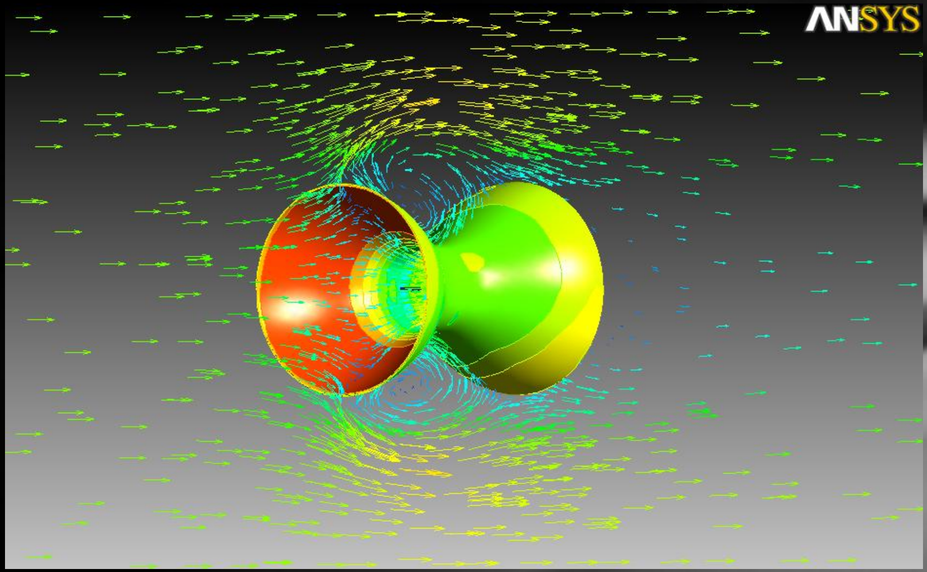
The City College of New York
Department of Mechanical Engineering
Applied Fluid Mechanics
Professor: Dr. Yiannis Andreopoulos



Diffuser Augmented Wind Turbine

Presented By: Jose Cortes

Abstract

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

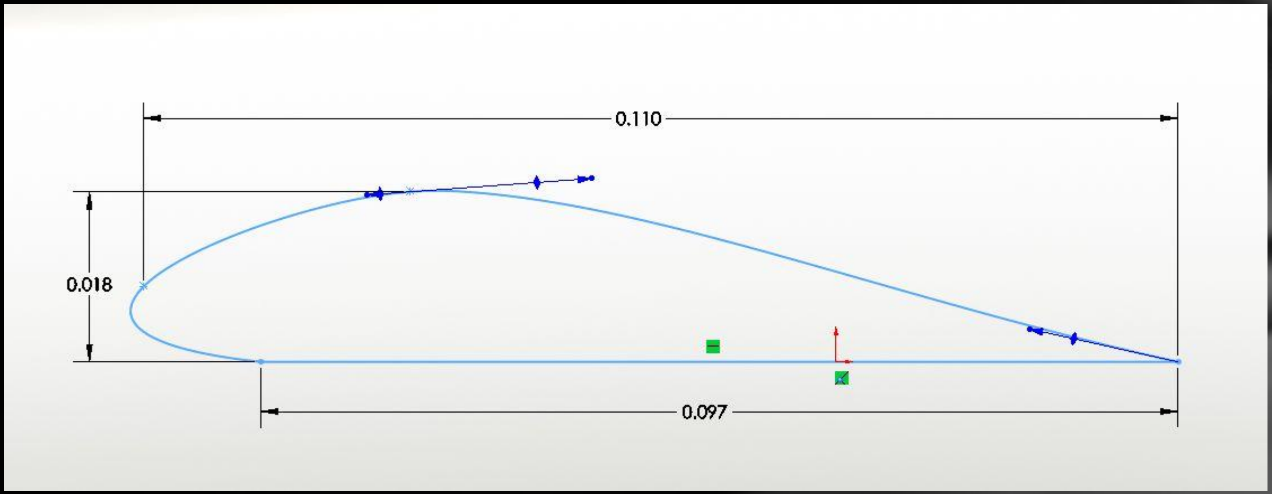
In this paper we seek to analyze a diffuser augmented wind turbine using a computational fluid dynamic approach. For our analysis we will be using Ansys-Fluent. The set up for the wind turbine consist of: **a converging nozzle which draws the air inside a cylinder increasing the velocity of the incoming 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.

