



Prifysgol Cymru
Y Drindod Dewi Sant
University of Wales
Trinity Saint David

**AN INVESTIGATION INTO OPTIMISING A STRUCTURAL DESIGN
FOR WIND FLOW ENERGY PRODUCTION**

BEng Engineering

Mechanical Engineering & Manufacturing

Major Project

M6X00417

BY: Scott Harri Thomas

University of Wales Trinity Saint David

Submitted: 24th April 2018

Declaration of Authenticity

This thesis is the result of the author's original research. It has been composed by the author and has not been previously submitted for examination, which has led to the award of a degree.

Signed:

Date:

Abstract

With the negative effects of climate change due to fossil fuel consumption becoming ever more apparent, renewable energy sources are quickly becoming more prevalent as modern energy generation methods.

A large percentage of the energy demand in modern society comes from commercial and domestic buildings, demanding 39% of total energy use and 68% of electricity consumption in 2016. By implementing 'Green' building designs and incorporating renewable energy methods into buildings, the reliance of energy from fossil fuels can be significantly reduced. Some examples of green building designs in relation to wind power are detailed in this report.

The objective of this report is to investigate the viability of a novel building design that utilises energy from the wind to produce a significant portion of the energy demanded by the building and its included appliances. The design uses stationary geometry features that direct the wind into a central channel that runs vertically through the building and is passed through a wind turbine. Testing is carried out using computational fluid dynamics software to simulate the flow conditions through the channels of the design, the dimensions of the geometry features are optimised through the analysis process to produce the highest wind velocities through the central channel and increase the power output from the turbine.

The results indicate that such a design would be capable of generating substantial percentages of the UK daily average household energy demand (12KWh). Generating approximately 25% in moderate conditions (4m/s wind for 6-hours) and up to 177% in heavy wind conditions (10m/s wind for 6-hours).

Further research is to be conducted the design of the guiding fins to increase the efficiency of the design, the possibility of using the flow in an air-conditioning system as well as testing of a physical model.

List of Illustrations/Tables

- Fig. 1 Gantt chart of project timeline (Thomas S. H., 2018).
- Fig. 2 Bahrain World trade centre(WayMarking, 2018).
- Fig. 3 The Secret Energy Turbine (SET) (NewAtlas, 2010).
- Fig. 4 Sketch of initial design (Thomas S. H., 2018).
- Fig. 5 Axial-flow turbine, half section view (eternoohydro, 2014).
- Fig. 6 Bulb turbine with flow channels depicted (WikiMedia, 2017).
- Fig. 7 The Venturi effect within a pipe (ScienceStruck, 2018).
- Fig. 8 2D sketch of concept design internal flow channels (Thomas S. H., 2018).
- Fig. 9 Inlet height and inlet radius (Thomas S. H., 2018).
- Fig. 10 2D plane surface, meshed for simulation (Thomas S. H., 2018).
- Fig. 11 Close up of inflation layers in the Venturi section (Thomas S. H., 2018).
- Table. 1: Results from 2D analysis (Thomas S. H., 2018).
- Fig. 12 Graph showing the relationship between the inlet radius and the maximum velocity in the structure (Thomas S. H., 2018).
- Fig. 13 Spilling occurring due to a large inlet radius (Thomas S. H., 2018).
- Fig. 14 Graph showing relationship between the inlet height and the maximum velocity in the structure (Thomas S. H., 2018).
- Fig. 15 Spilling occurring due to a large inlet height (Thomas S. H., 2018).
- Table. 2: Results from varied Venturi section experiment (Thomas S. H., 2018).
- Fig. 16 Optimal dimensions for inlet (Thomas S. H., 2018).
- Fig. 17 Full structure (left), Angled view (centre), half-section structure (right) (Thomas S. H., 2018).
- Fig. 18 Stagnation area in central channel indicated by blue/zero velocity flow characteristics (Thomas S. H., 2018).
- Fig. 19 Domain created in ANSYS which contains the half section of the concept design geometry (Thomas S. H., 2018).
- Fig. 20 Basic parameter selections for the simulations (Thomas S. H., 2018).
- Fig. 21 Refinement areas through the internal structure (Thomas S. H., 2018).
- Table. 3: Results from 3D analysis (Thomas S. H., 2018).
- Fig. 22 Graph showing how the inlet velocities and Venturi velocities are related (Thomas S. H., 2018).
- Fig. 23 Ratio of the inlet and Venturi velocities (Thomas S. H., 2018).
- Fig. 24 Spilling occurring at 10m/s (left) and 20m/s (right) (Thomas S. H., 2018).
- Fig. 25 Velocity contours through Venturi section (Thomas S. H., 2018).
- Fig. 26 Cross-section of Venturi section from 3D model (Thomas S. H., 2018).
- Fig. 27 Example of a response-surface (Thomas S. H., 2018).
- Fig. 28 Setup of the Venturi section in the domain read for RSM process. (Thomas S. H., 2018)
- Fig. 29 Distribution of shearing force on domain floor (left), flow conditions through structure (right) (Thomas S. H., 2018).
- Fig. 30 Response surface showing how the inlet and throat radii affect the measured shear force (Thomas S. H., 2018).
- Fig. 31 Final design, sketch and 3D model (Thomas S. H., 2018).
- Table. 4: Results from both the 3D analysis and the RSM further analysis (Thomas S. H., 2018).
- Fig. 32 Graph showing how the inlet velocities and Venturi velocities are related. RSM results seen in orange, original results in blue (Thomas S. H., 2018).

Fig. 33 Ratio of the inlet and Venturi velocities. RSM results seen in orange, original results in blue (Thomas S. H., 2018).

Table. 5: Calculated power outputs of the design concept (Thomas S. H., 2018).

Table of Contents

Declaration of Authenticity.....	i
Abstract.....	ii
List of Illustrations/Tables.....	iii
1. Introduction	1
1.1. Renewable Energy.....	1
1.2. Wind Power.....	1
2. Project Goal.....	2
2.1. Existing Green Buildings and Designs	4
2.2. Initial Concept and Inspiration.....	5
2.3. Importance of High Velocity Flow.....	7
2.4. Conclusion of Initial Concept	8
3. Computational Fluid Dynamics	9
4. Analysis	11
4.1. 2D Analysis	11
4.1.1. Simulation Set-up.....	12
4.1.2. 2D Analysis Experiment Details.....	13
4.1.3. Conclusion of 2D Analysis	17
4.2. 3D Analysis	18
4.2.1. 3D Simulation Set-up	19
4.2.2. 3D Analysis Experiment Details.....	21
4.3. Further Analysis	24
4.3.1. Shape Optimisation.....	25
4.3.2. Response Surface Methodology (RSM).....	26
4.3.3. Further Analysis Experiment Plan	27
5. Discussion of Results.....	33
6. Conclusion.....	35
7. Evaluations and Recommendations for Further Work	37

1. Introduction

1.1. Renewable Energy

Renewable energy sources can provide clean power for the world's societies. Renewable energy refers to an energy source that is sustainable and is one that often produces zero or almost zero pollution. This contrasts with conventional energy resources such as fossil fuels that are a limited resource and when burned for energy, release chemical compounds that alter the Earth's atmosphere negatively (Boyle, 1996).

Renewable energy methods are considered a clean sources of energy. With careful planning and implementation, renewable energy sources can:

- Minimize environmental impacts
- Produce minimum secondary wastes
- Provide sustainable energy based on current and future economic and social societal needs.

“The Sun is the source of all energies. The primary forms of solar energy are heat and light. Sunlight and heat are transformed and absorbed by the environment in a multitude of ways. Some of these transformations result in renewable energy flows such as biomass and wind energy” (Kotharia , Kaushikb, & Panwara, 2010).

1.2. Wind Power

Wind is considered a renewable source. Energy can be generated by converting masses of moving air into rotational shaft by with the use of a wind turbine (Piggot, 2001). Wind energy is an intermittent resource, meaning the power from the wind is not always available.

Regardless, wind power is the fastest growing form of renewable energy. In 2016 Europe invested 27.5 billion euros into wind energy development by installing 12.5GW of new wind power capacity to the grid. That year wind power provided 300TWh of energy to the EU covering 10.4% of the total electrical demand (WindEurope, 2016).

The power available in the wind is heavily influenced by geographical effects such as terrain. Weather patterns in the Northern hemisphere result in the UK having one of the best wind resources in Europe. 12,500sqkm in England and Wales and 37,500sqkm in Scotland have average mean wind speed of over 7.5m/s at an altitude of 25m above ground (Piggot, 2001)

2. Project Outline

This project investigated 'green building' and sustainable design, a practice that is used to increase the efficiency of a useable structure by careful design of how the building manages its resources, waste and net energy consumption (Worldgbc, 2016).

According to (U.S. Energy Information Agency, 2016) commercial and domestic buildings accounted for 39% of total energy use and 68% of electricity consumption in 2016.

This project aims to design and optimise a structural geometry that can capture low velocity winds, redirect and accelerate the flow through a central column, and pass the flow through a wind-turbine for energy generation.

The area that surrounds the inner structure will be used as functional space that will be energy-sustained by the internal generator. The concept is illustrated and detailed in further sections.

The geometrical features of the channels in which the air will flow are key to the performance of this conceptual system and must therefore be carefully designed to maximise its functional ability.

2.1. Project Time Plan

Project Time Table

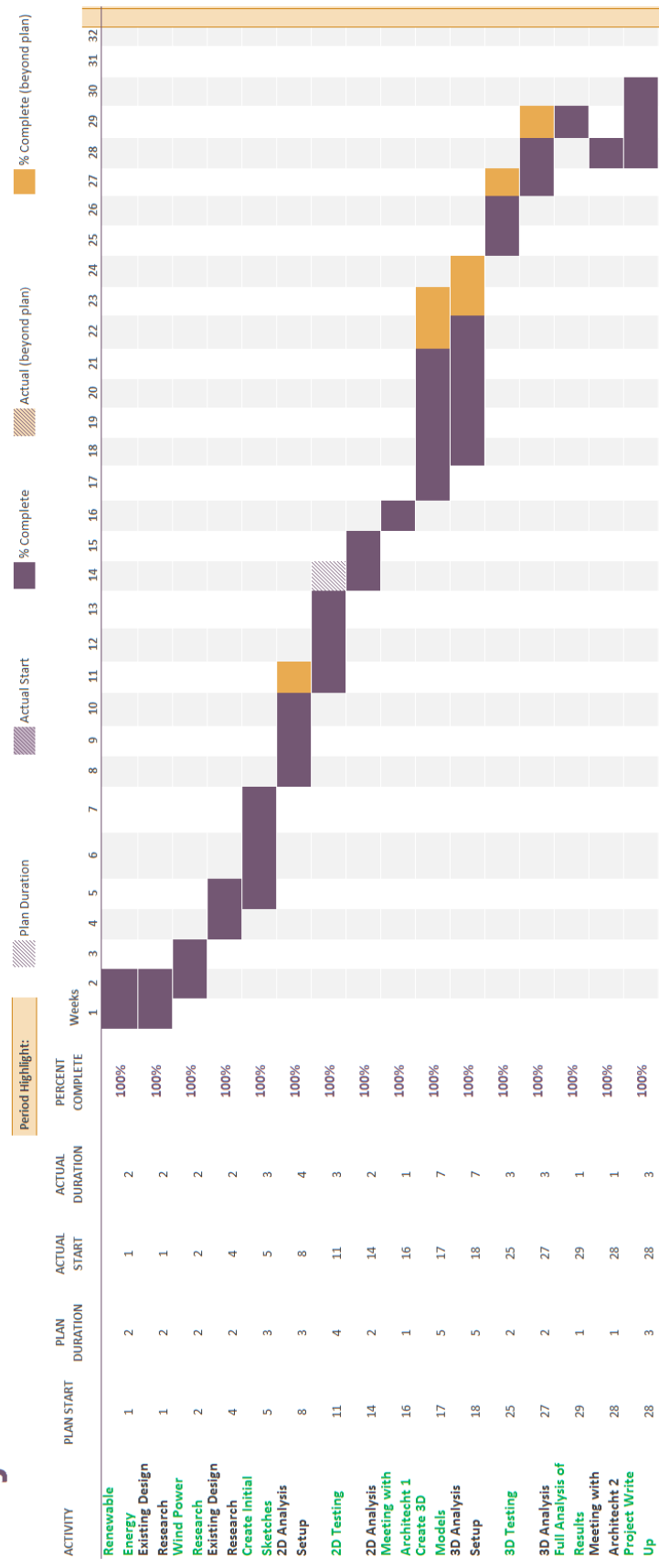


Fig. 1 Gantt chart of project timeline.

