The City College of New York Department of Mechanical Engineering Applied Fluid Mechanics Professor: Dr. Yiannis Andreopoulos



Diffuser Augmented Wind Turbine Presented By: Jose Cortes

<u>Abstract</u>

Production of electricity using wind turbines is completely clean and renewable. Fossil fuels create emissions that can be harmful to the atmosphere and contribute to global warming; wind power on the other hand provides an environmentally safe alternative. The use and implementation of wind turbines for power production is steadily growing with the demand for clean power generation. With the rising cost of raw materials, initial installation and energy production; customers expect to get the highest power generation per dollar invested. This demand for high efficiency drives us to find ways to quickly improve upon old designs and find alternative methods to maximize the power production.

Goals and Objective

In this paper we seek to analyze a diffuser augmented wind turbine using a computational fluid dynamic approach. For our analysis we will be using Ansys-Fluent. The set up for the wind turbine consist of: a converging nozzle which draws the air inside a cylinder increasing the velocity of the incoming air, followed by a short cylinder containing the wind turbine, followed by a diverging nozzle that creates a lower than atmospheric pressure at the outlet helping to draw out the air faster. The result is expected to be an increased in power generated in comparison to a bare wind turbine.

In this paper I will investigate the effectiveness of the converging-diverging nozzle adaptation in reference to a bare wind turbine model, and then I will try to improve upon my original design by increasing the inlet size and length at the converging nozzle and comparing the power generated with the other configurations.

<u>Background</u>

The blade aerodynamic profile is of outmost importance in for blade performance. Small alterations for the shape of the blade have great impact on the power curve and noise levels. There is a huge selection of aerodynamic profiles that can be selected along with blade shapes and lengths. We can dedicate an indefinite amount of time to the analysis of blade profiles and shapes, but the focus of our study is the impact of the converging-diverging nozzles on the power performance of a wind turbine, and for this reason we will create our own aerodynamic profile and shape for the wind turbine blade. The size of our model is rather small but it will be sufficient to draw conclusions and comparisons.

Profile and Shape







Basic Assumption

We will assume a steady, homogenous, irrotational and incompressible laminar flow at the inlet.

Static pressure at the upwind and downwind boundaries are equal to atmospheric pressure

Governing equations

We are going to apply horizontal momentum at the inlet of the cylinder containing the wind turbine and at the outlet. A is the swept area.

$$\sum F_{x} = -T = m(V_{in} - V_{out}) = \rho AV(V_{in} - V_{out}) \quad Eq. 1$$

The thrust at the turbine can also be calculates using the differential pressure between the inlet and outlet multiplied by the swept area A.

$$T = (p_{in} - p_{out})A \quad Eq.2$$

Axial thrust is applied on the wind turbine in the direction of the flow, the turbine applies an equal and opposite direction on the wind.

We can apply Bernoulli's equation to find to find the values of p_{in} and p_{out}

$$p_{in} - p_{out} = \frac{1}{2} \rho (V_{in}^2 - V_{out}^2) \quad Eq.3$$

We can now use equation 3 into equation 2, this yields:

$$V = \frac{1}{2} (V_{in} - V_{out}) \quad Eq.4$$

In which V is the stream velocity through the turbine.

Calculation of Wind Power

We start by taking by considering the kinetic energy that the air carrying:

$$KE = \frac{1}{2}mV^2 \quad Eq.5$$

We are interested in calculating the mass flow rate that is going through the wind turbine.

$$\dot{m} = V x A x \rho = \frac{m}{s} m^2 \frac{kg}{m^3} = \frac{kg}{s} Eq. 6$$

Where A is again the swept area, V is the average velocity of the air going through the wind turbine, and ρ is the density of the air.

The power is calculated by inserting the mass flow rate into Eq. 5, resulting in the following equation:

$$P = \frac{1}{2}A\rho V^3 = watts$$

This is the ideal power generated by the wind, real life power generation in wind turbines can range from 0.25P to 0.45P.

Solid Modeling

The original solid modeling was made with SolidWorks 2011, and it was Imported into the Ansys modeler as a IGS file, later it was remade with the Ansys Design modeler in order to facilitate changes in the geometry.