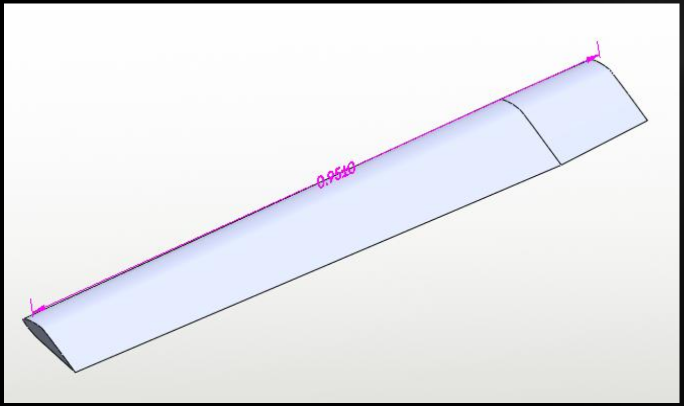
In this paper I will investigate the effectiveness of the converging-diverging nozzle adaptation in reference to a bare wind turbine model, and then I will try to improve upon my original design by increasing the inlet size and length at the converging nozzle and comparing the power generated with the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

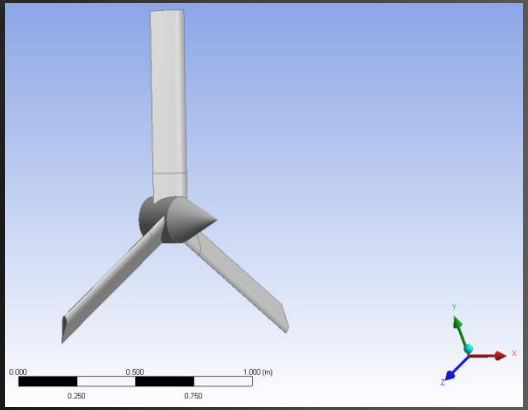
Background

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.

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

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}

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

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.

$$\dot{m} = VxAx\rho = \frac{m}{s}m^2\frac{kg}{m^3} = \frac{kg}{s} \quad Eq. 6$$

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.

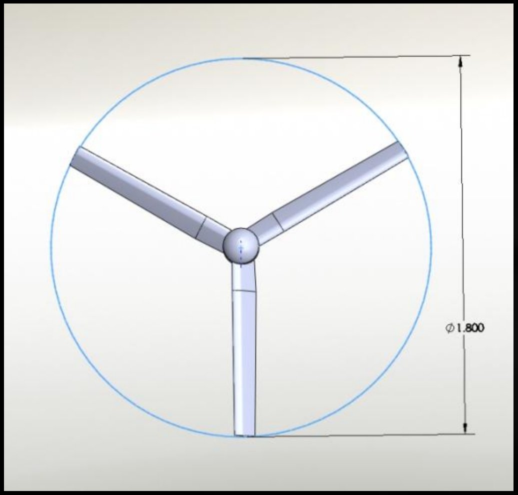
The power is calculated by inserting the mass flow rate into Eq. 5, resulting in the following equation:

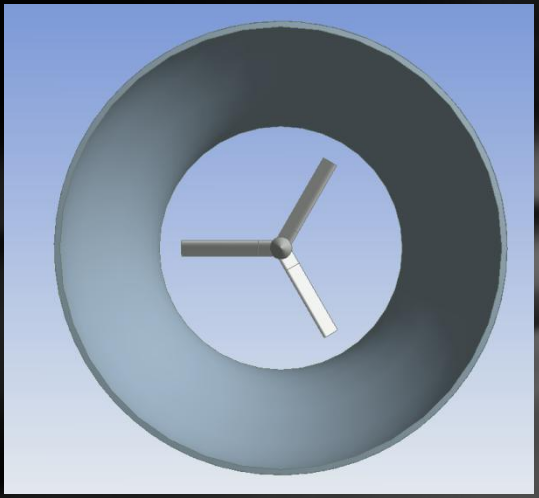
$$P = \frac{1}{2}A\rho V^3 = watts$$

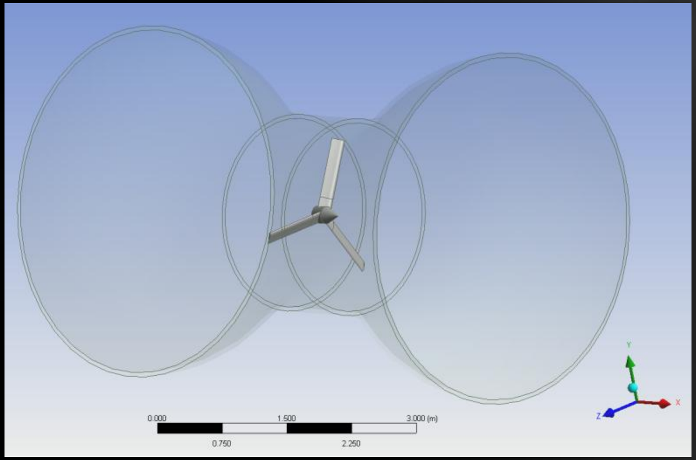
This is the ideal power generated by the wind, real life power generation in wind turbines can range from 0.25P to 0.45P.

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.