2.2. Existing Green Buildings and Designs

Located in the city Manama, the Bahrain World trade centre stands at 787 feet with 50-floors and opened in 2008, the building integrates wind turbines into its design in order to provide 15% of the buildings power consumption. The building can be seen below in Figure 2.



Fig. 2 Bahrain World trade centre.

The towers are linked by three sky bridges that support the three large wind turbines. The wind turbines are roughly 95 feet in diameter and face the Northern prevailing winds. The wind turbines on their own generate 225 kilowatts of power, totalling in 675 kilowatts together.

(SHArchitecture, 2016)

A common concern for the integration of wind turbines into building is the visual intrusion that is involved. A novel solution to this problem comes from Rupert Sweet-Escott and his Secret Energy Turbine (SET). The turbine is disguised to look like a chimney and is placed on top of residential houses to provide green energy for the home.

The turbines can produce up to 250W and handle and is said to “work in near silence because of its unusual design, which features vertically mounted blades fixed under the influence of two opposing magnets” (Dvorak, 2010). These can be seen below in Figure 3.



Fig. 3 The Secret Energy Turbine (SET).

2.3. Initial Concept and Inspiration

The initial concept for the design was based upon finding a solution to decrease the total energy that domestic and commercial buildings draw from the grid with the use of a wind turbine. The turbine would be situated in an optimal position on the building for generating maximum energy.

The initial idea was to create a structure that had a geometry that directed wind over the buildings outer surface and with the use of a 'backward-facing step', induce strong flow concentrations that would turn the turbine. The concept is illustrated below in Figure 4.

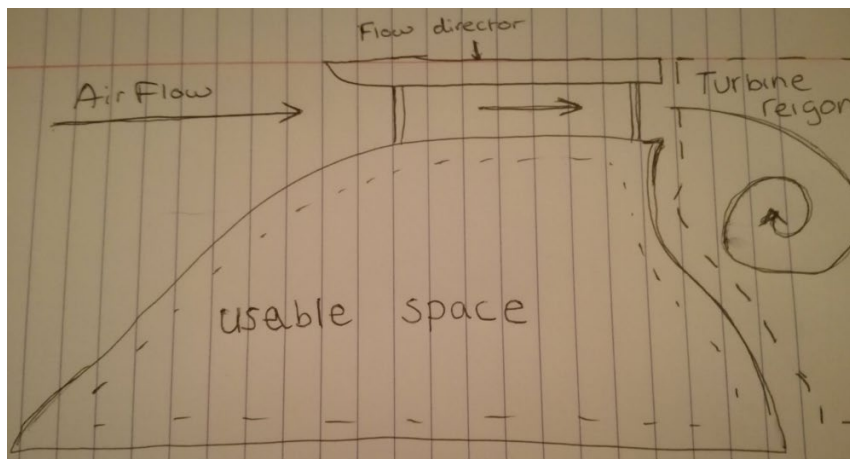


Fig. 4 Sketch of initial design.

During the early development stages, a few concerns were raised regarding the practicality of the design. The first of which was that the building would have a relatively large footprint in order to efficiently channel wind to the rear end of the building.

The most concerning issue was that the system was 'unidirectional', meaning it would only function adequately when the wind was approaching from the same direction as the structures channels. Efforts could be made to position the buildings facing the areas prevailing winds but if the wind is approaching from any other direction, the structure may not be able to function.

The latter of the two main concerns lead to added criterion for the design that the structure is to be able to 'catch' and direct wind from any direction.

This lead to the idea of simply revolving the geometry from the initial design (Fig.4) so that an inlet was formed that could take in wind from a 360-degree range. The result of this revolution was that a central column or duct was formed that ran vertically through the centre of the structure.

Research was conducted into energy generation systems that utilise similar designs.

An existing design was found that closely resembled the concept of this project, the design was known as an 'axial-flow turbine' and can be seen in Figure 5.

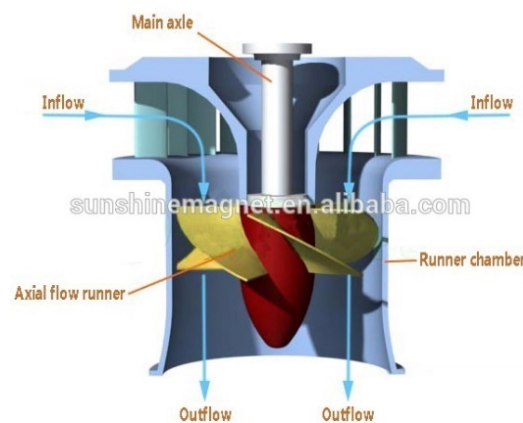


Fig. 5 Axial-flow turbine, half section view.

The axial-flow turbine is a type of hydro-power turbine, water enters the system through the inflow areas, flows over the axial turbine and the fin shape produces a rotational force to turn the turbine shaft (Helmy, 2003).

This design proved the concept of directing fluid flow into a vertical duct (runner chamber) was feasible.

The second existing design was the 'bulb turbine', depicted in Figure 6. Bulb turbines are usually positioned on shores near the tide or downstream from a water source where water can be forced through a channel structure and pass the flow through an axial turbine (GE Renewable Energy, 2018).

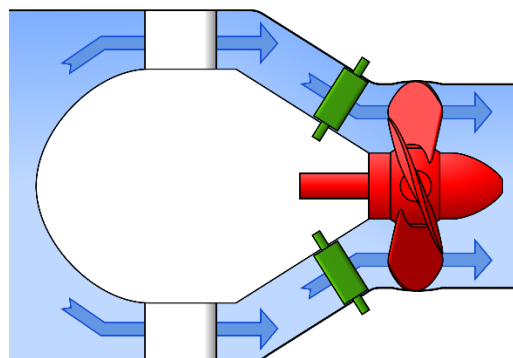


Fig. 6 Bulb turbine with flow channels depicted.

The research into the bulb turbine also lead to the inclusion of narrowed section in the flow channel. The narrowed section will produce an increased flow velocity in that area according the Bernoulli's' continuity equation which shows as the cross-sectional area of a flow channel is decreased, the flow velocity will increase in order to maintain an equal volume flow rate (Hall & Keynes, 1974). This is also known as the Venturi effect, demonstrated in Figure 7.

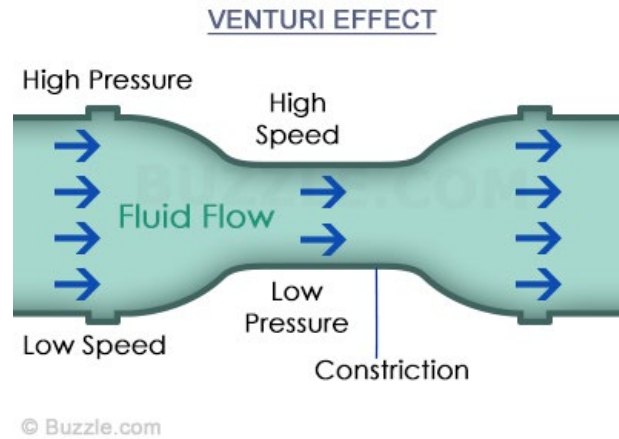


Fig. 7 The Venturi effect within a pipe.

It was theorised that, for the concept design, the turbine could be placed within this narrowed section to take advantage of the accelerated flow at that point and maximise the power output of the system.

2.4. Importance of High Velocity Flow

It is important to achieve high wind velocities as it is the most influential factor to consider when generating power from wind.

$$P_w = \frac{1}{2} \rho A v^3$$

The equation for the power available from wind is given as

Where:

P_w = power

ρ= air density

A = area swept by turbine blades

V = velocity

(Sarkar and Behera 2012)

From the equation it is seen that if the velocity is doubled the power output increases by a factor of eight (Piggot 2001).

Therefore, if the design could take in wind traveling at a given velocity through its inlets and accelerate it to twice the initial speed through the narrow section in the vertical column, then the

turbine will produce eight times as much power than it would if it was simply placed outside of the design in the same conditions.

Alternatively, to produce an equivalent power output the turbine could be placed in the open environment, but would have to increase in size to sweep an area eight times greater than the one seen in the design. Not only would this increase the amount of space needed to accommodate the turbine but due to its greater weight would also increase the cut-in speed of the turbine (Gitano-Briggs, 2012).

2.5. Conclusion of Initial Concept

The aforementioned design principles formed the basic layout for the fully-formed initial design. It was noted that the wind turbine would be positioned somewhere along the central duct and that an exhaust system would need to be implemented to return air to the surroundings

A temporary lower limit of 800mm diameter was set for the narrow section where the turbine would be positioned and was later assessed in the conclusion section to determine whether the diameter could be decreased further. As the swept area of the turbine is made smaller it becomes difficult to generate the power to meet the daily average demand of 12KWh (WindExchange, 2018). According to the wind power equation the smaller the turbine area, the less power it can generate.

The design can be seen in Figure 8 as a 2D cross-sectional sketch.

3. Computational Fluid Dynamics

The testing and analysis process relied on simulating wind flow conditions around and through the body of the concept design. This was achieved using an analysis tool called computational fluid dynamics (CFD).

CFD uses finite elements that tie together the properties and state of fluid elements inside of solid boundary condition elements (Ferziger and Peric 2002).

To model the flow conditions as accurately as possible an appropriate turbulence model was used.

“Turbulent flow is a type of fluid (gas or liquid) flow in which the fluid undergoes irregular fluctuations, or mixing, in contrast to laminar flow, in which the fluid moves in smooth paths or layers” (Encyclopedia Britannica, 2006).

Turbulent flow is inherently difficult to model due to its random nature and therefore numerical solutions are used to predict this flow but can never achieve complete accuracy.

RANS Models

The Numerical method used for the simulations in this project uses the Reynolds-averaged Navier-Stokes (RANS) approach. This method relies on averaging the equations for the motion of flow over time and over a coordinate in the domain in which the flow does not vary. This method requires approximations and therefore a turbulence model must be chosen. RANS models are beneficial to use when just a few quantitative properties of turbulent flow need to be determined (Ferziger & Peric, 2002).

Turbulence Models

The RANS method leads to a set of open ended equations known as the RANS equations. To close these equations and attain a solution a turbulence model must be selected. The turbulence model aims to predict energy dissipation, momentum and transportation of mass. (Ferziger & Peric, 2002) States that all these flow qualities are mediated by the fluid viscosity and can therefore be represented as an increased viscosity within the model.

The specific turbulence model chosen for the simulations was the K-epsilon model. This model provides the equations that ‘close’ the RANS equations, the equations in the K-epsilon model solve for both turbulent viscosity and the dissipation rate of kinetic energy. In particular the Realisable K-epsilon model was used for its greater capacity for solving separated flows that occur far from the boundary wall (Cable, 2009).

Flows were expected to be separated as the geometry of the structure alters the path of the flow to a large degree.

Wall Functions

In turbulent conditions the viscous layer of fluid that resides near the geometry surface becomes so thin that it is difficult to use enough mesh elements to resolve it. This problem can be avoided by using wall functions that rely on a logarithmic region which is incorporated into the velocity profile equation of the turbulence model (Ferziger & Peric, 2002).

The specific wall function chosen for the simulations was the Non-Equilibrium wall function. This function was paired with the turbulence model for its capabilities for solving highly separated and stagnated flows (Nazif & Tabrizib, 2013).

Certain areas within the concept design were predicted to cause flow stagnation, the most prominent areas being the curved surface of the inlet area and the base of the exhaust section.

4. Methodology & Analysis

Once the initial concept was fully established, a preliminary sketch was created in Solidworks (3D modelling software). The sketch shows a cross-section of the inner flow chambers of the design. This sketch provided a starting point for the analysis process and was to be altered according to the findings of the CFD simulations.

The Initial dimension of the sketch were based upon those of two-story large scale structures like schools (each story is roughly 4.6m) (OECD, 2009). The height of the design from ground level to the base of the inlet was initially 10m and the width of the structure was limited to a 5m radius. The optimal dimensions for the inlet and the central column, including the Venturi section were determined through-out the analysis process.