Ansys Design Modeler Setup

Once the solid model was completed, I surrounded it with a cubic enclosure. This enclosure provides a fluid volume which fills the empty spaces that are not occupied by the solid model. The solid model is later suppressed leaving only the fluid enclosure for the analysis.





Solid Model Modification due problems with large blade displacements

This modification was implemented after being unsuccessful in getting a solution in the transient model using a dynamic mesh. The problem has to do with the large displacements that occur as the wind turbine rotates. Fluent provides smoothing to deal with small displacements and remeshing to deal with large displacements.

Smoothing introduces spring like characteristics to the elements, allowing them to deform as the model undergoes small displacement. Unfortunately for large displacement smoothing does not work well, producing a negative cell error during the program execution.

For large displacement remeshing is the adequate choice. Remeshing allows the user to set the largest and smallest element volume as well as the quality of these elements. However due to the large displacements of the blades and the increasing angular speed is very difficult to successfully complete the analysis without encountering a negative volume error.

Solid Model Modification due problems with large blade displacements

In order to eliminate negative volume errors, it was necessary to enclose the entire turbine blade inside a cylindrical volume. The purpose is to rotate the volume that encloses the turbine blades and not the turbine blades inside the mesh. The UDF will compute the components of the force hitting the turbine blade faces (which will be set a "wall" type boundary condition) and apply the angular velocity to the cylinder containing the turbine, thus resulting in the same angular velocity.





Solid Model Modification due problems with large blade <u>displacements</u>

Omega value is obtained from the turbine blades (as the flow comes in contact with the wall boundary condition) through the UDF, but the rotation is applied to the entire cylinder containing the turbine causing it to rotate.





Rotation Vs. Flow Time (Nozzle Removed Configuration I)



Mesh Generation



Details of "Face Sizing" - Sizing			
	Scope		
	Scoping Method	Geometry Selection	
	Geometry	6 Faces	
	Definition		
	Suppressed	No	
	Туре	Element Size	
	Element Size	0.1 m	
	Behavior	Soft	
	Curvature Normal Angle	Default	
	Growth Rate	Default	
l	Growth Rate	Default	

Details of "Face Sizing 2" - Sizing			
	Scope		
	Scoping Method	Geometry Selection	
	Geometry	8 Faces	
Definition			
	Suppressed	No	
	Туре	Element Size	
	Element Size	0.2 m	
	Behavior	Hard	

Details of "Patch Conforming Method 3" - Method				
Ξ	Scope			
	Scoping Method	Geometry Selection		
	Geometry 1 Body			
Ξ	Definition			
	Suppressed	No		
	Method	Tetrahedrons		
	Algorithm	Patch Conforming		
	Element Midside Nodes	Use Global Setting		

Details of "Patch Conforming Method 2" - Method			
	Scope		
	Scoping Method	Geometry Selection	
Geometry 1 Body			
Definition			
	Suppressed	No	
	Method	Tetrahedrons	
	Algorithm	Patch Conforming	
	Element Midside Nodes	Use Global Setting	
	ose olobal setting		

Mesh Generation

Outline Project 🗄 🖶 🎯 Model (K3) 🔄 🖓 Geometry 🛶 📦 FAN HOLLOW 🗙 😭 Solid E Coordinate Systems Connections 🖻 🗸 🧒 Mesh Patch Conforming Method 2 Patch Conforming Method 3 Face Sizing - Named Selections TURBINE BOX WALLS 🖓 INLET OUTLET Cylinder Wall Inside Surface Cylinder wall outside on box Details of "Mesh" Defaults Physics Preference Mechanical Relevance 0 Sizing Use Advanced Size Function On: Proximity and Curvature Relevance Center Medium Initial Size Seed Active Assembly Cmeething Madium

	Smoothing	Medium
	Transition	Fast
	Span Angle Center	Medium
	Curvature Normal Angle	Default (45.0 °)
	Proximity Accuracy	0.5
	Num Cells Across Gap	Default (3)
	Min Size	Default (8.6073e-003 m)
	Max Face Size	Default (0.860730 m)
	Max Size	Default (1.72150 m)
	Growth Rate	Default (1.850)
	Minimum Edge Length	0.124670 m
÷	Inflation	
t	Advanced	
÷	Defeaturing	
-	Statistics	
	Nodes	247703
	Elements	168808
	Mesh Metric	None



Mesh Generation







In order to facilitate the selection of the boundary conditions, we will assign names to the different surfaces in the model. Notice the "Cylinder wall inside and outside surfaces", these were named in order to specify the mesh interaction. The mesh interaction set the parameters to allow the solution to flow through the interface.