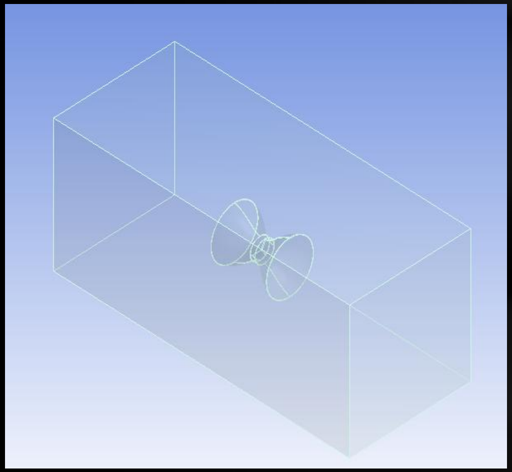


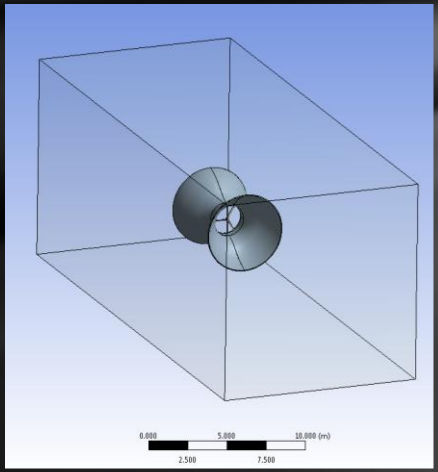




Ansys Design Modeler Setup

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

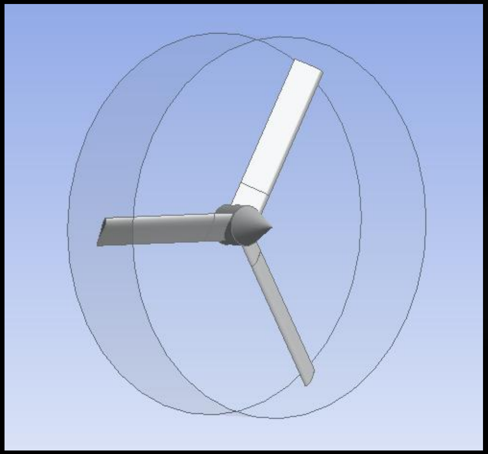
This modification was implemented after being unsuccessful in getting a solution in the transient model using a dynamic mesh. The problem has to do with the large displacements that occur as the wind turbine rotates. Fluent provides smoothing to deal with small displacements and remeshing to deal with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

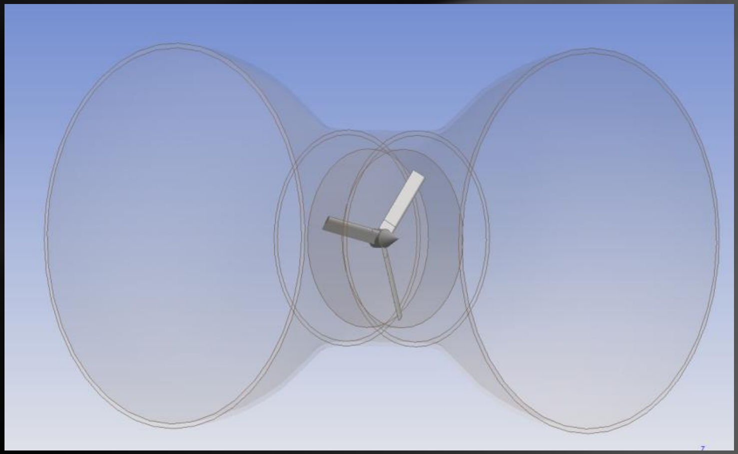
Smoothing introduces spring like characteristics to the elements, allowing them to deform as the model undergoes small displacement. Unfortunately for large displacement smoothing does not work well, producing a negative cell error during the program execution.

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.