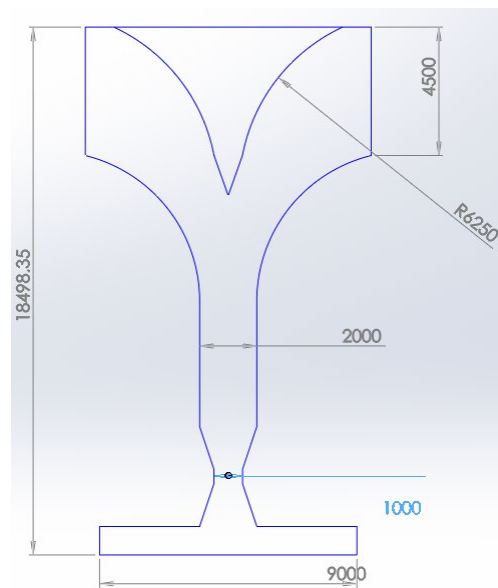


Fig. 8 2D sketch of concept design internal flow channels.

4.1. 2D Analysis

Using the 'surface' feature in Solidworks, a 2-dimensional plane was generated from the sketch and imported into ANSYS (chosen CFD software) ready for the initial fluid-dynamic simulations. 2D analysis allows for rapid simulation processing and can help develop an understanding of how the geometrical features of the design are affecting the wind-flow through the inner flow chambers.

The main goal of the 2D analysis was to determine optimal dimensions for both the height of the inlet and the radius of curvature for the upper surface of the inlet. These can be seen in Figure 9.

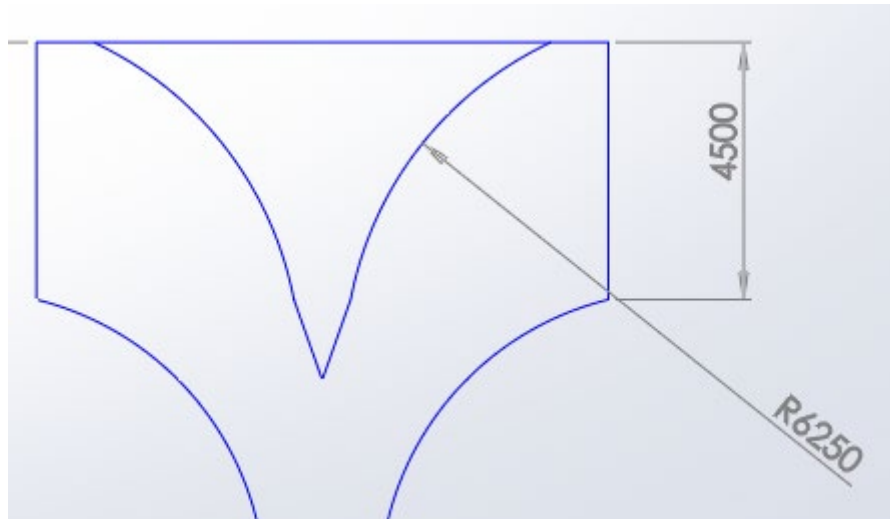


Fig. 9 Inlet height and inlet radius.

4.1.1. Simulation Set-up

Once imported to ANSYS the 2D plane surface was meshed appropriately. Each edge of the plane surface was selected and split up into small subsections, the subsection lines ran from the edge and over the plane surface. Where the subsection lines intersected, an element was formed and creates a surface of finite element mesh as seen in Figure 10.

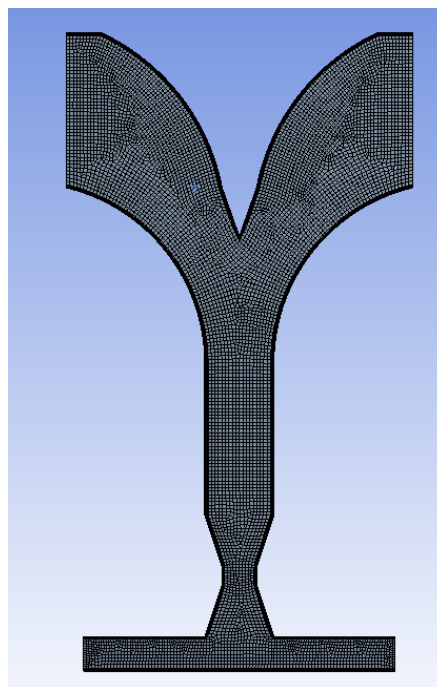


Fig. 10 2D plane surface, meshed for simulation.

A fine mesh was used to capture accurate flows with particular focus and finer mesh at the inlet area and Venturi-section. This was necessary to generate reliable results for the inlet optimisation and to also accurately measure the maximum wind velocity through the Venturi-section.

Each edge of the plane surface was given a named selection which will correspond to its boundary conditions in the simulation.

The left inlet edge (Fig. 8) is chosen as the velocity-inlet which simulated wind flow entering the inner structure from the left direction at a chosen velocity.

The right inlet edge and both exhaust edges were chosen as pressure outlets. This is where the flow can leave the system.

All other edges were chosen as walls which contains the flow until it reaches an outlet.

An inflation layer was added to all edges excluding the inlet and outlets to accurately solve the flow conditions at the boundary layer where the flow directly contacts the geometry. These layers can be seen running along each edge in Figure 11 below.

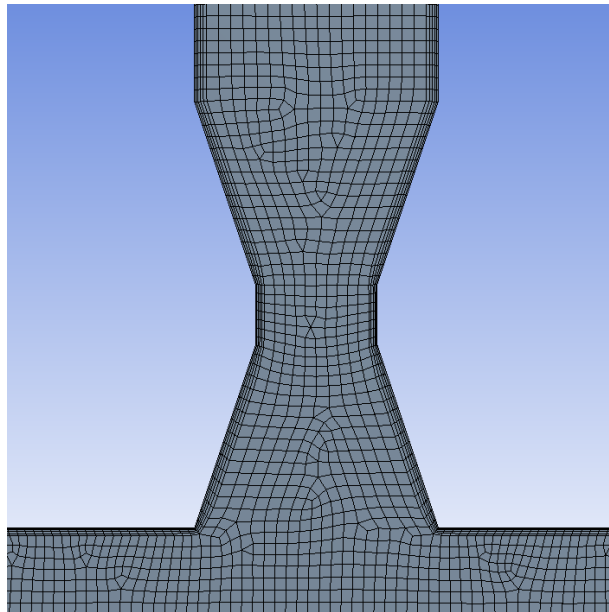


Fig. 11 Close up of inflation layers in the Venturi section.

In the final step, values were selected for the boundary conditions and the appropriate solution methods for the simulation were chosen. The chosen solution method used the Realisable K-epsilon turbulence model with Non-Equilibrium wall functions. For further detail on these selections refer to the 'Turbulence Model' section in chapter 3.

A wind velocity of 4m/s was chosen for the inlet velocity which is an estimate supported by average annual wind speeds for the UK of 7.5m/s at a 25m altitude (Piggot, 2001).

Each pressure outlet is set to atmospheric pressure to mimic real-life operating conditions.

4.1.2. 2D Analysis Experiment Details

For the first set of simulations the inlet radius was fixed and the inlet height was varied. Once all inlet heights were simulated and recorded the inlet radius was increased and again the inlet heights were varied.

This continued in incremental steps until an optimised solution was concluded in the data.

The goal of each geometrical feature of the structure is to achieve the highest wind velocities through the system, therefore the key variable to measure in these simulations is the maximum wind velocity.

Table. 1: Results from 2D analysis.

Inlet Height (m)	Max Velocity (m/s) Radius = 5m	Max Velocity (m/s) Radius = 6m	Max Velocity (m/s) Radius = 8m	Max Velocity (m/s) Radius = 10m	Max Velocity (m/s) Radius = 12m
1.5	6.21	7.32	6.48	5.73	5.29
2.5	9.56	10.8	9.35	8.03	7.38
3.5	12.8	14.7	11.5	9.91	9.09
4.5	13.8	16.7	12.7	10.6	9.68
5.5	15.1	17.8	13.3	11	9.91
6.5	15.9	18.1	13.3	10.9	9.45

Table 1 contains the recorded values for the maximum wind velocity measured inside the structure for a given inlet height and corresponding inlet radius.

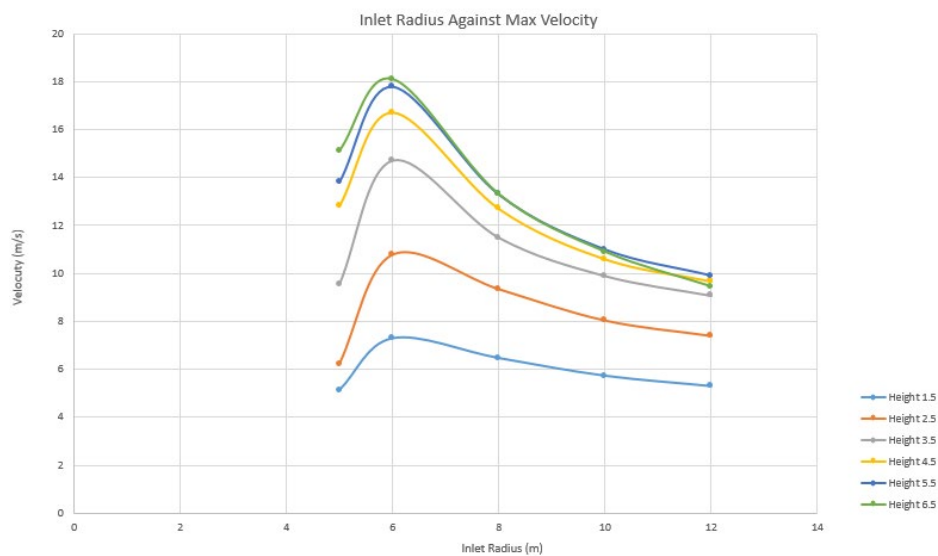


Fig. 12 Graph showing the relationship between the inlet radius and the maximum velocity in the structure.

In Figure 12 Each separate line represents a given inlet height. From this graph was clear to conclude the optimal inlet radius was 6m at every inlet height. The reason for this is that the 6m radius conformed the best to the overall structure, creating a steady pathway that redirected the flow downwards towards the Venturi-section.

At 5m radius the curved pathway redirected and reversed the flow back out of the structure. At radii higher than 6m a phenomenon which will be referred to as 'spilling' occurred. This happens when the curved pathway does not provide enough of a downward velocity component and thus the flow that entered the structure is free to exit out the opposite side. This can be seen in Fig 13.

All Figures of the 2D surface are coloured in relation to the flow velocity, low velocities are coloured blue/green and high velocities are coloured yellow/red.

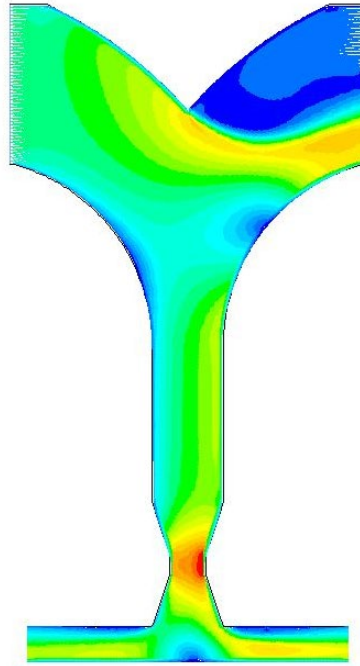


Fig. 13 Spilling occurring due to a large inlet radius.

Here in Figure 13, the velocity inlet where the flow enters is on the left side, the radius of curvature is 12m, a very shallow radius that was not able to direct the flow downwards and as such the flow spills through the right side of the structure. This is a highly inefficient design as the mass of the flow that is spilled could have been used for power generation.

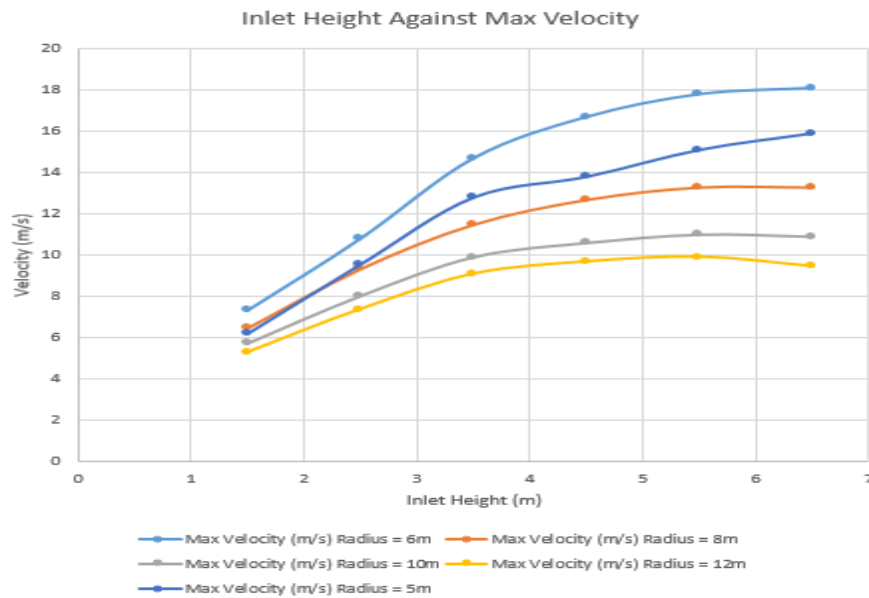


Fig. 14 Graph showing relationship between the inlet height and the maximum velocity in the structure.