Steady State Case

The purpose of the steady case in this analysis serves two main goals. First it will allow us to detect any problems with in the meshing, boundary conditions and other settings, and second it will allow us to observe the effect of the converging-diverging nozzle under steady state conditions, which is part of our study. For the steady state condition I will be using my original design, also note that the wind turbine is not rotating, since we have not yet applied the UDF and dynamic mesh settings.



In the general setting we selected: Type: Pressure Based, Velocity Formulation: Absolute, **Time: Steady**

In the Model window we selected only the laminar model to be on, every other model inclusion is set to be off. We will not be using the energy equation, since we are not dealing with compressible. We need to use the **laminar model** in order to appreciate the formation of vortices and take into account the losses due to internal friction of the fluid.

In the material section we selected the **density of air to be constant**. This is important to let the program know that the analysis is being performed using a non-compressible fluid.

Boundary conditions

Boundary Conditions			
Zone			
_turbine box_walls			
cylinder_wall_inside_surface cylinder_wall_outside_on_box inlet			
interior-6 interior-fan_hollow			
interior-solid nozzle outlet			
wall-14 wall-15			
Phase Type ID			
mixture v interface v 11			

Boundary conditions- All Cases Turbine was set to wall (stationary). **Box_walls** was set to symmetry. Cylinder walls were set to interface. **Inlet** was set to velocity Inlet with **5m/s** x-dir. Interrior-fan_hollow was set to interior. Interior Solid was set to interior. **Nozzle** was set to wall (stationary). **Outlet** was set to pressure outlet. The other boundary conditions in the list are

set automatically by the interface settings.

Symmetry boundary conditions are used when the geometry or the pattern of flow solution possesses mirror symmetry, but it can also be used to model zero-shear slip in viscous walls. Velocity Inlet and pressure outlet are adequate when dealing with incompressible flows.

Boundary conditions

		Create/Edit Mesh Interfaces		U	x
Velocity Inlet	×	Mesh Interface	Interface Zone 1	Interface Zone 2	
	Ì	c	cylinder_wall_inside_surface	cylinder_wall_outside_on_box	
Zone Name inlet					
Inter		c	cylinder_wall_inside_surface	cylinder_wall_inside_surface	
Momentum Thermal Radiation Species DPM Multiphase	JDS		cylinder_wall_outside_on_box	cylinder_wall_outside_on_box	
Velocity Specification Method Magnitude and Direction	▼				
Reference Frame Absolute	▼	Interface Options	Boundary Zone 1	Interface Wall Zone 1	
Velocity Magnitude (m/s)	constant 🗸	Periodic Boundary Condition	wall-14		
5	constant	Periodic Repeats	Boundary Zone 2	Interface Wall Zone 2	
Supersonic/Initial Gauge Pressure (pascal)	constant 🔹	Coupled Wall	wall-15		
Coordinate System Cartesian (X, Y, Z)				Interface Interior Zone	
	•			interior-6	
X-Component of Flow Direction -1	constant 👻	Periodic Boundary Condition			
Y-Component of Flow Direction	constant 🗸	Type Offset			
		Translational X (m)	Y (m) 0 Z (m) 0		
Z-Component of Flow Direction 0	constant 👻	Rotational			
		✓ Auto Compute Offset			
OK Cancel Help					
		Create	Delete Draw List Close	Help	

Average wind speeds ranges from 5 to 8.5 in most areas per year in the US.