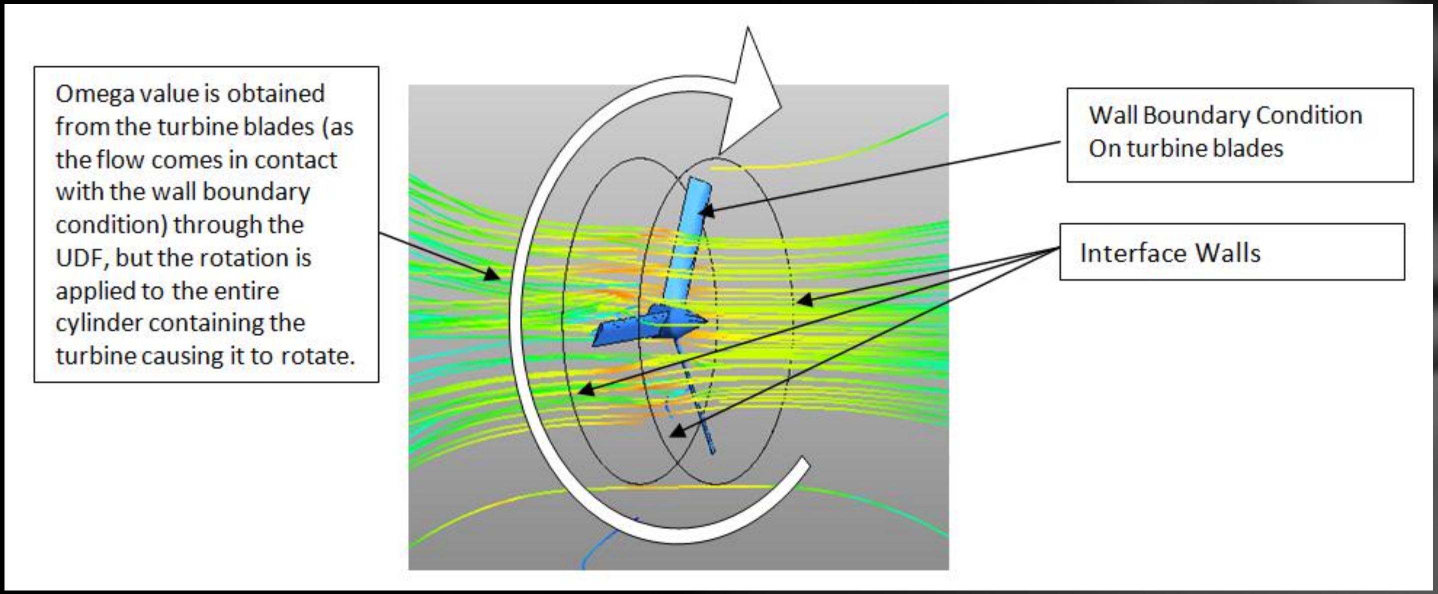
Solid Model Modification due problems with large blade displacements

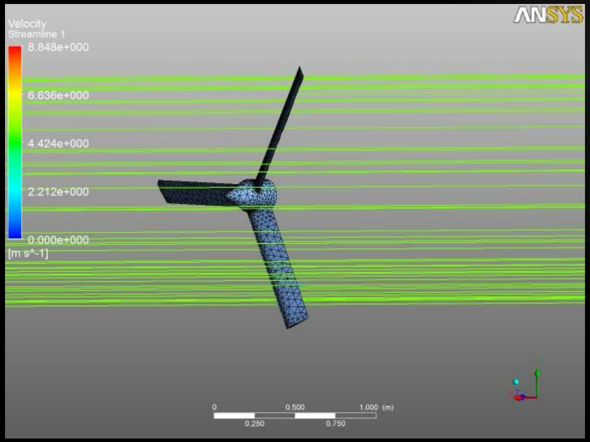
In order to eliminate negative volume errors, it was necessary to enclose the entire turbine blade inside a cylindrical volume. **The purpose is to rotate the volume that encloses the turbine blades and not the turbine blades inside the mesh. The UDF will compute the components of the force hitting the 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.**

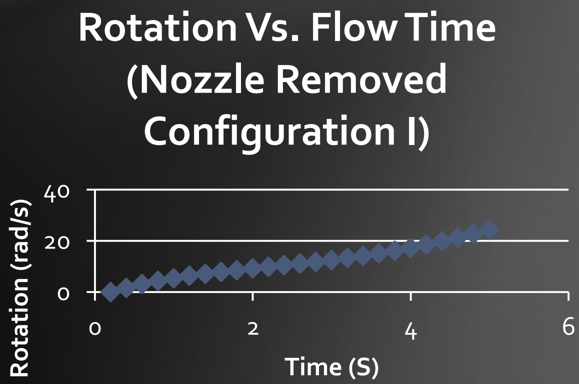




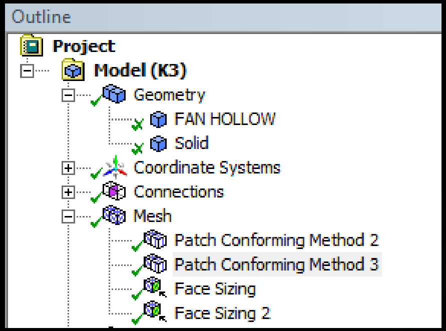
Solid Model Modification due problems with large blade displacements







Mesh Generation



Details of "Face Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	6 Faces
Definition	
Suppressed	No
Type	Element Size
Element Size	0.1 m
Behavior	Soft
Curvature Normal Angle	Default
Growth Rate	Default

Details of "Face Sizing 2" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	8 Faces
Definition	
Suppressed	No
Type	Element Size
Element Size	0.2 m
Behavior	Hard

