

# Modelling the Output of a Flat-Roof Mounted Wind Turbine with an Edge Mounted Lip

S. J. Wylie<sup>1</sup>, S. J. Watson<sup>1</sup>, D. G. Infield<sup>2</sup>

<sup>1</sup>Centre for Renewable Energy Systems Technology, Department of Electronic and Electrical Engineering,

Loughborough University, Loughborough, Leicestershire, LE11 3TU, United Kingdom

Tel: +44 1509 635303 (S. J. Wylie), Email : [s.j.wylie@lboro.ac.uk](mailto:s.j.wylie@lboro.ac.uk)

Tel: +44 1509 635348 (S. J. Watson), Email : [s.j.watson@lboro.ac.uk](mailto:s.j.watson@lboro.ac.uk)

Fax : +44(0)1509 63 5301

<sup>2</sup>Institute of Energy and Environment, University of Strathclyde, Glasgow, G1 GXW, United Kingdom

Tel: +44 141 5482373, Email: [david.infield@eee.strath.ac.uk](mailto:david.infield@eee.strath.ac.uk)

**Keywords:** Computational Fluid Dynamics, Small wind turbines, Roof mounted turbines.

## Abstract:

This paper presents a computational fluid dynamic (CFD) simulation of the wind flow over a flat roof with and without an aerodynamically shaped lip on the leading edge of an isolated building. The intention is to show the advantage of mounting a relatively inexpensive aerodynamically shaped lip on the building to delay flow separation so that a roof mounted wind turbine can benefit from the accelerated wind over a flat roof building without having to be mounted on a relatively high mast outside of the flow separation region immediately above the roof surface. By altering a hypothetical rotor's resistance, a comparison of extracted power against the theoretical power available to the rotor will be made. From the simulations it can be seen that placement of an aerodynamically shaped lip on the buildings leading edge does increase power extraction from the theoretical turbine rotor, by 7% at its maximum. By increasing the amount of rotor resistance it can be seen that this can inhibit flow separation at the building's leading edge and increase wind speeds through the rotor of up to 15% and hence the power available to the rotor is increased. Although the results from this work are positive, more detailed analysis is needed into the aerodynamic optimisation of the lip shape and rotor placement.

## 1. Introduction:

Due to increasing demand for sustainable energy technologies there has been an increased interest in mounting wind turbines on roofs in the urban environment. The domestic sector has seen a number of small machines come into the market. At the same time, there has been an interest in larger machines for mounting on commercial buildings, many of which have relatively large flat roofs. A tall flat roofed building will experience accelerated winds over the top of the building, however, the wind flow will tend to separate at the leading edge of the building and this separation region immediately above the roof surface will tend to experience low turbulent winds [1]. To benefit from the advantage of the speed-up above the separation region requires a turbine mounted high up outside of the separation region which will require a rigid and potentially expensive mounting mast. An alternative, particularly where winds are prevalent from a one direction, is to mount a relatively inexpensive aerodynamically shaped lip on the edge of the building facing the most frequent wind direction. This, lip, will tend to accelerate the flow and delay separation so that an appropriately mounted wind turbine close to surface of the roof can benefit from these increased winds. This paper reports a computational fluid dynamics (CFD) analysis of the wind flow with and without this lip to calculate the expected output of a cross-flow turbine mounted on a flat roof in order to assess the benefits of the lip. The rotor resistance is varied to assess maximum power extraction for situations which include a building mounted lip and for those which do not.