Other Settings

Monitors
Residuals, Statistic and Force Monitors
Residuals - Print, Plot
Statistic - Off
Drag - Write
Lift - Write
Moment - Write
Edit

Solution Methods	
Pressure-Velocity Coupling	
Scheme	
SIMPLE 👻	
Spatial Discretization	
Gradient	^
Least Squares Cell Based 👻	
Pressure	
Standard 👻	
Momentum	
First Order Upwind 👻	
	Ŧ
Transient Formulation	
Non-Iterative Time Advancement Frozen Flux Formulation Pseudo Transient Default	

Solution Initialization	
Initialization Methods Hybrid Initialization Standard Initialization Compute from Inlet Reference Frame Relative to Cell Zone Absolute	•
Initial Values	_
Gauge Pressure (pascal) 0 X Velocity (m/s) -4.999998 Y Velocity (m/s) 0 Z Velocity (m/s) 0 0	
Initialize Reset Patch	

Moment Monitor	23
Options	Wall Zones
Print to Console Plot Window -1 Vindow Vindow File Name	turbine wall-14 wall-15
cm-historynonozzle	
Moment Center	
X (m) Y (m) Z (m) 0.01307 0.00045 4.2e-05	
Moment Axis	
X Y Z 1 0 0	Highlight Zones
OK Plot Clear	Cancel Help









Discussion for steady state case

We can see from fig 18 a dramatic increase in the velocity at the wind turbine. The free stream velocity is 5m/s while the maximum velocity at the inlet is of approximately 10m/s. We can also appreciate from these graphs the formation of vortices around the nozzle surface, and fluid separation from the walls of the nozzle. Vortices form behind the wind turbine due to the interference of the turbine blades which causes velocity discontinuities.





Transient Analysis

In the next four studies the same boundary conditions apply but there are few changes due to the introduction of a UDF program and the implementation of the dynamic mesh.

In the general setting we change the **time parameter to: Transient**.

Dynamic Mesh	Mesh Method Settings	Mesh Method Settings	
☑ Dynamic Mesh	Smoothing Layering Remeshing	Smoothing Layering Remeshing	
Mesh Methods Options Image: Settings Options Image: Settings Image: Settings Image: Settings Settings	Method Spring Diffusion Parameters Spring Constant Factor 0, 1 Boundary Node Relaxation 1 Convergence Tolerance 0,001	Remeshing Methods Sizing Function V Local Cell On V Local Face On CutCell Zone Variation 2.5D Rate Use Defaults Use Defaults	
Dynamic Mesh Zones box_walls - Stationary inlet - Stationary interior-fan_hollow - Rigid Body interior-solid - Deforming outlet - Stationary	Number of Iterations 20 Diffusion Parameter 0	Parameters Minimum Length Scale (m) 1e-05 Maximum Length Scale (m) 0.001 Maximum Cell Skewness 0.9 Maximum Face Skewness 0.7 Size Remeshing Interval 5 Mesh Scale Info Use Defaults	
Create/Edit Delete Delete All	OK Cancel Help	OK Cancel Help	

Transient Analysis

Dynamic Mesh Zones	Same	×	Solution Methods
Zone Names interior-fan_hollow	Dynamic Mesh Zones box_walls		Pressure-Velocity Coupling
Type Stationary Rigid Body Deforming User-Defined	inlet interior-fan_hollow interior-solid outlet		Scheme PISO Skewness Correction 1 Neighbor Correction 1
Motion Attributes Geometry Definition Meshin	g Options		Skewness-Neighbor Coupling
Adjacent Zone fan_hollow Co	ell Height (m) 0.001 constant		Spatial Discretization
Adjacent Zone fan_hollow C	ell Height (m) 0.001 constant		Gradient Least Squares Cell Based Pressure Standard Momentum
Create Draw	Delete All Delete Close H	elp	First Order Upwind 🔻

Dynamic Mesh Zones, Interior-fan_hollow is the cylinder that contains the turbine blades and it is given rigid Body motion governed by the UDF.

Transient Analysis

Run calculation details for all cases

Time step 0.2

Max iterations /Time Step = 50

Number of Steps = 25

Flow time = 5s

<u>UDF</u>

For our simulation we will use the Define_CG_Motion function in order to formulate the rotation of the blade. The function requires six arguments, time step, linear velocity, angular velocity, time and time step. We will define the initial linear velocity and angular velocity to zero.

We need to modify the UDF program given in the UDF manual to fit our case. The sample UDF is given for a piston, we will need to modify the program to allow for 1 rotational DOF.