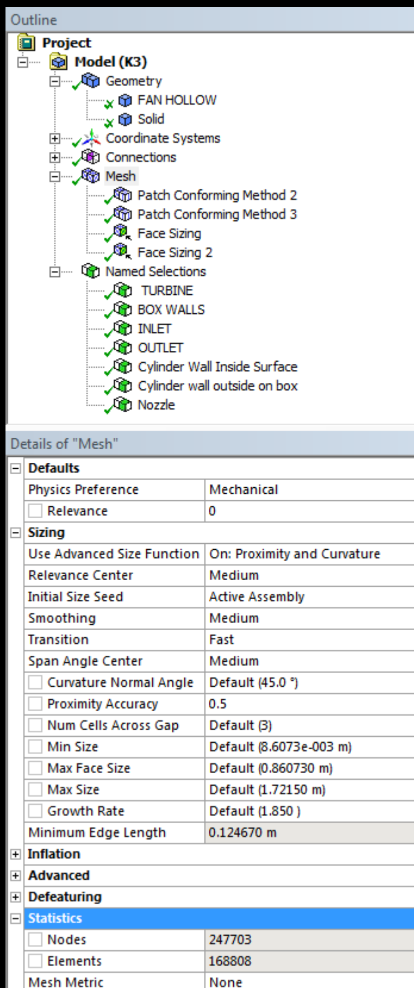
Details of "Patch Conforming Method 3" - Method

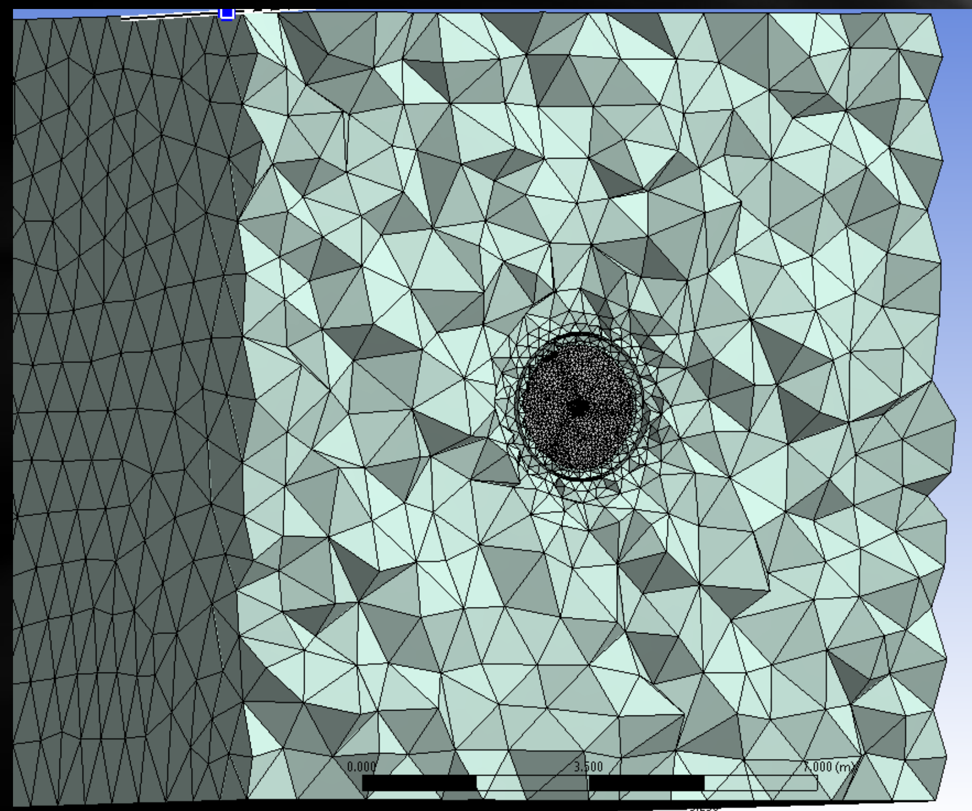
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	Tetrahedrons
Algorithm	Patch Conforming
Element Midside Nodes	Use Global Setting

Details of "Patch Conforming Method 2" - Method

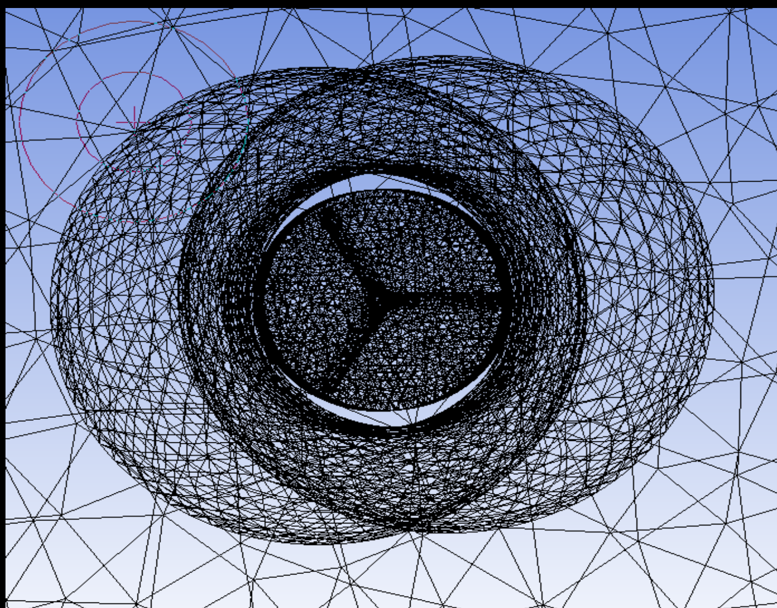
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	Tetrahedrons
Algorithm	Patch Conforming
Element Midside Nodes	Use Global Setting