Figure 14 shows the relationship between the inlet height and the max velocity in the structure. Each line represents a given inlet radius.

Here again it is seen that the inlet radius of 6m provides the highest velocities through all heights. As the inlet height is increased there is an increase in max velocity. This is a logical result as when the inlet height is increased the total volume of air entering the structure will increase, when that volume of air is constricted at the Venturi-section it will be forced to increase its velocity to maintain the total volume flow rate of the system (Hall & Keynes, 1974).

Above heights of 4.5m 'spilling' was again observed, due to the increasing area of the flow path the flow becomes less confined and escapes through the opposite side.

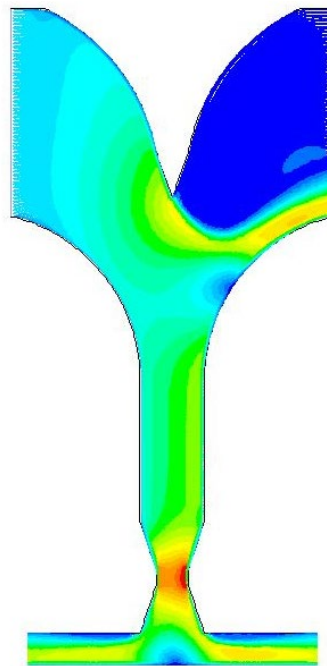


Fig.15 Spilling occurring due to a large inlet height.

Figure 15 shows the experiment with a 6.5m inlet height and a 6m radius curvature. It was seen that at this height, even when the radius was optimal for directing flow downward, spilling still occurred. As the inlet height was increased the available path that would cause the flow to spill became larger.

In all cases max velocity was seen at the Venturi-section as expected.

A further, small experiment was conducted to determine if the position of the venture section along the vertical column had any effect on the max velocity.

The 2D surface with inlet height 5m and inlet radius 6m was chosen and the position of the Venturi section was varied in 3 position (top, middle, and bottom).

Table. 2: Results from varied Venturi section experiment.

Venturi Position	Max Velocity (m/s)
Top	16.7
Middle	16.2
Bottom	16.2

The results from the experiment seen in Table 2 concluded no significant change in max velocity, only a slight increase when the bottom position is used. The Venturi section was therefore kept at the bottom of the vertical column.

4.1.3. Conclusion of 2D Analysis

The results of the 2D analysis led to the selection of an inlet height of 5m and an inlet radius of 6m. The 6m radius was chosen for its optimal ability to redirect the flow and the 5m inlet height was able to capture large amounts of wind while minimizing the flow lost to spilling. This is seen in Figure 16.

It is critical to ensure the spilling was minimised at this stage as the effect was predicted to amplify in the 3D environment due to a large range of freedom of the flow path

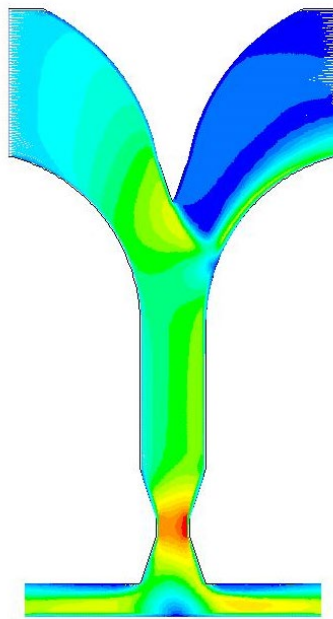


Fig. 16 Optimal dimensions for inlet.

It is important to consider the limitations and assumptions that are inherent with 2D analysis.

The greatest limitation was that it only simulates flow through the single plane, this is unrealistic as complex flow interactions will occur from various angles in a 3D environment (Arici & Özel, 2012). The 2D set-up assumes that the total amount of wind flow will enter the system perfectly. In reality certain amounts of the flow will pass around the inlet structure and can be rejected from entering the system due to the limited capacity of the structure.

With these factors taken in to consideration it was concluded that for this experiment the 2D analysis is a highly idealised situation and the figures from the data should be considered over-estimated.

The concluded optimised dimensions still hold their credibility but max wind velocities are expected to be significantly lower in the 3D simulations.

4.3. 3D Analysis

The Solidworks 'revolve' feature was used on the 2D sketch to create a 3D solid body with the newly optimised inlet dimensions.

The 3D body was 'shelled' to create the flow pathways and a wall thickness of 50mm was chosen. This thickness can be changed according to building standards and necessary structural support. The 50mm thickness was chosen because a lesser thickness would create errors in the meshing process.

Two necessary features were then added to the 3D model:

1. A set of four guiding fins positioned equally around the inlet circumference. These will confine the flow and channel it towards the central column.
2. Exhaust vents positioned at the base to return the flow to the surrounding area.

These features can be seen at the top (fins) and bottom (exhaust) of the structure illustrated in Figure 17.

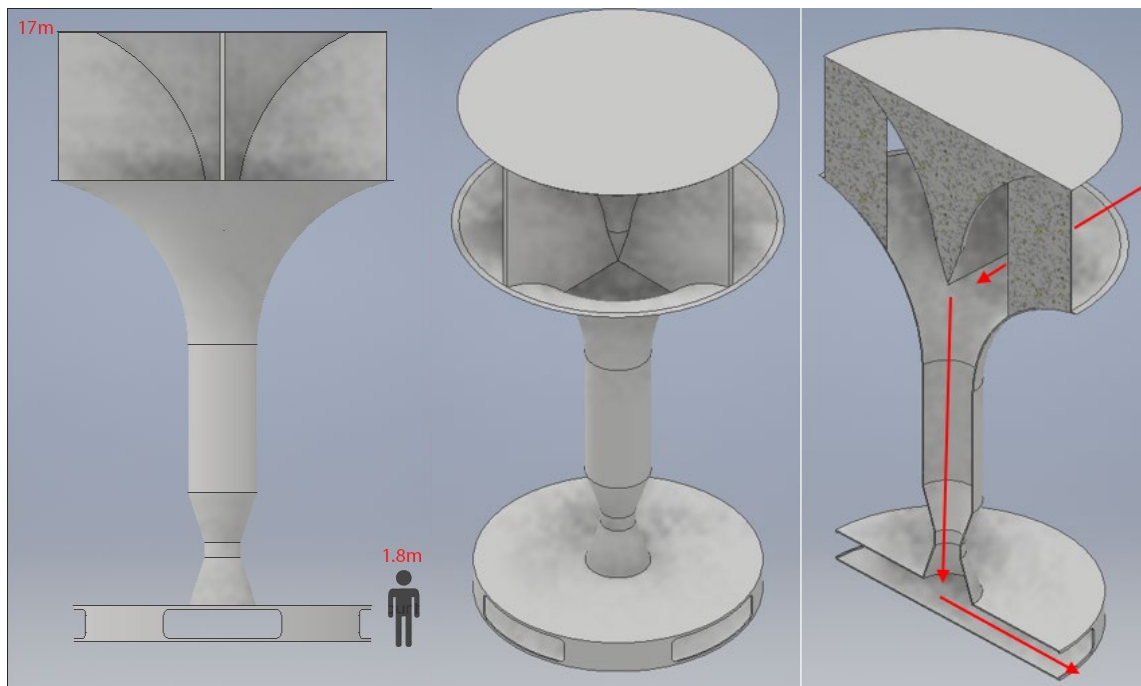


Fig. 17 Full structure (left), Angled view (centre), half-section structure (right).

Not illustrated in Figure 17 is the inclusion of an angled guiding fin that runs along the circumference of the inlet. This feature was added to direct more flow into the inlet.

Also, during the initial simulation run on this model it was observed that for the 3D model, the entrance of the venture section was too low and a stagnation point formed before reaching the Venturi section. This can be seen in Figure 18, therefore the venture section was extended upwards to begin the acceleration process earlier in the structure. The extended section is visible in Figure 18.

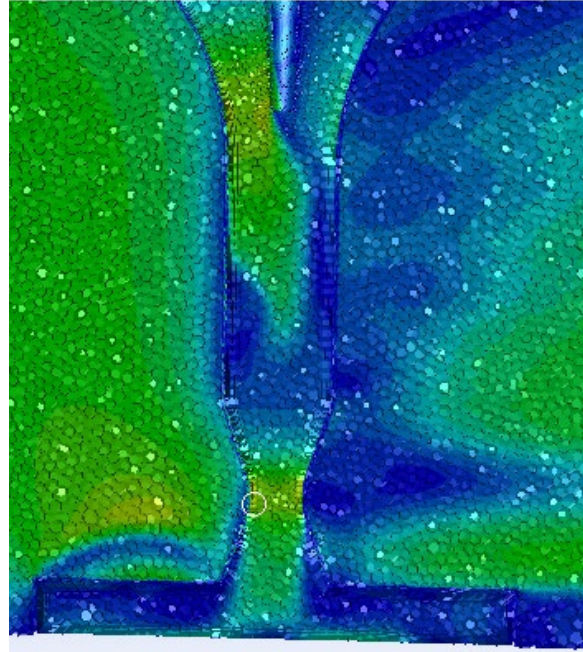


Fig. 18 Stagnation area in central channel indicated by blue/zero velocity flow characteristics.

4.3.1. 3D Simulation Set-up

For the 3D analysis the model was first inserted into a domain that represents the area in which it will operate.

Once the correct domain was generated the geometry of the structure was subtracted from the geometry of the domain using the 'Boolean' feature. This leaves only one solid body that contains all the possible pathways that the flow can take, including the pathway through the structure. Only half of the full geometry was modelled along the symmetry wall in order to reduce the amount of calculations needed for the solution and decrease the running time of the simulation (Kaiser, Friedewald, & Nitzsche, 2017). The result of this process can be seen in Figure 19.

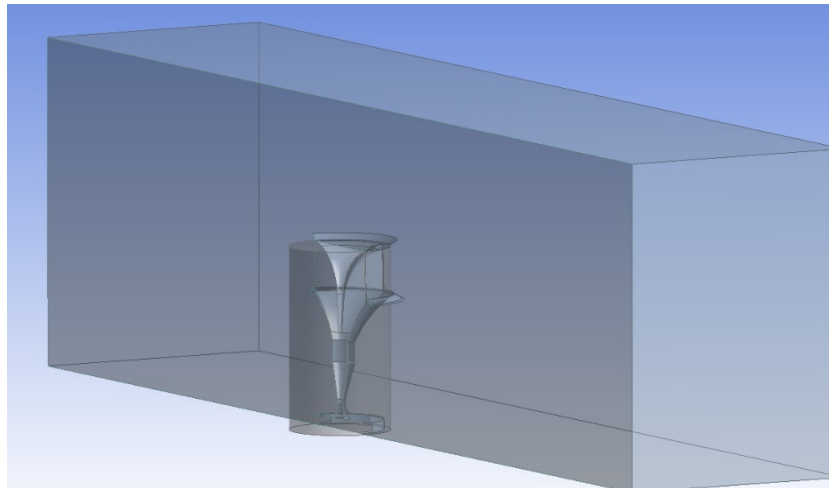


Fig.19 Domain created in ANSYS which contains the half section of the concept design geometry.

Next, the domain was meshed. The meshing parameters for this model were based upon an investigation into ANSYS meshing parameters conducted by (Thomas, 2018) where air flow was simulated over a bluff body and measurements of drag and lift from the simulation were compared to experimental data. The parameter selections for the simulations in this project can be seen in Figure 20.

This model achieved a percentage error of only 2.7% showing that it is capable of accurately solving flow characteristics.

Although the geometries being investigated were different, they are both stationary bodies with simple geometries and are subject to relatively low wind velocities.