## 2. Background:

The single most important factor when considering installation of any wind power device is the available resource to the specific site, most importantly the mean annual wind speed of the site. Available power in the wind increases with the cube of the mean wind speed. The specific mean wind speed at a site will affect its energy capture, yield, payback time, and most importantly from a developer's point view the revenue generated. Up until now wind farm developers have mainly exploited rural areas, as they are usually the areas of highest wind resource. Offshore wind farm development is now becoming increasingly popular, with these sites experiencing even higher wind speeds than onshore rural areas. As the number of possible sites decreases onshore and with the added cost and complexity of offshore installations developers are showing a growing interest in urban wind application. Urban areas typically experience wind speeds around two thirds of those experienced by rural ones, coupled with the fact that urban winds are also highly turbulent. This added turbulence will increase turbine fatiguing but can provide moderate gains in power extraction. Urban wind application does, however, have many other obstacles to overcome. The key issues being visual intrusion, noise, and vibrations from the turbine structure. Urban power generation will also raise issues with grid connection and metering as well as regulation, planning permission, grants and costs [2].

Despite these potential obstacles, there is some enthusiasm for urban wind generation and there is a developing market for this type of technology. Typical turbine designs have 1 – 2m rotor diameter with a rated power of 0.5 – 2.5 kW. They typically sell for around £3500 installed and use a mast of up to 3m tall.

The scope of this work is to model a turbine similar to that of a cross flow turbine, used in hydro generation schemes, and to see if its performance can be enhanced with the addition of an aerodynamically shaped lip on the building's edge close to the rotor.

## 2.1. CFD

Wind resource assessment is a very, if not the most, important part of the process for developing a wind farm. Generally this is done by using a combination of mathematical models along with field measurement data. The mathematical model is the tool which provides a description of the site, which can be used together with the measured data to produce a detailed account of a specific site's resource. Most of the mathematical models will include averaging and simplified expressions to express the behaviour of the atmospheric boundary layer (ABL). To solve all the equations related to the ABL would be far too complex and as site selection, or turbine configuration in this case, become more complex the analytical process of solving this would be near impossible.

An alternative approach is to use a CFD (computational fluid dynamics) code to model a particular site. This offers the ability to model any type of site, but with more complex terrain comes more computational demand.

For this particular example it would be very difficult to model the affects of the building, lip, and rotor all together so CFD has been used to give as accurate as possible a solution for the problem.

## 2.2. k – ε Turbulence Model

As is implied by the name, the k – ε model describes turbulence using two parameters, namely k and ε, both of which have model evolution equations. From previous work [5], k (turbulent kinetic energy) and ε (dissipation) are defined by equations (1) and (2).

$$k = \frac{(u_*)^2}{\sqrt{C_\mu}} \quad (1)$$

k is assumed constant throughout the model domain for this problem, and it takes a value of  $0.886807 \text{ m}^2\text{s}^{-2}$

$$\varepsilon(y) = \frac{(u_*)^3}{\kappa(y + y_o)} \quad (2)$$

Where y refers to the height above ground level in the CFD model and  $y_o$  the surface roughness height. This is a standard turbulence model used in CFD calculations and is of sufficient detail to model the site's turbulent effects.

### 3. Methodology:

The flow around a free standing building with and without an aerodynamically shaped lip on the building's upstream edge was computed using the Reynolds Averaged Navier Stokes (RANS) equations implemented in the commercial CFD software ANSYS CFX 11. ANSYS CFX is based on a coupled solver for mass and momentum and uses an algebraic multi-grid algorithm for convergence acceleration. Utilising the  $k - \varepsilon$  turbulence model, the average velocity can be calculated passing through the rotor and from that the theoretical power it extracts.