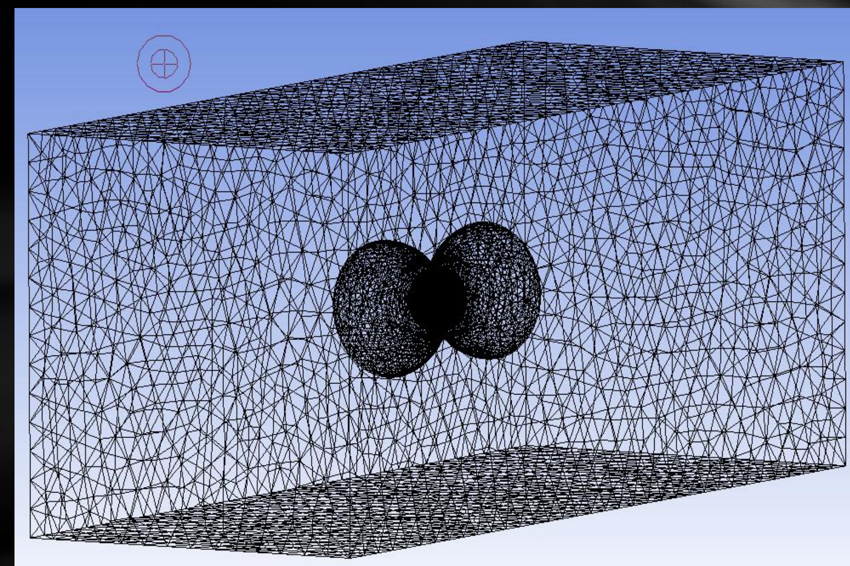
Mesh Generation

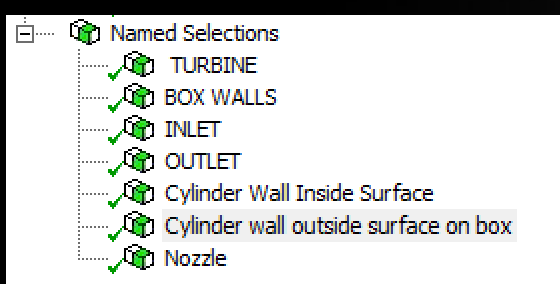




Mesh Generation



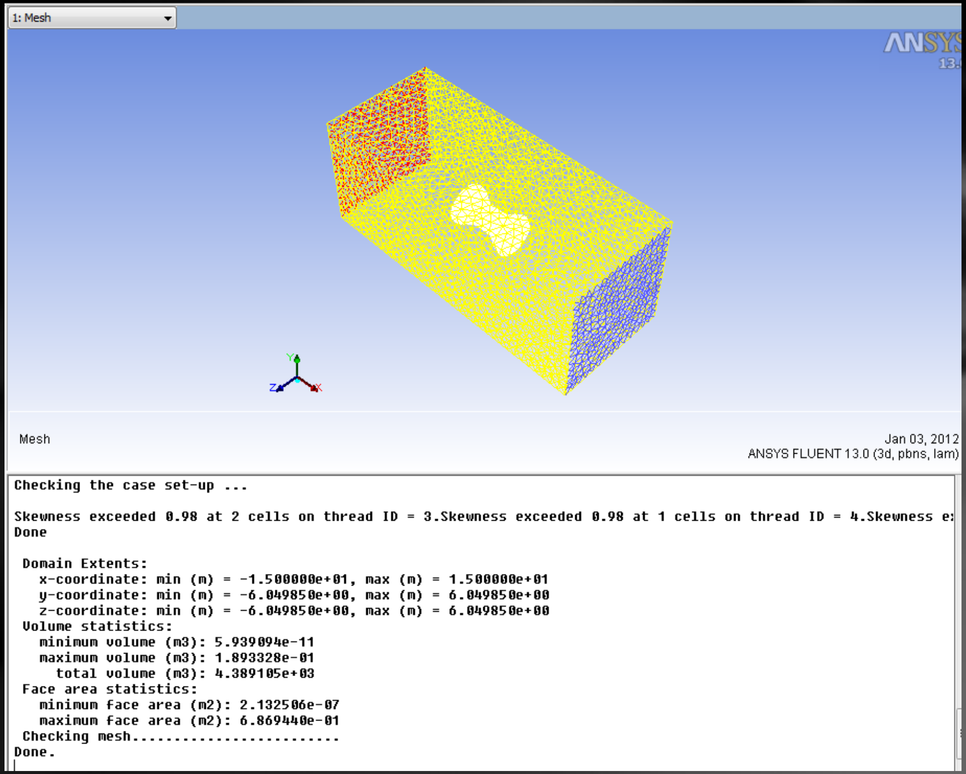




In order to facilitate the selection of the boundary conditions, we will assign names to the different surfaces in the model. Notice the “Cylinder wall inside and outside surfaces”, these were named in order to specify