De	Details View		
Ξ	Analysis Tools	i	
	Analysis Tool	Entity Information	
	Entity	Body	
	Body Type	Solid Body	
	Body Volume	0.011213 m ³	
	Body Area	0.9865 m ²	





<u>UDF</u>

```
#include "udf.h"
static real Ux_prev = 0.0;
```

```
DEFINE_CG_MOTION(turbine_rotation, dt, vel, omega, time, dtime)
{
Thread *t;
face_t f;
real NV_VEC (A); //defines a vector A[0]i + A[1]j + A[2]k
real force, du;
NV_S(vel, =, 0.0); //sets all velocity components to zero for x,y,z (eg vel[0]=0.0)
NV_S(omega, =, 0.0); //sets all rotations components to zero for x,y,z (eg omega[0]=0.0)
//omega[0] = 100;
if (!Data_Valid_P ()) //Prevents program execution if the enviroment is not set to avoid errors
return;
```

```
t = DT_THREAD (dt);
force = 0.0;
begin_f_loop(f, t)
{
        F_AREA (A, f, t);
        force += F_P (f, t) * NV_MAG (A); //force, penperdicular component to area of contact (dot product)
}
end_f_loop (f, t)
du = dtime * force /(0.011213*2719*0.9); // dtime * force/((volume*density(aluminum)*swept area radius)
Ux_prev += du;
Message ("\n time = %f, Ux_omega = %f, force = %f\n",time, Ux_prev, force);
omega[0] = Ux_prev; // [0] x dir, [1] y dir, [3] z dir
```









Pressure Contour 2 2.473e+001 1.912e+001 1.351e+001 7.898e+000 2.287e+000 -3.324e+000 -8.935e+000 -1.455e+001 -2.016e+001 -2.577e+001 -3.138e+001 [Pa]



<u>Case I – Nozzle Removed</u>

Rotation Vs. Flow Time (Nozzle Removed Configuration I)


Demonstration of Turbine Rotation



<u>Power Output</u>: All Values are taken at a 5 seconds flow time

Power Output: All Values are taken at a 5 seconds flow time

Console Data:

UDF output (refer to the attached file for entire console print out) time = 4.600000, Ux_omega = -21.242466, force = -194.213547 time = 4.800000, Ux_omega = -22.676086, force = -196.687820 time = 5.000000, Ux_omega = -24.228716, force = -213.015610

Force Report – Moment Axis (100)

 Zone
 Pressure
 Viscous
 Total

 turbine
 -0.22938613
 0.00020540987
 -0.2291807

$$P_{avaliable} = \frac{1}{2}A\rho V^3 = \frac{1}{2}\pi \left(\frac{1.8}{2}\right)^2 (1.224)5^3 = 194.6 \text{ watts}$$

$$P_{generated} = \tau \omega = Nmx \frac{rads}{sec} = 5.55 watts$$

$$Power \ Captured = \frac{P_{generated}}{P_{avaliable}} x100\% = 2.85\%$$





Pressure Contour 3 1.862e+001 4.241e+000 -1.014e+001 -2.453e+001 -3.891e+001 -5.329e+001 -6.767e+001 -8.206e+001 -9.644e+001 -1.108e+002 -1.252e+002 [Pa]





Rotation Vs. Flow Time (Nozzle Configuration II)

Power Output: All Values are taken at a 5 seconds flow time

Console Data:

UDF output (refer to the attached file for entire console print out) time = 4.600000, Ux_omega = -1021.215942, force = -4816.365723 time = 4.800000, Ux_omega = -1056.742920, force = -4874.176758 time = 5.000000, Ux_omega = -1092.648193, force = -4926.077637

Force Report – Moment Axis (100)

Zone	Pressure	Viscous	Total
turbine	-0.51939837	0.00021137946	5 -0.51918699

$$P_{avaliable} = \frac{1}{2}A\rho V^3 = \frac{1}{2}\pi \left(\frac{1.8}{2}\right)^2 (1.224)8.7^3 = 1025.52 \text{ watts}$$

$$P_{generated} = \tau \omega = Nmx \frac{rads}{sec} = 567.29 watts$$

$$Power \ Captured = \frac{P_{generated}}{P_{avaliable}} x100\% = 26\%$$











Rotation Vs. Flow Time (Nozzle Configuration III)