#### 3.1. The CFD Model

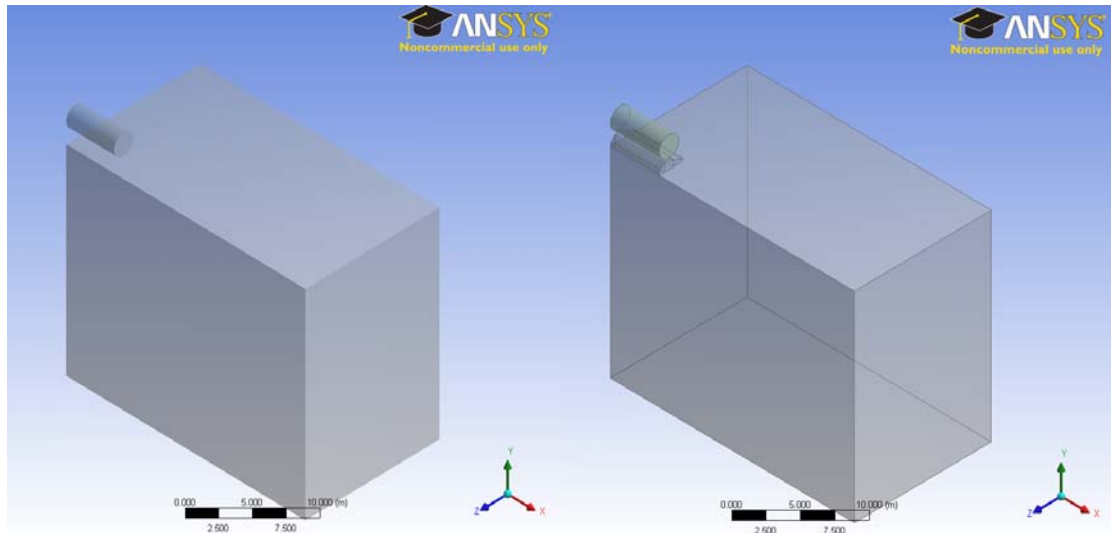
The model was run as a steady state simulation and is based on previous work to determine the performance of a ducted wind turbine [6]. Figure 1 (b) is the schematic of the building, rotor, and lip as modelled in ANSYS CFX, figure 1 (a) is the case with no lip attached. To cut down on computational time, only half the building, lip, and rotor were modelled. This was achieved by assuming a symmetry plane along one wall passing through the centre of the building geometry. The building has a half width of 25m, height of 20m, and a length of 14m. The rotor has a half width of 5m, diameter of 2m, and the centre of the rotor is raised 2m above the flat roof surface. The distance from the upstream edge of the building to the centre of the rotor was set to 1m. The lip spans 2.5m from the building's upstream edge and is split in to three sections, with a smooth chamfer to discourage flow separation. Figure 2 (a) is a schematic view of the lip. The domain which contains the geometry was large enough so that its walls would not influence the flow close to and around the building. The inlet of the domain has a half-width of 130m and a height of 100m. The domain extends 500m from the inlet in the stream wise direction to an outlet. The upstream face of building, and hence the lip, are 200m downstream of the inlet.

So that a suitable solution could be reached, in the solver, it is necessary to apply appropriate boundary conditions to the domain and the geometry. A free slip boundary condition was implemented at the top and side walls of the domain. This would ensure that the flow would not be retarded near to the walls and affect the solution. A logarithmic wind profile was applied to the inlet of the domain with a constant static pressure at the outlet. This ensures that the flow will only travel in one direction (from inlet to outlet). To try and make the simulation more realistic, values for the turbulent kinetic energy, k, and eddy dissipation,  $\varepsilon$ , were defined manually at the inlet of the domain; rather than use ANSYS CFX to estimate these (as described in section 2.2). A constant value with height was assumed for turbulent kinetic energy and the eddy dissipation was defined as a function of height, y, and the friction velocity,  $u_*$ . Along the ground surface of the domain a no slip wall condition was implemented, with the roughness of the surface being set to 30 times that of the aerodynamic roughness length. This is known as the equivalent sand grain roughness [3,4] and ensures that there is no step change in roughness once the logarithmic profile enters the domain. The building and lip structures had the no slip wall condition applied to them both.