the mesh interaction. The mesh interaction set the parameters to allow the solution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



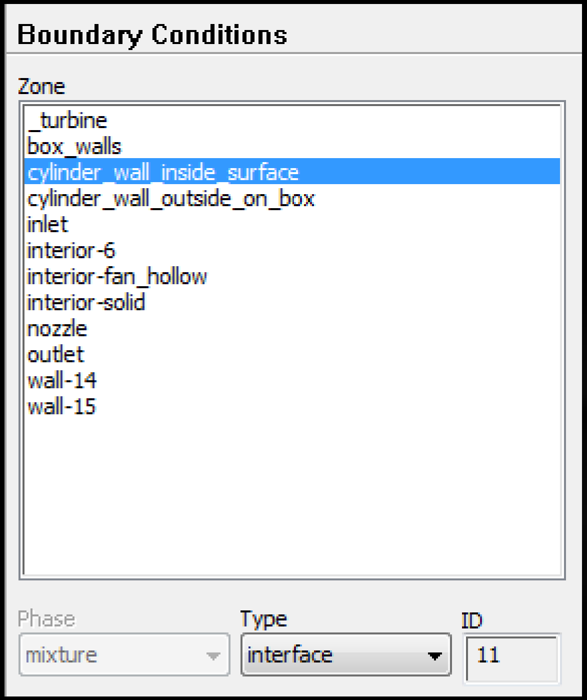
General Settings

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- 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.

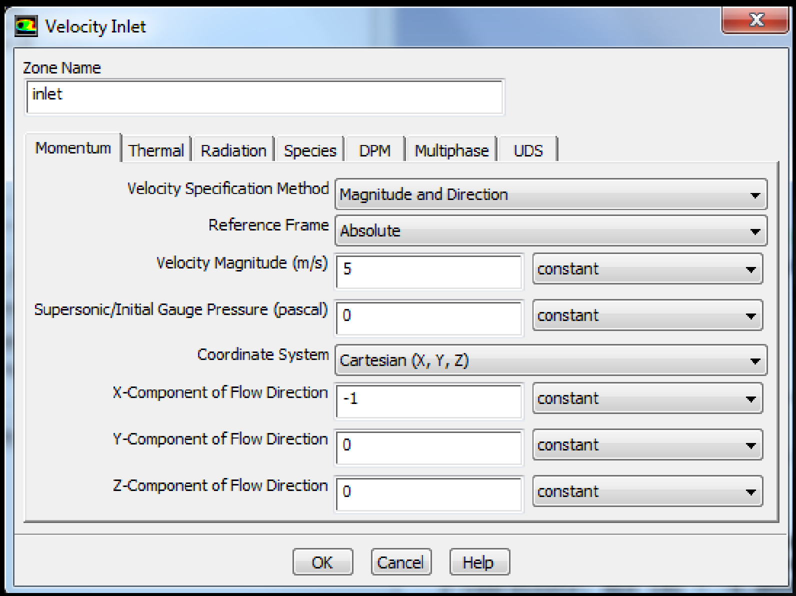
Nozzle was set to wall (stationary).