Solver Preference	Fluent
☐ Relevance	0
Export Format	Standard
Shape Checking	CFD
Target Skewness	Program Controlled
Element Midside Nodes	Dropped
Sizing	
Size Function	Proximity and Curvature
Relevance Center	Fine
Initial Size Seed	Active Assembly
Smoothing	High
Transition	Slow
Span Angle Center	Fine
☐ Curvature Normal A...	12.0 °
☐ Num Cells Across G...	Default (3)
Proximity Size Functio...	Faces and Edges
☐ Min Size	Default (1.8734e-002 m)
☐ Proximity Min Size	Default (1.8734e-002 m)
☐ Max Face Size	2.0 m
☐ Max Tet Size	2.0 m
☐ Growth Rate	Default (1.20)
Automatic Mesh Base...	On
☐ Defeature Size	Default (9.367e-003 m)
Minimum Edge Length	3.3705e-002 m

Fig.20 Basic parameter selections for the simulations.

Refinement areas (areas with increased mesh density) were added inside and around the structure body to capture more detail of flow behaviour and solve the flow more accurately around key geometry features. These are seen below in Figure 21.

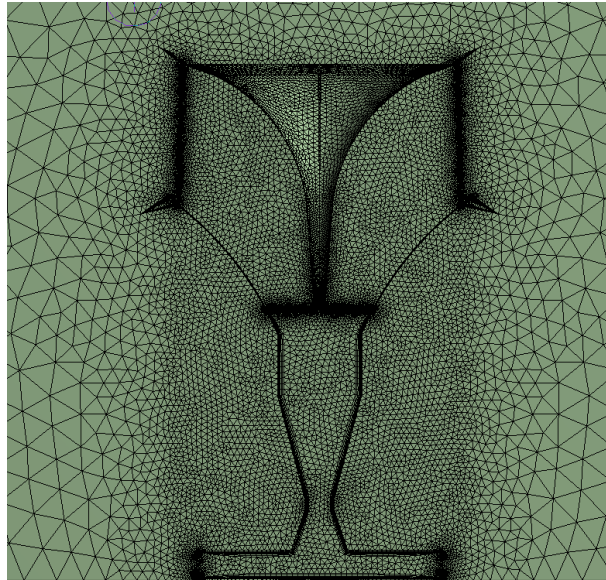


Fig. 21 Refinement areas through the internal structure.

Similar to the 2D process, named selections were created to describe the boundary conditions of the domain.

In this case one face of the domain was selected as the velocity inlet and the opposite face as the pressure outlet.

The face that has the structure body impressed into it was selected as a 'symmetry' wall. This functioned to mirror the structure geometry and the flow conditions to the opposite side. This can only be done as the structure is symmetrical about its central axis. This reduces the amount of calculation in the simulation leading to faster solve times (Kaiser, Friedewald, & Nitzsche, 2017).

All other faces were automatically selected as 'walls' which tells the model to treat these surfaces as solids that fluid cannot pass through but can interact an 'slip' over (Ferziger & Peric, 2002).

4.3.2. 3D Analysis Experiment Details

Simulations were run on the model, varying the inlet velocity to determine how the structure will perform through a range of wind speeds. The velocities through the Venturi-section were then measured in the post processing package of ANSYS by taking measurements at three positions (far right, far left and middle) in the Venturi section and averaging the measurements.

Table. 3: Results from 3D analysis.

Inlet-velocity	Venturi-Velocity	Inlet-Venturi Ratio
2	3.75	1.88
4	8.25	2.06
6	11.7	1.95
10	16	1.6
20	27	1.35
40	54	1.35

Table 3 contains the varied inlet velocities and their corresponding Venturi velocities measure through the venture section and also notes the ratio between these the initial and measured velocities.

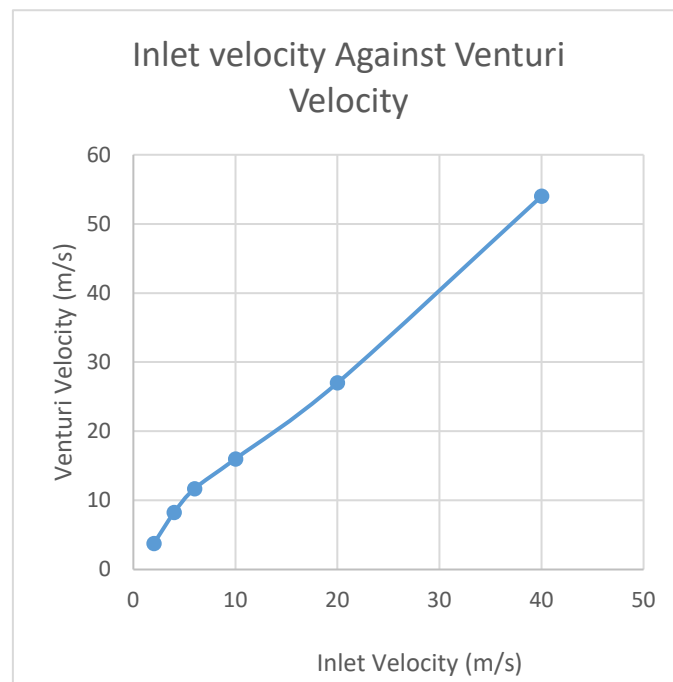


Fig. 22 Graph showing how the inlet velocities and Venturi velocities are related.

In Figure 22, the graph shows a fairly linear relationship between the inlet velocity and the max velocity seen through the Venturi-section. This demonstrates that the structure is capable of performing its main task at higher velocities and, importantly, at low velocities.

The degree to which the structure accelerates the flow varies through the range of wind speeds. The following graph in Figure 23 shows the Ratio between the inlet velocity and Venturi velocity, ranging from 1.35-2.06.

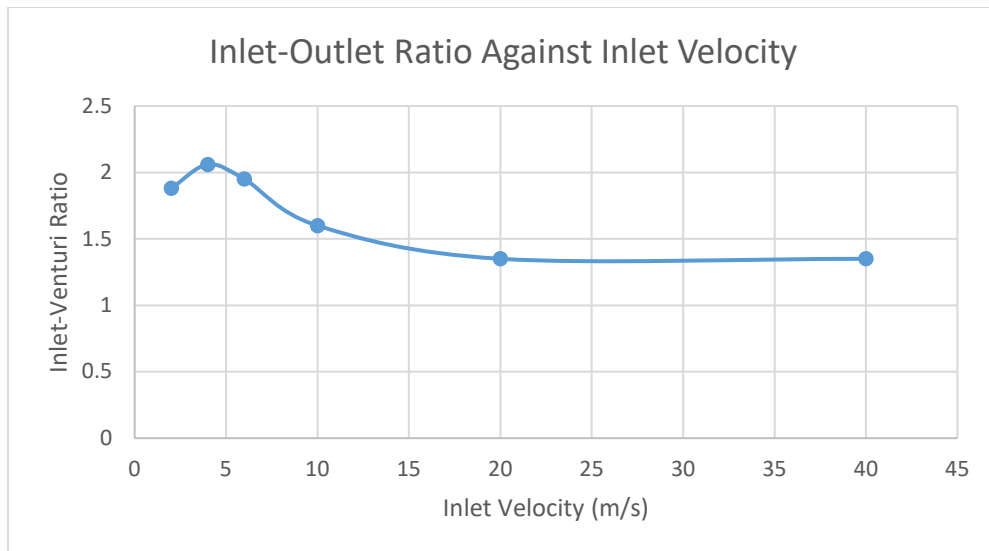


Fig. 23 Ratio of the inlet and Venturi velocities.

The ratio can also be thought of as the amplification factor. Peak amplification is seen at 4m/s inlet and tapers off at an amplification of 1.35 as the inlet velocity increases above 20m/s.

With the peak amplification occurring at 4m/s, two main conclusions can be suggested:

One, the 2D analysis was successful in optimising the inlet dimension as a 4m/s inlet velocity was used to drive the process. This also strengthens the validity of the 3D model used in the simulations as the model produced data that is clearly representing key features that were present in the 2D setup.

Two, the success of the 2D optimisation suggests that the same process could be used to optimise the inlet dimensions using any given inlet speed. This could mean that structure could be optimised for a chosen wind speed that corresponds to the average wind speeds in any given location.

The results also show the structures ability to accelerate low velocity winds. At an inlet speed of just 2m/s the structure was able to produce a 3.75m/s flow through the venture section. This is advantageous as wind turbine have a 'cut-in' speed which is the minimum wind speed a turbine needs to start generation power, this cut-in speed is typically 3-4m/s (BWEA, 2005). Theoretically the structure could provide energy generation on low wind days or in areas of low annual wind speeds.

The reason for the drop-off in the amplification factor is due to the increased amounts of spilling seen at higher inlet velocities. This was predicted based upon two assumptions:

The structure has a limited volume and therefore will have a maximum volume flow rate it can handle before it starts to reject the inlet winds.

The higher kinetic energy of the inlet wind allows the flow to overcome the directing forces imparted on it by the structure and it is easier for the flow to pass through and spill out the opposite side. This can be seen in Figure 24 which compares the amount of spilling at two different inlet velocities.

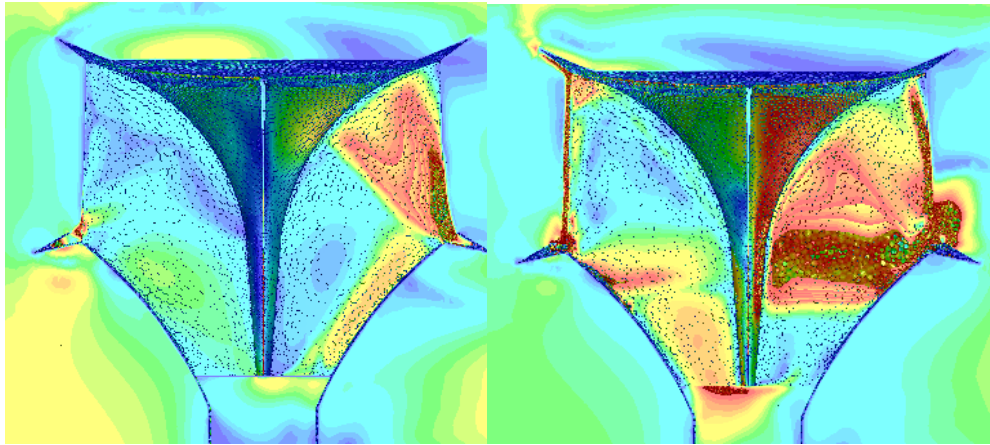


Fig. 24 Spilling occurring at 10m/s (left) and 20m/s (right).

Testing was concluded after the 40m/s simulation, the data shows that the structure can perform at higher wind velocities, however, one must account for the ‘cut-off’ speed of a wind turbine. This is the rated speed at which the turbine is shut off to protect the turbine and its components from damage (BWEA, 2005).

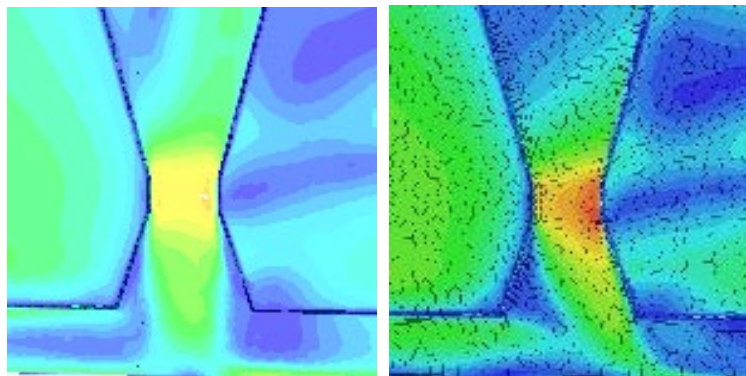


Fig. 25 Velocity contours through Venturi section.

Figure 25 shows the flow conditions through the Venturi section at inlet velocity of 4m/s (left) and 20m/s (right).

4.4. Further Analysis

An experiment was conducted into the optimisation of the Venturi section of the structure. Its purpose is to constrict the area of the flow-path and thus increase the flow velocity according to Bernoulli's Principle (Gorse 2012).

The success of this analysis was measured by assessing the ability of the Venturi section to accelerate the wind. A fully optimised Venturi section will produce the highest internal wind velocities from a range of ambient wind velocities. It is important to achieve high wind velocities as it is the most influential factor to consider when generating power from wind.