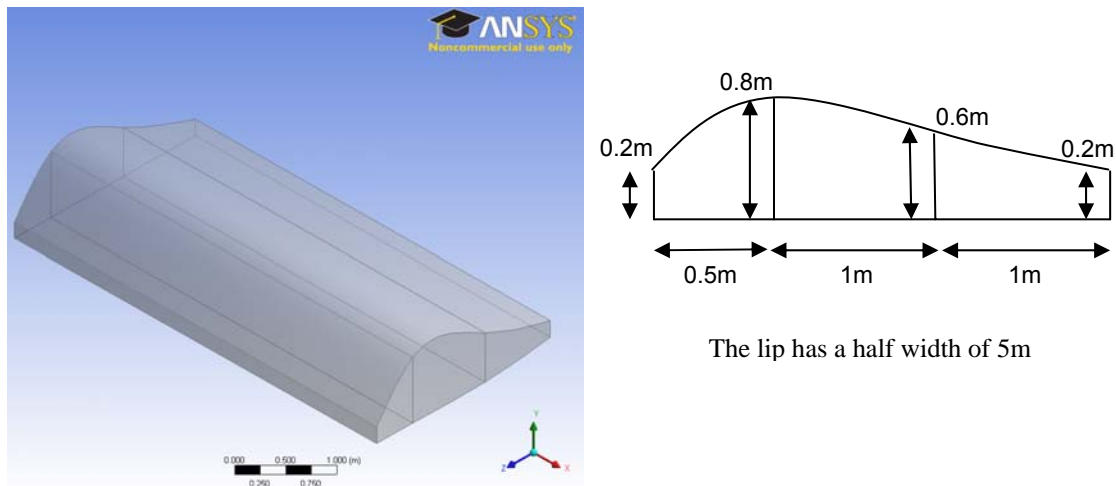
An unstructured tetrahedral mesh was used close to the rotor and lip structure. This was to ensure that the greatest changes in flow and pressure would be picked up accurately by the solver. To ensure that a detailed account of the flow structure close to the ground and along

the building faces were obtained an extended inflation layer was placed around the building and along the ground surface. The height of the inflation layer was extended so that it was above the height of the building and so encompassed the lip and rotor also in order to help accurately resolve flow which was parallel to the surfaces.

The rotor was modelled as a cylindrical volume, which was created to model a cross flow turbine. This type of turbine is similar to a vertical axis wind turbine (VAWT) but on its side. It was defined as a sub-domain rather than having a wall condition applied to it. In practice, it was very impractical to model each individual blade of the rotor, so momentum extraction was simulated by giving the cylindrical volume a resistance to the flow; defined in ANSYS CFX as a resistive loss coefficient.



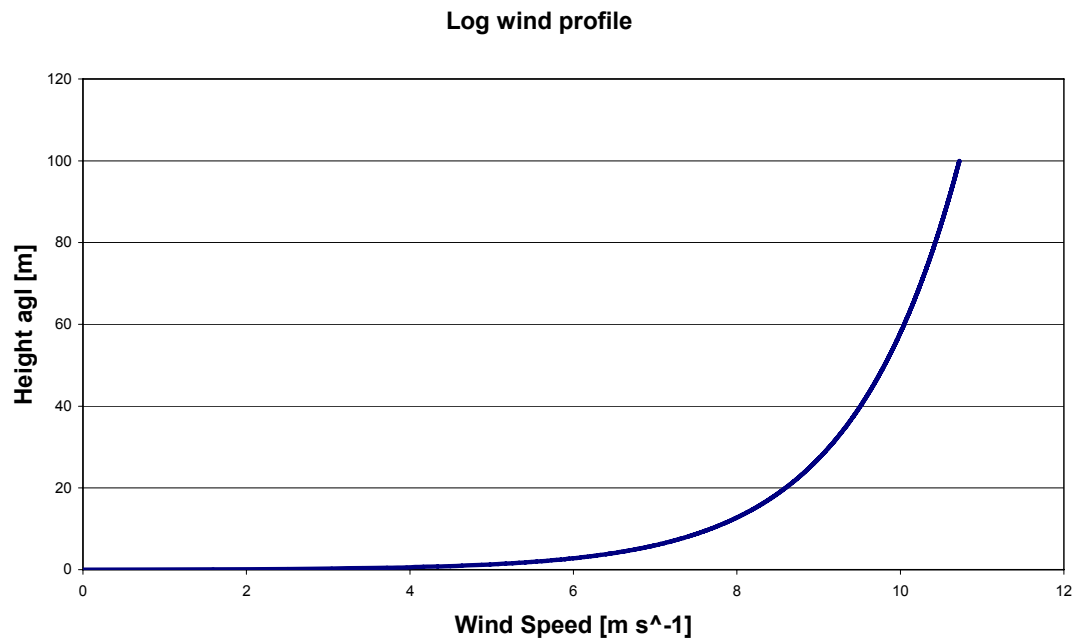
(a) (b)  
**Figure 1. (a) Building with no lip attached, (b) Building with lip attached. Both as modelled in ANSYS CFX .**