Power Output: All Values are taken at a 5 seconds flow time

Console Data:

UDF output (refer to the attached file for entire console print out) time = 4.600000, Ux_omega = -1066.474976, force = -4879.613281 time = 4.800000, Ux_omega = -1100.707886, force = -4696.633301 time = 5.000000, Ux_omega = -1133.746826, force = -4532.828125

Force Report – Moment Axis (100)

Zone	Pressure	Viscous	Total
turbine	-0.39420759	-9.7518579e-0	5 -0.39430511

$$P_{avaliable} = \frac{1}{2}A\rho V^3 = \frac{1}{2}\pi \left(\frac{1.8}{2}\right)^2 (1.224)8.16^3 = 846.16 \, watts$$

$$P_{generated} = \tau \omega = Nmx \frac{rads}{sec} = 447 watts$$

$$Power \ Captured = \frac{P_{generated}}{P_{avaliable}} x100\% = 52.8\%$$





Pressure Contour 2 2.632e+001 1.416e+001 1.992e+000 -1.017e+001 -2.234e+001 -3.450e+001 -4.667e+001 -5.884e+001 -7.100e+001 -8.317e+001 -9.533e+001



Rotation (rad/s)

Time (s)

Rotation Vs. Flow Time (Nozzle Configuration IV)

Power Output: All Values are taken at a 5 seconds flow time

Console Data:

UDF output (refer to the attached file for entire console print out) time = 4.600000, Ux_omega = -1189.663208, force = -4250.572266 time = 4.800000, Ux_omega = -1220.224854, force = -4192.950684 time = 5.000000, Ux_omega = -1250.372559, force = -4136.159180

Force Report – Moment Axis (100)

Zone	Pressure	Viscous	Total
turbine	-0.49006197	0.00020831742	2 -0.48985365

$$P_{avaliable} = \frac{1}{2}A\rho V^3 = \frac{1}{2}\pi \left(\frac{1.8}{2}\right)^2 (1.224)8^3 = 797.36 \text{ watts}$$

$$P_{generated} = \tau \omega = Nmx \frac{rads}{sec} = 612.43 watts$$

$$Power \, Captured = \frac{P_{generated}}{P_{avaliable}} \, x100\% = 76.80\%$$

Discussion & Results

The results show that the Nozzles increase the amount of power generated by the wind turbine. As the diameter and length of the converging nozzle increased so did the amount of power generated and the percentage of the power captured from the power available.

	Power	Power	Power
Setup	Available	Generated	Captured
No			
Nozzle	194.6 watts	5.55 watts	2.85%
Setup 1	1025.52 watts	567.29 watts	26%
Setup 2	846.16 watts	447 watts	52.8%
Setup 3	797.36 watts	612.43 watts	76.8%

Conclusion and Improvements





Conclusion and Improvements

We found that the application of the converging-diverging nozzles improves the power output of wind turbines. Better results could be obtained by using a finer mesh, a smaller time step and more iterations. Whether or not the manufacture of wind turbine blades with converging-diverging nozzles is cost effective is another question. Wind turbines in general require a moderate amount of maintenance, adding a converging-diverging nozzle will significantly increase the initial cost of the turbine. The application of this nozzle can be particular useful in areas of low to moderate wind speeds where the need for electricity outweighs the maintenance cost and initial investment.

<u>References</u>

Fluent UDF manual

Fluent Tutorial Manual

David Hartwanger and Dr Andrej Horvat - 3D MODELLING OF A WIND TURBINE USING CFD . UK 2008

Michael Moeller Jr., and Dr. Kenneth Visser- Analysis of Diffuser Augmented Wind Turbines, Clarkson University 2011

Websites

How to calculate wind turbine power http://www.smallwindtips.com/2010/01/how-to-calculate-wind-power-output/ 2011

Calculation of wind Power - http://www.reuk.co.uk/Calculation-of-Wind-Power.htm , 2006 – 2011

Dynamic Mesh Zones Dialog Box -

https://www.sharcnet.ca/Software/Fluent12/html/ug/node1142.htm 2009

The End