Compared to other features of the structure, the Venturi section was considered relatively complex (see Figure 26) as it has several features, all interdependent, that combine to produce a certain performance output. The features that affect the output are:

1. Diameter of inlet
2. Diameter of Throat
3. Diameter of divergent outlet
4. Rate of taper of convergent entrance

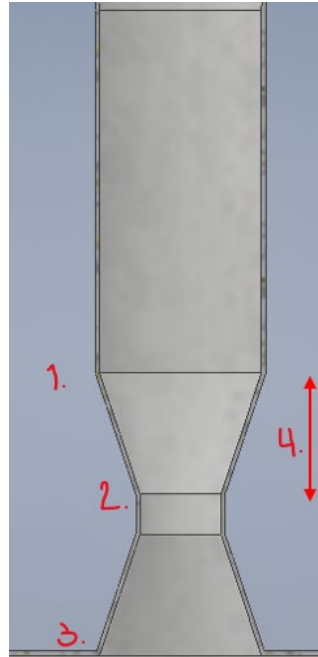


Fig. 26 Cross-section of Venturi section from 3D model.

If the same approach were used to optimise the Venturi section as the inlet, one would have to vary each of the dimensions in respect to each other and collect the results through a large number of simulations.

Instead, a specialised shape optimisation tool was used to quickly arrive at a solution for the optimum layout of dimensions.

4.4.1. Shape Optimisation

‘The purpose of shape optimisation is to find the optimal lay-out of a structure within a specified region’ (Bendose and Sigmund 2004). Shape optimisation has a range of practical applications, but each case will follow a similar process.

To create the lay-out of a shape optimization problem, first define:

Input variables - define the possible configurations of the design and may consist of any physical dimensions, loading and material coefficients that can vary in the design and manufacture of the part. The optimal design will consist of the best possible combination of design variables that meet the design goals.

State Variables - represent the structural response which can for example be recognized as stress, strain or displacement. Furthermore, the state variables depend on the design variables. These variables will be measured and will ‘steer’ the optimisation to a solution.

The Objective - represents an objective function that could either be minimized or maximized. A typical objective could be the weight, stiffness or volume of a component. When the analysis is run, an optimization algorithm samples design points within the range of the input variables, searching for the combination that achieves the optimal design.

(Larsson 2016) (Cunningham 2014).

For optimising the Venturi section, the lay-out was as follows:

- **The objective** - To maximise the wind velocity through the throat of the Venturi section.
- **Input variables** - The initial dimensions for the vertical column have been selected, the region describe by these dimensions will act as the limited area in which the geometry of the Venturi section can be altered.
- **State variable** – A temporary plane surface was introduced to the model that was positioned at an angle to the direction of flow at the exit of the throat section. The state variable can be set as the shearing force that is seen on this plane surface and therefore maximum pressure will correspond to a maximum velocity through the throat (see Figure 28).

This information will form the structure for the software to reach an optimised solution.

4.4.2. Response Surface Methodology (RSM)

Response surface methodology (RSM) is a collection of mathematical and statistical techniques used for model building. By careful design of experiments, the objective is to optimize a response (output variable) which is influenced by several independent variables (Akbarian and Morshedi 2014).

The basic concept of RSM is as follows:

- Find a suitable approximation for $y = f(x)$ using a low order polynomial
- Move towards the region of the optimum
- When curvature is found find a new approximation for $y = f(x)$ {generally a higher order polynomial} and perform the “Response Surface Analysis” (Wang 2015).

The final function can be expressed mathematically as $Y = f(x_1, x_2, x_3...) + e$. Where ‘Y’ is the output variable and is a function of the independent variables ($x_1, x_2, x_3...$) and ‘e’ represents other sources of uncertainty not accounted for in the function, turbulence or thermal factors for example (Carley, Kamneva and Remi 2004). The response surface can then be plotted to visualise the data.

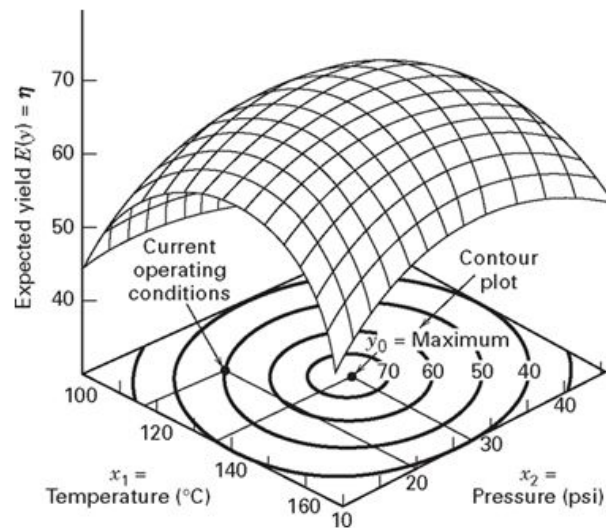


Fig. 27 Example of a response-surface.

Figure 27 shows an example of a response surface and how the independent variables; temperature and pressure will affect the output variable of yield value for a given material.

The RSM tool can then select optimum values of the independent variables that either maximise or minimise the output variable depending on what is needed for a solution.

A similar approach to RSM was used to obtain solutions to design analysis problems seen in (Raoufi, et al. 2008) and (Ortega-Casanova 2017). Both utilise RSM to analyse the effect a geometry has on internal flow and both projects had success in achieving an optimised design. This was a strong indication that RSM is an appropriate choice for optimising the Venturi section geometry.

(Thomas, 2018)

4.4.3. Further Analysis Experiment Plan

Using ANSYS SpaceClaim, the individual dimensions of the Venturi section were selected from the 3D model, this included the inlet radius and throat radius. Each of the dimensions were converted into 'driving dimensions' in a process known as parameterisation. This process allowed the RSM tool to manipulate the value of each of these dimension so they could be varied through a set of simulations.

The Venturi section was isolated and positioned in a new domain, this was done so that a state variable could be assigned for the RSM process. The Venturi section was angled at 45 degrees to the domain floor (see Figure 28), this created a shear force on the floor from the flow exiting the outlet. The shearing force was selected as the state variable which was used to drive the RSM process.

By telling the RSM program to maximise this state variable the dimensions of the Venturi section was optimised for producing the highest possible shear on the floor, the highest pressure seen on the floor will directly correspond to the highest flow velocity through the Venturi section.

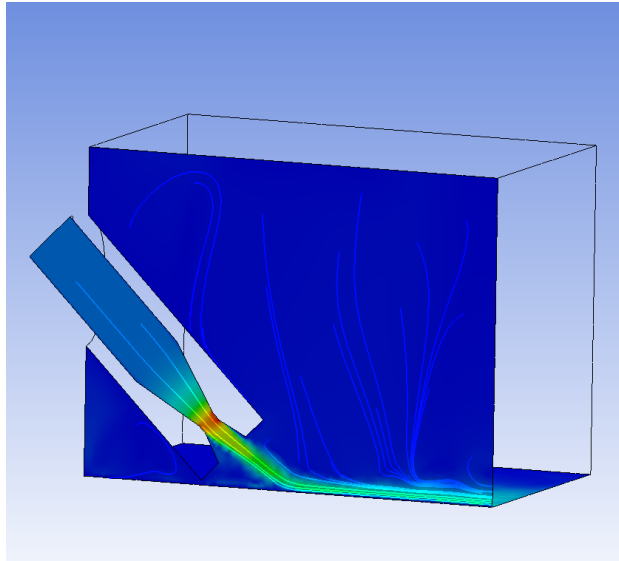


Fig. 28 Setup of the Venturi section in the domain read for RSM process.

The initial simulation was run to obtain a reference point and was identical to the simulations seen in the 3D analysis with the flow conditions being solved through the structure. These flow conditions formed the base of the RSM process.

The flow conditions were analysed in ANSYS post processing, here it was possible to view the pressure that the flow generated on the surface of the base wall.

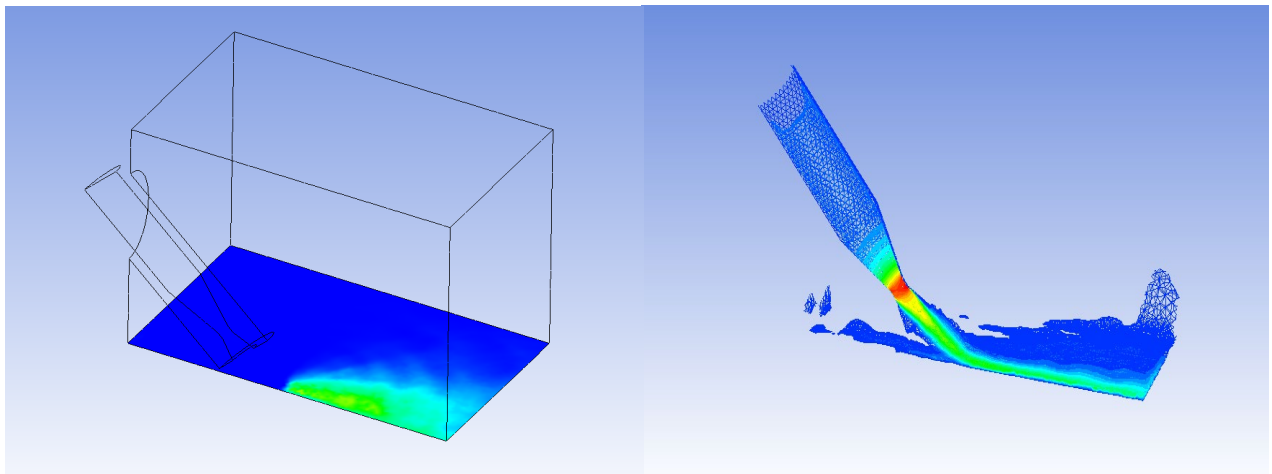


Fig. 29 Distribution of shearing force on domain floor (left), flow conditions through structure (right).

Here in Figure 29 the flow conditions and the distribution of the shearing force on the domain floor can be seen, the force is relatively small as the flow velocity is low. Maximum pressures seen on the surface were approximately 10Pa.

In the RSM tool the dimensions of the Venturi section were specified to vary between:

Inlet radius 600 – 700mm (measured half way through the tapered section).

Throat radius 300 – 600mm

The RSM program automatically generates 10 simulations with the variable dimensions, solves each one and records the result of the shearing force on the domain floor. Using the results a response surface graph was generated as seen in Figure 30.

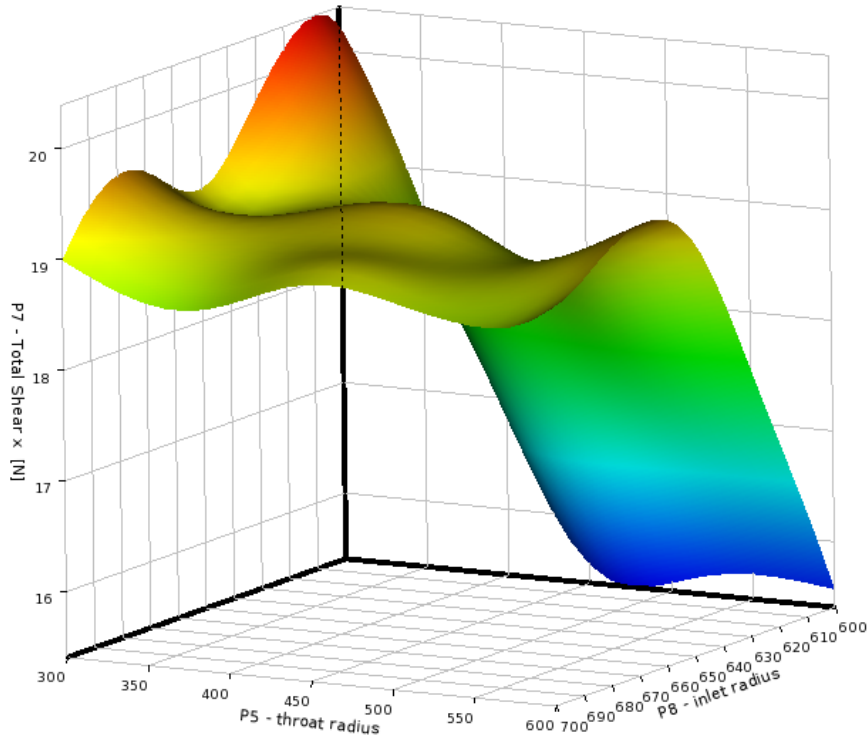


Fig. 30 Response surface showing how the inlet and throat radii affect the measured shear force.