(a) (b)  
**Figure 2. (a) Lip as modelled in ANSYS CFX, (b) Detailed view of lip layout and measurements**

#### 4. Results and Analysis:

A number of different CFD simulations were carried out for the two different cases, one with the aerodynamically shaped lip attached to the building's leading edge and one without. The rotor position was not changed for both sets of simulations, neither was the lip position (or shape) for the simulations which included this. The boundary conditions, defined in section 3.1 were also kept constant for all of the simulations. The only parameter which was modified was the resistive loss coefficient of the rotor structure, which was modelled for 6 different cases ( $0.25 \text{ m}^{-1}$ ,  $0.5 \text{ m}^{-1}$ ,  $0.75 \text{ m}^{-1}$ ,  $1 \text{ m}^{-1}$ ,  $1.25 \text{ m}^{-1}$ ,  $1.5 \text{ m}^{-1}$ ) for the lipped condition and two extra cases ( $2 \text{ m}^{-1}$ ,  $2.25 \text{ m}^{-1}$ ) for the no lipped case. This is an isotropic loss coefficient and is defined per unit of length. Figure 3 shows the logarithmic profile, applied to the inlet of the domain.



**Figure 3. Logarithmic wind profile used as inlet wind speed in simulations**

Figure 4 (a and b) show the surface streamline plots for the two different cases modelled. For both simulations, the flow structure is broadly similar, with flow separation at the leading edge and also in the wake of the building. There is the expected separation region over the top of the building with a small area of rotating flow behind the rotor. Figure 5 (a and b) show a closer view of the flow near to the rotor. For the un-lipped case there is a small region of accelerated flow at the building edge but there is a region just behind the rotor where the flow had detached itself from the roof. If this is compared to the case where the lip is present, it can be seen that the region of accelerated flow is extended along the surface of the lip. Also, the flow stays attached to the lip once it has passed the rotor and the rotating turbulent air is delayed for longer than in the un-lipped case. This implies that the lip is delaying the separation of air flow over the building and giving the rotor cleaner, less turbulent air to extract power from. If the air flow along the building is observed more closely it can be seen the flow stays attached to the leading edge a lot more than for the case where the lip is not present. As both cases have the same boundary conditions and inlet profile this implies the lip is causing the flow to stay attached to the building face and is less turbulent than the case without the lip being present. From a wind engineering point of view this is beneficial, as less turbulent air will cause less fatiguing of the rotor and lip structure.

