. Damak, Z. Driss, and M. S. Abid, "Optimization of the helical Savonius rotor through wind tunnel experiments," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 174, pp. 80–93, Mar. 2018,
- [158] P. Marsh, D. Ranmuthugala, I. Penesis, and G. Thomas, "The influence of turbulence model and two and three-dimensional domain selection on the simulated performance characteristics of vertical axis tidal turbines," *Renewable Energy*, vol. 105, pp. 106–116, May 2017,
- [159] Y. M. Dai and W. H. Lam, "Numerical study of straight-bladed Darrieus-type tidal turbine," *Proceedings of Institution of Civil Engineers: Energy*, vol. 162, no. 2, pp. 67–76, May 2009,
- [160] R. Lanzafame, S. Mauro, M. Messina, and S. Brusca, "Development and Validation of CFD 2D Models for the Simulation of Micro H-Darrieus Turbines Subjected to High Boundary Layer Instabilities †," *AIP Conference Proceedings*, vol. 2191, no. 1, p. 20109, 2019,
- [161] K. Golecha, T. I. Eldho, and S. V. Prabhu, "Influence of the deflector plate on the performance of modified Savonius water turbine," *Applied Energy*, vol. 88, no. 9, pp. 3207–3217, Sep. 2011,
- [162] G. Kailash, T. I. Eldho, and S. V. Prabhu, "Performance study of modified savonius water turbine with two deflector plates," *International Journal of Rotating Machinery*, vol. 2012, 2012,
- [163] K. Sahim, K. Ihtisan, D. Santoso, and R. Sipahutar, "Experimental study of darrieus-savonius water turbine with deflector: Effect of deflector on the performance," *International Journal of Rotating Machinery*, vol. 2014, 2014,

LIST OF PUBLICATIONS

Journal Articles

Tantichukiad, Komsan and Yahya, Azmi and Mohd Mustafah, Anas and Harith, Hazreen Haizi and Mohd Rafie, Azmin Shakrine and Mat Su, Ahmad Suhaizi (2021) Computational studies on the effects of a central shaft and a guiding wall on the savonius hydrokinetic turbine performances. *Advances in Agricultural and Food Research Journal (AAFRJ)*, 2. pp. 1-17. ISSN 2735-1084

Tantichukiad K, Yahya A, Mohd Mustafah A, Mohd Rafie AS, Mat Su AS. Design evaluation reviews on the savonius, darrieus, and combined savonius-darrieus turbines. *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*. 2023;237(6):1348-1366. doi:10.1177/09576509231163965

K. Tantichukiad, A. Yahya, A. M. Mustafah, A. S. Mohd Rafie, and A. S. Mat Su, "Computational Fluid Dynamics Evaluations on New Designs of the Delta-Shaped Blade Darrieus Hydrokinetic Turbine", *Int. J. Automot. Mech. Eng.*, vol. 21, no. 2, pp. 11247–11262, Jun. 2024.