The response surface is shaped in a 3-dimensional space according to the results of the experiment. Areas with high peaks indicate optimum results and are coloured orange/red, whereas the low areas are suboptimal results coloured in green/blue.

The optimum areas are those that produced the highest values of shear stress as measured on the domain floor and directly correspond to the highest velocity seen through the Venturi section.

From the response surface graph it was seen that a clear optimum point was established at an inlet diameter of 600mm and throat radius of 300mm this is seen as the highest peak on the graph, producing almost 21N of shear force on the domain floor.

However this area and the dimensions it represents was disregarded as the throat dimension would be too small to accommodate a wind turbine with sufficient power capacity for the UK daily average household energy consumption according to (WindExchange, 2018).

The 300mm throat radius was included in the experiment to test the validity of the results. The highest shear forces were theorised to be seen where the throat radius is very small as the more the throat area is decreased the higher the velocity will be through the throat in accordance to Bernoulli's continuity equation.

This theory was supported with the highest shear and optimum result corresponding to the smallest throat radius.

With this added confidence in results, the next step was to choose an area on the surface that would optimise the Venturi section and still comply with the minimum dimension requirements set out in the research section.

The dimensions were selected:

Inlet radius – 650mm

Throat radius – 450mm

This area is seen as the middle peak on the response surface, it produced the second highest value of shear force with a value of 19.54N and is slightly larger than the 400mm minimum radius.

To verify these chosen dimensions, the full 3D model of the structure was updated accordingly and was put through a set of simulations identical to the experiment from the 3D analysis. By doing this the original and optimised versions of the Venturi section were directly compared. The updated 3D model and Venturi section can be seen below in Figure 31.

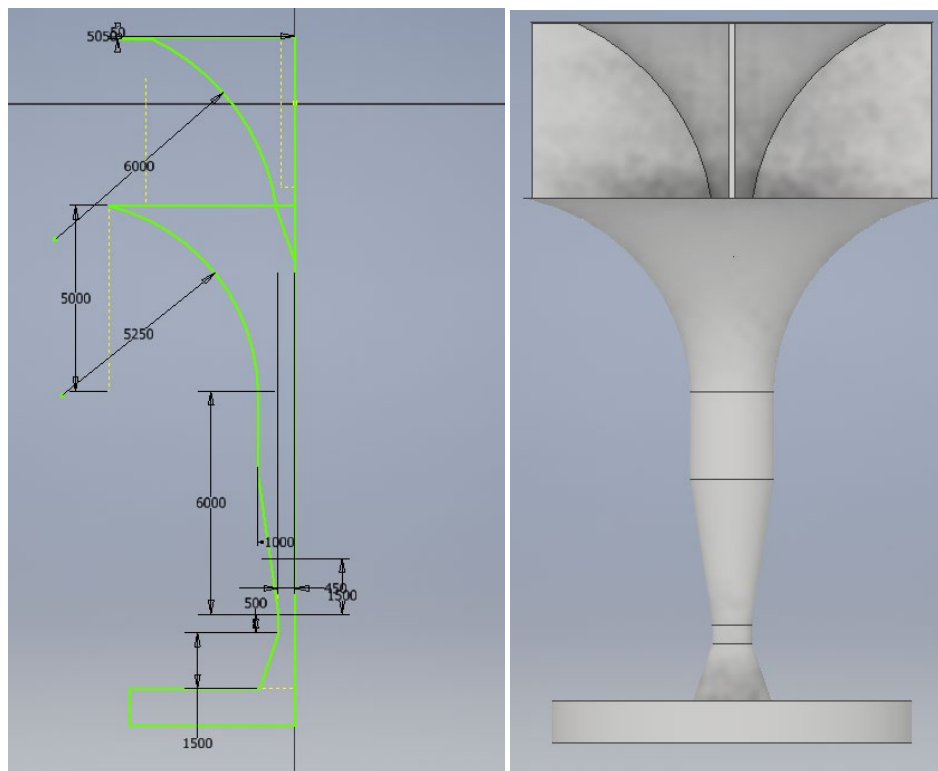


Fig. 31 Final design, sketch and 3D model.

Not depicted in Figure 31 are the exhaust holes (4 equally spaced rectangles 1m long by 0.5m high).

The results from the repeated experiment are detailed below in Table 4.

Table. 4: Results from both the 3D analysis and the RSM further analysis.

Inlet-velocity	Venturi-Velocity	Inlet-Venturi Ratio	RSM-Velocity	RSM Inlet-Venturi Ratio
2	3.75	1.88	3.84	1.92
4	8.25	2.06	8.94	2.24
6	11.7	1.95	11.88	1.98
10	16	1.6	17.2	1.72
20	27	1.35	28.84	1.44
40	54	1.35	52.7	1.32

The optimisation of the Venturi section returned higher wind velocities for all inlet velocities barring the 40m/s inlet velocity.

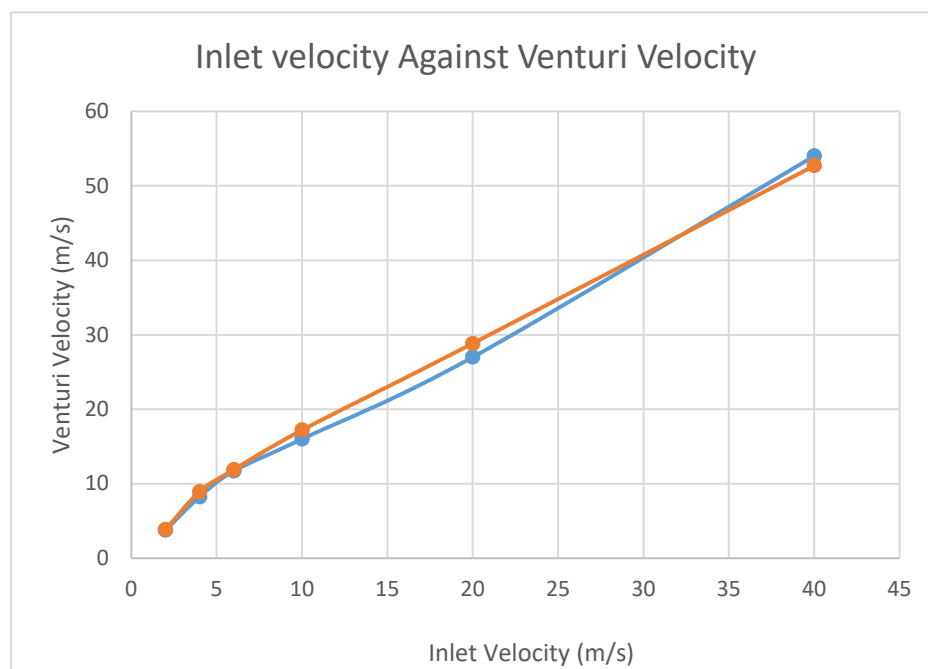


Fig. 32 Graph showing how the inlet velocities and Venturi velocities are related. RSM results seen in orange, original results in blue.

The graph from Figure 32 shows the newly attained results from the RSM process, this is represented as the orange line and displays the relationship between the inlet wind velocity and the maximum velocity measured in the Venturi section.

The blue line represents the results from the original 3D model

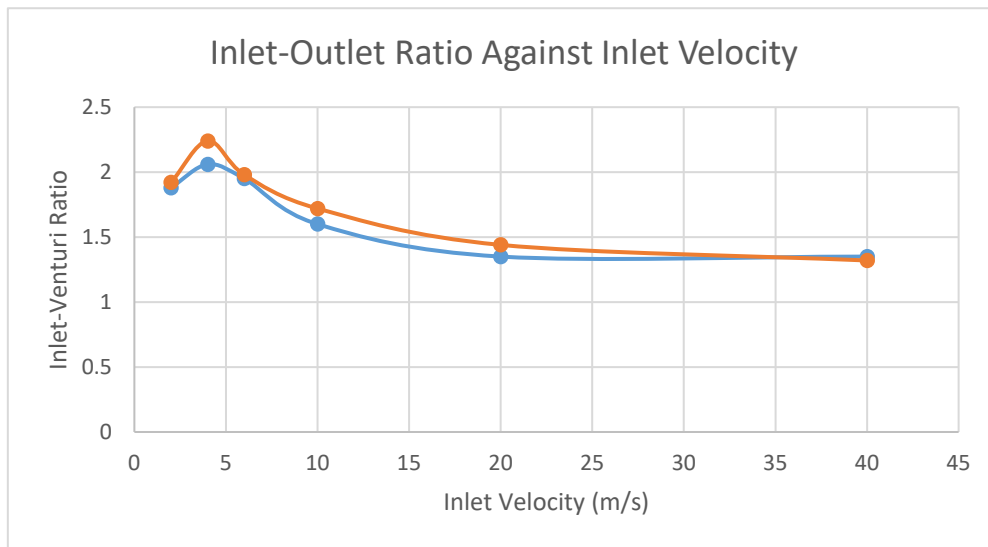


Fig. 33 Ratio of the inlet and Venturi velocities. RSM results seen in orange, original results in blue.

Here in Figure 33 the improvements in performance can be seen clearly, although the improvements at most inlet velocities were relatively minor, there was a marked improvement at the 4m/s inlet velocity. The newly developed model produced an amplification factor of 2.24 with the original model producing only 2.06.

Also noted was the fact that the new model seems to hold a higher amplification factor for longer as the inlet velocity is increased further. This is up to a point of the 40m/s inlet velocity where the new model is actually outperformed by the original model.

5. Discussion of Results

This section will serve as an appraisal for each step in the analysis process and will comment on the accuracy and validity of the results achieved.

2D Analysis Discussion

The 2D analysis was a crucial starting point for the optimisation process. However there are 2 major concerns with the accuracy of the results:

1. The model is limited to 2D flow and therefore is a simplified version of the real life case where flows patterns can be more complex and more difficult to predict.
2. The way in which the model was setup did not account for 'rejection' of flow from the system, all flow was directly inserted into the flow channels and therefore the velocity values seen in the 2D analysis results are over estimated.

The results still provided the best possible indication for an optimal design.

3D Analysis Discussion

As discussed in the research section, the motion of fluids can be challenging to predict and model due to the nature of turbulent flows. This was accounted for with an appropriate selection of a turbulence model for achieving results.

The selections for all parameters for the 3D analysis model were based on a previous experiment similar in nature which achieved a percentage error of only 2.7% when measured against experimental data.

The fact that key data features from the 2D analysis were also present in the 3D analysis showed that the results from the 3D simulations were actually representing the flow conditions that would be expected when interacting with the structure geometry, as opposed to generating random results.

The 3D analysis was successful in proving that the initial concept design could be taken from a 2D environment into its designated 3D environment and perform its intended function. It showed that the design could take in low velocity winds and accelerate them to a velocity that is above the average cut-in speed for an average wind turbine (3-4m/s).

The design was most efficient at accelerating the 4m/s inlet velocity, showing that the 2D optimisation process was successful. The 3D analysis proved that the design would be successful in completing its main intended goal of converting the UK average annual wind velocity of 4m/s to a speed of 8.25m/s. This exceeded the goal set out in the initial concept section of amplifying the inlet velocity by a factor of two.

At this point in the analysis process there is one source of uncertainty:

The most impactful source is the inaccuracies that arise when attempting to model the turbulent flow through the channels of the design and modelling the boundary conditions that dictate how the flow interacts with the solid surfaces.

Assuming that the simulations run in this process hold the same percentage error as the model that the simulations were based on, the error in the results achieved would be $\pm 2.7\%$.

When this percentage error is considered, the design still achieves its goal of accelerating 4m/s flow

to 8.03m/s.

RSM Discussion

The RSM approach to optimisation is a useful tool for determining optimal geometries of a component/system. It can drastically reduce the time it takes to arrive at an optimised design and in some cases will produce a design that would otherwise never be considered.

There are two major factors to consider when appraising the results of RSM:

1. To generate an optimised design, the RSM uses the results that are provided by the CFD simulation. Any inaccuracies within the CFD model will be used in the RSM and therefore the accuracy of the optimised design relies on the accuracy of the CFD simulation.
2. RSM uses $Y = f(x_1, x_2, x_3...) + e$ as the base for all its calculations. The important term to note is 'e', the unknown factors that can affect the output variable.

Without knowing the value and nature of these factors they cannot be included in the RSM approach and therefore the true extent of the engineering problem cannot be modelled. To achieve a high accuracy, the RSM should include all known factors when possible.