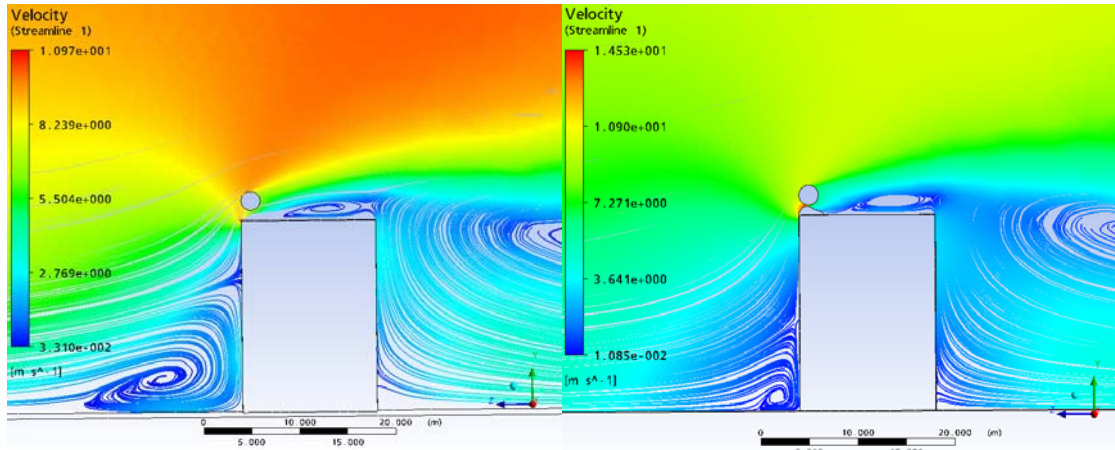


Figure 4. (a) streamline plot for un-lipped case, (b) streamline plot for lipped case.

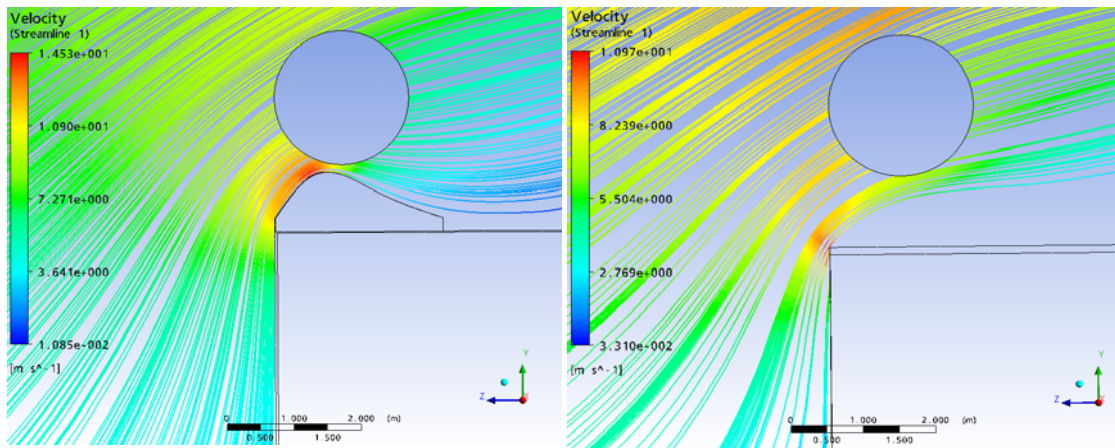


Figure 5. (a) streamline plot of lipped case, zoomed in on rotor, (b) streamline plot of un-lipped case, zoomed in on rotor.

For both cases (with and without lip attached)  $C_p$  is to be calculated for each value of resistive loss coefficient. The power extracted by the rotor will be calculated using the simple relationship:

$$\text{Power} = \text{Force} * \text{velocity}$$

The velocity is an average value across the rotor volume, calculated in the ANSYS CFX post processor, and the force is the total force on the rotor in the stream wise direction. Table 1 (a and b) show the theoretical power extracted by the rotor for each resistive loss coefficient in both simulation types.  $C_p$  is calculated by:

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

Where  $P_w$  is the power extracted by the rotor;  $\rho$  is the density of Air ( $1.225 \text{ kg m}^{-3}$ );  $A$  is the cross sectional area intercepted by the rotor ( $10\text{m}^2$ ); and  $U_o$  is the free stream wind speed.

$U_o$  was taken as  $8.65 \text{ ms}^{-1}$ . This value of the free stream wind speed was calculated by taking a point 100m upstream of the building's leading edge at the height of the centre of the rotor.

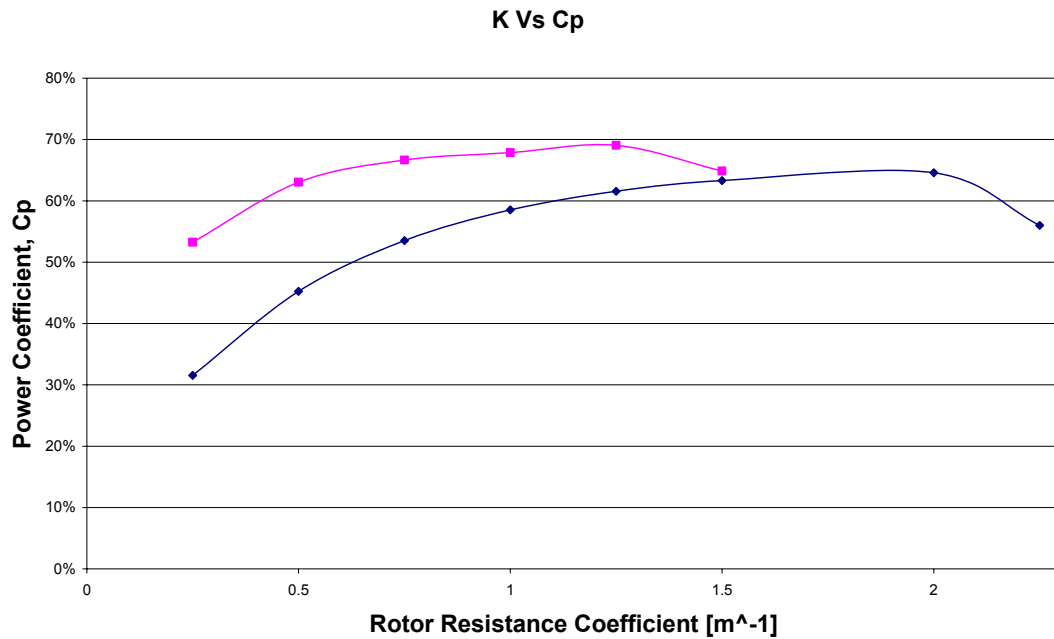
Rotor Resistance Coefficient ( $m^{-1}$ )	Ave' V through rotor ( $ms^{-1}$ )	Force on rotor (N)	Power through rotor (W)
0.25	8.26143	151.494	1251.56
0.5	7.75348	231.51	1795.01
0.75	7.3249	289.898	2123.47
1	6.96008	333.731	2322.79
1.25	6.64586	367.476	2442.19
1.5	6.37216	394.068	2511.06
2	5.91767	432.976	2562.21
2.25	5.72648	447.563	2222.00

**Table 1 (a). Theoretical power extracted by rotor for un-lipped case**

Rotor Resistance Coefficient ( $m^{-1}$ )	Ave' V through rotor ( $ms^{-1}$ )	Force on rotor (N)	Power through rotor (W)
0.25	9.72588	217.2	2112.46
0.5	8.39981	297.709	2500.70
0.75	7.9635	332.012	2643.98
1	7.08714	380.017	2693.23
1.25	6.61265	414.401	2740.29
1.5	6.23857	412.68	2574.53

**Table 1 (b). Theoretical power extracted by rotor for lipped case**

Figure 6 shows how  $C_p$  varies with different values of resistive loss coefficient,  $K$ . Note that the traditional Betz limit is exceeded as the building causes an acceleration of the upstream flow whereas the Betz limit only applies in the open field case. It can be seen that there is an increase in extracted power for the case where the lip is present. The second point to mention is that the performance of the rotor with the lip present starts to fall away quicker than that of the same cases for the no lipped case. This would suggest that the lip has reached its optimum performance for that particular configuration. As the rotor resistance is increased the oncoming flow will start to be influenced more and more, until eventually the resistance is so high that it will simply deflect around the rotor. Added to the fact that the rotor has the lip, which will also deviate the flow around it, the combined effect will mean that reduced flow will travel through the rotor much sooner than for the un lipped case. Even though the performance drops off sooner for the case with the lip attached, the added power extraction still makes this a better option to consider. Although detailed costing of both situations has not been done, the results are promising enough to warrant further investigation into the roof mounted turbines with an aerodynamically shaped lip.