Some factors are inherently difficult to model accurately such as the thermal effects and turbulence. Each of these factors will affect the flow inside the system to a degree.

(Nagendra 2012) Adds that "The challenge for RSM to be a viable design tool is to model the continuum and capture the rich repertoire of physical phenomena accurately".

The further analysis yielded a successful outcome according to the results from Table 4. The newly dimensioned Venturi section achieved higher wind velocities through almost the full range of inlet velocities.

6. Conclusion

This project set out to prove the viability of a design concept, a novel design for a 'green building' that could produce a significant percentage of its energy usage internally with the use of a wind turbine.

Considering all the results generated from the testing and analysis process, it can be said that the concept is theoretically viable. With the advantageous capabilities of accelerating low velocity winds and amplifying any given ambient wind (up to cut off speed) to boost the power output of the wind turbine and provide a higher percentage of the household energy demand.

The results show the possibility of a modifiable optimisation process, meaning that the structure could be optimised for a given environment to produce the best power generating capabilities.

Based upon the results the theoretical maximum power outputs can be calculated using the wind power equation and assuming a turbine that is 900mm in diameter can be housed in the throat of the venture section.

Table. 5: Calculated power outputs of the design concept.

Inlet Velocity	Venturi Velocity	Max Power Output (Watts)	Power Outputs with Betz Limit	Percentage of Av Household Energy Supplied (6hr constant power)
2	3.84	89	40	2
4	8.94	1,118.00	497	24.85
6	11.88	2,624	1167	58.53
10	17.2	7963	3542	177

Table 5 above shows the flow velocities at the inlet and through the Venturi section.

The maximum theoretical power output was calculated using the wind power equation using a 900mm diameter turbine and air density at 1.23Kg/m³. This power output value is not accurate as any turbine will be subject to the Betz limit which describes the highest possible efficiency for a wind turbine. In theory the maximum efficiency is 59.3%, however most wind turbines will only achieve 75-80% of the maximum efficiency.

With the Betz limit included the power outputs were recalculated with an overall efficiency coefficient of 44.475%, this is the lower limit of efficiency for wind turbines.

The final column of Table 5 shows the percentage of power supplied by the concept design if the wind were held constant for a 6 hour period. This calculation is based on the reported UK daily average household consumption, 12KWh (GreenAge, 2015).

At the lowest ambient wind velocities(1-2m/s), the percentage of power supplied by the system is relatively insignificant but could still assist in the reduction of energy demanded from the grid. The real benefits of the system can be seen when wind velocities reach that of the estimated UK annual average of 4m/s. When this wind velocity is held constant for a 6-hour period the power generated could cover nearly one quarter of the daily household energy demand.

On a particularly windy day, if ambient wind was held at 6m/s for a 6-hour period, the percentage of power supplied raises to 58.53% of the 12KWh daily average. This could significantly reduce energy demand from an external source and with a calculated power output of 497W it would be capable of supplying power to several large household appliances at the same time (ofgem, 2018).

In more extreme conditions a 10m/s inlet velocity can be accelerated to 17.2m/s, this is close to the shut off speed of 24m/s for an average wind turbine (BWEA, 2005). Under these conditions, held constant for a 6-hour period the system would theoretically produce 177% of the daily household energy requirements, covering the 12KWh demand and generating a 9.24KWh surplus. These environmental conditions would be rare but are within the safe operating limit for an average wind turbine. The surplus energy produced could possibly be stored for low wind days.

7. Evaluations and Recommendations for Further Work

Self-Evaluation

I believe that the main goal for this project was achieved. All of the results from the analysis stages indicate that the design could successfully generate a substantial amount of energy that could be used to provide the necessary power that the structure will consume to run its desired appliances. By incorporating the concept design in new and even existing buildings, the structure could become, at least, partly energy-sustainable.

Appropriate models and parameters were selected for the simulations during the analysis stage and the generated results were largely coherent with observed flow characteristics and trends being visible from 2D analysis all the way into the final, further analysis stage.

The most time consuming task in the project was the set-up process for the simulation models, in particular the 3D models were challenging to create. In the early stages of the project, the layout of an optimal simulation, i.e. the position of the structure in the domain and the inlet and outlet positions, were not fully understood.

If I were to repeat the project I would allocate more time to the simulation setups.

The RSM process was particularly challenging, although it is possible to include many variables into the optimisation process, I was only able to setup the process using two variables. Again better time management would have allowed for more research into the methods of RSM to include more variables.

I believe that the level of optimisation achieved through the analysis process was fairly high and it is reflected in the calculated power outputs of the concept design.

Recommendations for Further Work

The first recommendation is related to the further optimisations of the inlet area. During the 2D and 3D analysis it was seen that the inlet was not fully efficient at utilising all the flow that entered the system, the phenomenon of 'spilling' was observed where air would enter through the inlet and, due to factors related to the geometry of the inlet, a significant amount of air would be lost through the other side of the inlet.

Further analysis could be conducted into the features of the inlet, to design them in such a way that minimises spilling. One possibility is to investigate the effect that the shape of the guiding fins have on containing the flow to the desired flow path, it is theorised that a helix or spiral shaped fin set-up could possibly achieve such a result as it eliminates the linear path from the original design that allowed for a straight-forward path for the flow to easily spill.

Reducing the amount of air spilled by the system would enable the use of taller inlet areas and would increase the volume of air that can pass through the structure, which in turn could further improve the efficiency of the amplification of low velocity winds.

Further investigation could be conducted to conclude a hypothesis developed upon analysis of the 3D simulation results. It was seen that optimisation process from the 2D analysis process, which used a 4m/s inlet velocity to derive optimal dimensions, was directly translated to the 3D environment where peak amplification was again seen at 4m/s.

This suggests the possibility of a repeatable optimisation process that could be performed for any desired inlet velocity, thus creating a complete structure that is 'fine-tuned' for the environment that it will function in.

By repeating the 2D and 3D analysis processes and varying the inlet velocity that drives the optimisation, the generated results could be analysed to assess whether the same trend of direct translation from 2D optimisation to 3D maximum efficiency occurs.

The final recommendation for further work is to investigate the viability of utilising the design and its flow characteristics for the use in a possible air conditioning and heating system for the surrounding usable space of the structure.

In the current design, large volumes of air flow through the centre of the structure and also under the floor of the internal usable space.

An investigation could be conducted into the thermal properties of this air-flow to design a novel cooling system. Possibly, the flow could be passed through a heater and syphoned from its regular flow path to distribute heat around the usable space.

The investigation could look at the impact that this syphoning would have on the efficiency of the whole system and conclude on whether such a heating method would use less energy than regular methods.

In later stages, small scale physical prototypes could be tested and compared to the simulation results. Then, to fully develop this concept design research must be done to determine the exact wind turbine that will be used, the building materials and methods, the electronic components to be used and also the costing for the design and planning permission.

8. References

- Akbarian, M., & Morshedi, A. (2014). *APPLICATION OF RESPONSE SURFACE METHODOLOGY: DESIGN OF EXPERIMENTS AND OPTIMIZATION: A MINI REVIEW*. Indian Journal of Fundamental and Applied Life Sciences .
- Arici, Y., & Özel, F. H. (2012). *Comparison of 2D vs. 3D Modeling Approaches*.
- Bendose, M. P., & Sigmund, O. (2004). *Topology Optimisation: Theory, Methods and Applications*. Springer.
- Boyle, G. (1996). *Renewable Energy - Power for a Sustainable Future*.
- BWEA. (2005). *BWEA Briefi ng Sheet*.
- Cable, M. (2009). *An Evaluation of Turbulence Models for the Numerical*. Ontario: Queen's University.
- Carley, K., Kamneva, N., & Remi, J. (2004). *Response Surface Methodology CASOS Technical Repor*. Carnegie Mellon University.
- Cunningham, P. (2014). Two Approaches to Design Optimization During Finite Element Analysis. *CAE Associates*.
- Dvorak, P. (2010, March 22). *WindPower Engineering & Development*. Retrieved from <https://www.windpowerengineering.com/small-wind-design/is-it-a-chimney...or-a-turbine/>
- Encyclopedia Britannica. (2006). Trubulent Flow Physics. *Encyclopedia Britannica*.
- eternoohydro. (2014). Retrieved from <http://www.eternoohydro.com/turbines/axial-flow-turbines.html>
- Ferziger, J. H., & Peric, M. (2002). *Computational Methods for Fluid Dynamics*. New York: Springer.
- GE Renewable Energy. (2018). *Bulb Hydro Turbine*. Retrieved from <https://www.gerenewableenergy.com/hydro-power/large-hydropower-solutions/hydro-turbines/bulb-turbine>
- Gitano-Briggs, H. (2012). *Low Speed Wind Turbine Design*.
- Gorse, C. (2012). *Oxford Dictionary of Construction, Surveying & Civil Engineering*. Oxford University Press.
- GreenAge. (2015, Nov 9). *How much energy does my home use?* Retrieved from The Green Age: <https://www.thegreenage.co.uk/how-much-energy-does-my-home-use/>
- Hall, W., & Keynes, M. (1974). *Lokking at Fluids in Motion*. The Open University.
- Helmy, M. L. (2003). *Axial flow turbine design for wind power*. Science Direct.
- Kaiser, C., Friedewald, D., & Nitzsche, J. (2017). *COMPARISON OF NONLINEAR CFD WITH TIME-LINEARIZED*.
- Kotharia , S., Kaushikb, S., & Panwara, N. (2010). *Role of renewable energy sources in environmental protection: A review* . Elsevier .

- Larsson, R. (2016). *Methodology for Topology and Shape Optimization: Application to a Rear Lower Control Arm*. CHALMERS UNIVERSITY OF TECHNOLOGY .
- Nagendra, S. (2012). *Response surface methodology: advantages and challenges* . General Electric Corporate R&D.
- Nazif, H. R., & Tabrizib, H. B. (2013). *Applying a non-equilibrium wall function in $k-\epsilon$ turbulent modelling of hydrodynamic circulating flow*. Elsevier.
- NewAtlas. (2010). *Secret Energy Turbine: rooftop wind power in stealth mode*. Retrieved from <https://newatlas.com/secret-energy-wind-turbine/14299/>
- OECD. (2009). *"Standardised design" for schools*.
- ofgem. (2018). *Typical Domestic Consumption Values*. Retrieved from ofgem: <https://www.ofgem.gov.uk/gas/retail-market/monitoring-data-and-statistics/typical-domestic-consumption-values>
- Ortega-Casanova, J. (2017). *Application of CFD on the optimization by response surface methodology of a micromixing unit and its use as a chemical microreactor*. ELSEVIER.
- Piggot, H. (2001). *A Guide to Choosing Wind Power*. ISBN.
- Raoufi, A., Shams, M., Farzaneh, & Ebrahimi, R. (2008). *Numerical simulation and optimization of fluid flow in cyclone vortex finder*. ELSVIER.
- Sarkar, A., & Behera, D. K. (2012). *Wind Turbine Blade Efficiency and Power Calculation with Electrical Analogy* . International Journal of Scientific and Research Publications.
- ScienceStruck. (2018). *An In-depth Explanation of the Venturi Effect and its Applications*. Retrieved from <https://sciencestruck.com/explanation-applications-of-venturi-effect>
- SHArchitecture. (2016, May 4). sustainable architecture: bahrain world trade center. *SHArchitecture*, p. 1.
- Thomas, S. (2018). *3D Computational Fluid Dynamics : Ahmed*.
- Thomas, S. (2018). *Applying FEA to your final year project*.
- Thomas, S. H. (2018).
- U.S. Energy Information Agency. (2016). How much energy is consumed in U.S. residential and commercial buildings? *eia*.
- Wang, D. T.-Y. (2015). *Design and Analysis of Experiments*. Cheng Kung, TAIWAN.
- WayMarking. (2018). *Bahrain World Trade Center - Manama, Bahrain Image*. Retrieved from <http://www.waymarking.com/gallery/image.aspx?f=1&guid=977a2827-4c08-46e6-b828-8758d0c55290>
- WikiMedia. (2017). Retrieved from <https://commons.wikimedia.org/wiki/File:BulbTurbineSketch.svg>
- WindEurope. (2016). *Wind in power: 2016 European statistics*. Retrieved from WnidEurope.
- WindExchange. (2018). *Small Wind Guidebook*. Retrieved from WindExchange: <https://windexchange.energy.gov/small-wind-guidebook>

Worldgbc. (2016). What is green building? *World Green Building Council*.