**Figure 6. graph to show Cp Vs resistive loss coefficient for both lipped and un-lipped simulations.**

To see exactly where the no lipped case peaked a few more simulations were run, namely  $K = 2 \text{ m}^{-1}$  and  $K = 2.25 \text{ m}^{-1}$ . From figure 7 it can be seen that the no lipped case peaks at a value of  $K = 2$ , compared to that  $1.25 \text{ m}^{-1}$  for the lipped case. Although the no lipped case performance does continue to increase for longer than the lipped case it never exceeds the maximum performance for the lipped case. Table 2 (a and b) shows the percentage increase in velocity and Cp for the lipped case, compared to the un-lipped case. From table 2 it can be seen that the maximum value of Cp for the lipped case is 7% higher than the un-lipped case, however, the cross flow turbine would need to be designed to have a lower thrust coefficient to take advantage of this.

Rotor Resistance Coefficient (m <sup>-1</sup> )	Cp (un-lipped)	Cp (lipped)
0.25	31.55%	53.25%
0.5	45.24%	63.03%
0.75	53.52%	66.64%
1	58.55%	67.88%
1.25	61.56%	69.07%
1.5	63.29%	64.89%
2	64.58%	
2.25	56.01%	

**Table 2. Cp for un-lipped ad lipped cases as a function of resistive loss coefficient.**

## 5. Conclusions:

The CFD modelling of an aerodynamically shaped lip on the edge of a flat roofed building has shown promising results. It is seen that for a hypothetical cross flow rotor an extra 7% of energy could be extracted, compared to that of the same rotor without the lip. The performance of the lip does starts to fall away quicker than that of the un-lipped case and the rotor would need to have a lower thrust coefficient than the lipped case to take advantage of this increase, leading to the idea of an optimum configuration for the lip. As, in general, urban

wind speeds are lower than that of their rural counterparts any extra gain in extracted power would be welcomed to the market.

An aerodynamically shaped lip would be a relatively cheap way to improve the quality of wind flow that a turbine would see. The cross flow turbine used in this example would also be mounted closer to the surface of the roof, compared to turbines mounted on masts for example. This would reduce structural problems and would also reduce costs in materials used for fabrication and installation.

## **6. Recommendations**

With the promising results from this work further work would be warranted with regards to the configuration of the aerodynamically shaped lip. Further CFD simulations to identify a relationship between rotor resistance, lip shape and configuration, and also rotor placement with respect to the lip and the building could further increase the benefit of this type of urban wind generation set up. Once this extra work has been completed, the lip design could be modelled on a number of different building types and also for buildings in close vicinity to one another to see how they interact with the wakes generated by adjacent buildings.

## **7. References:**

- [1] Mertens S. "Wind energy in the built environment", Multi-science, 2006.
- [2] Dayan E. Small scale, building integrated, wind power systems," *BRE Information Paper IP 12/05*, BRE Centre for Sustainable Development, September 2005.
- [3] M. White, F.M., "Viscous Fluid Flow", McGraw-Hill, 1979.
- [4] H. Schlichting., "Boundary Layer Theory", McGraw-Hill, 1979.
- [5] B. E. Launder and D. E. Spalding DE, "Mathematical models of turbulence", Academic Press, 1972.
- [6] S.J.Watson, D. G. Infield, J. P. Barton, S. J. Wylie., "Modelling of the Performance of a Building – Mounted Ducted Wind Turbine", 2007.