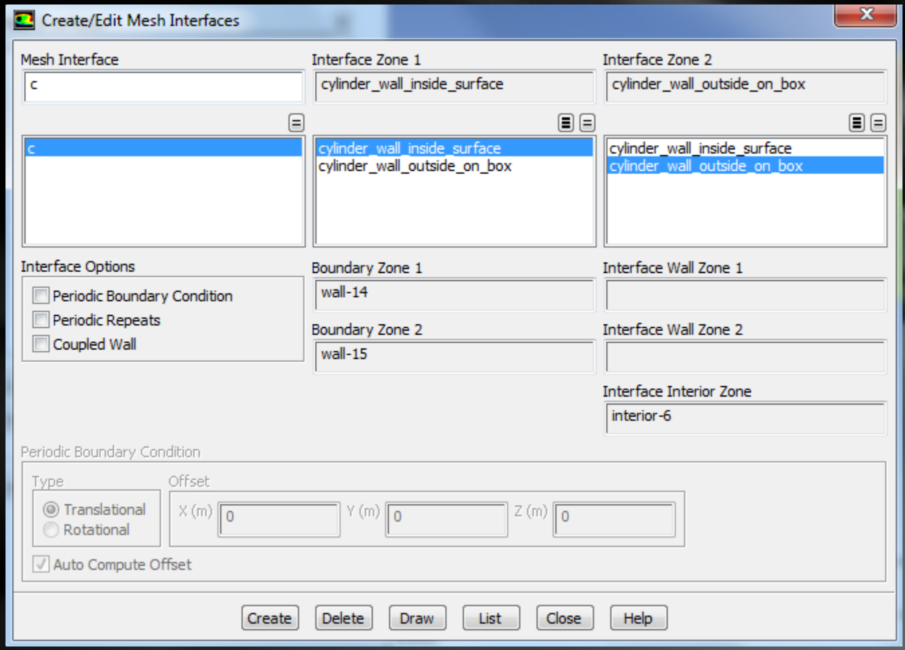
Outlet was set to pressure outlet.

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.

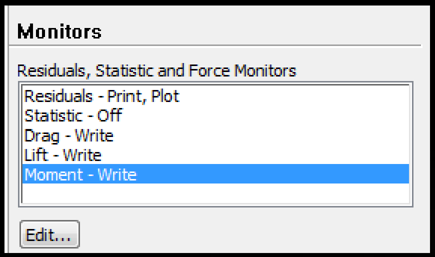
Boundary conditions

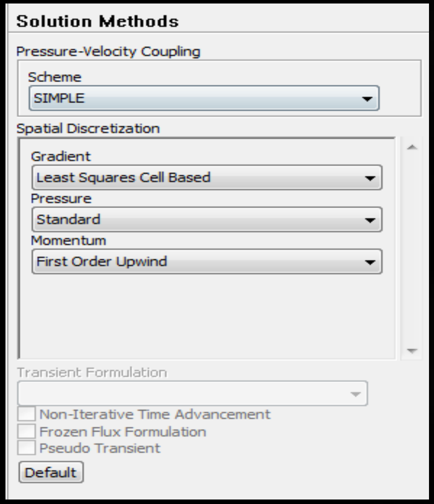


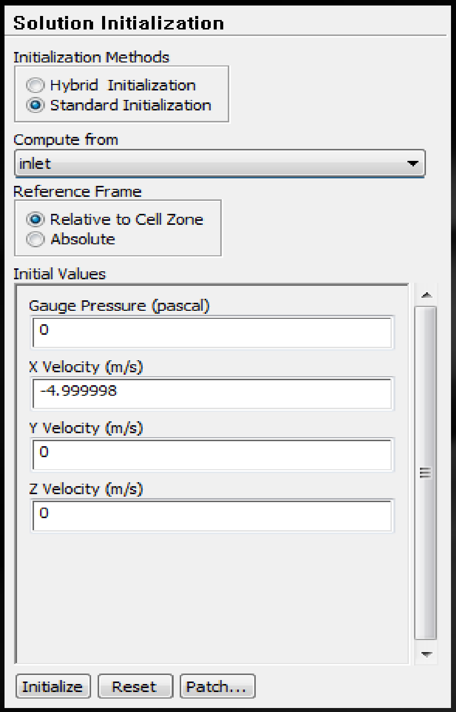


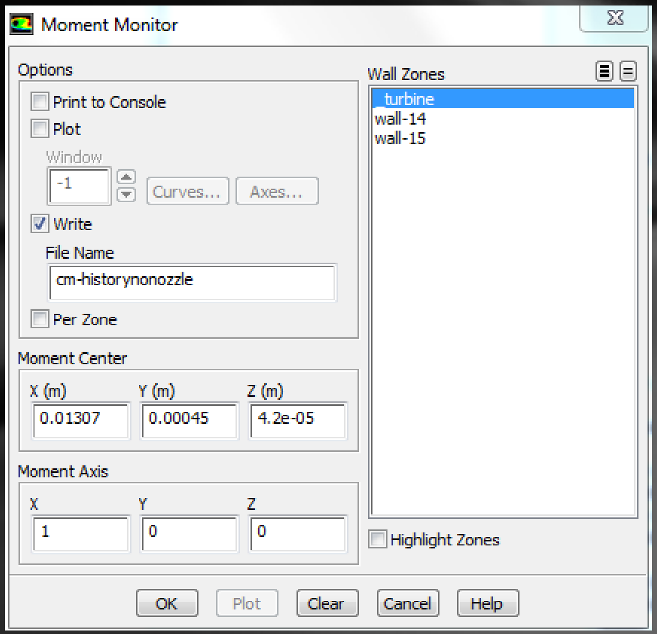
Average wind speeds ranges from 5 to 8.5 in most areas per year in the US.

Other Settings

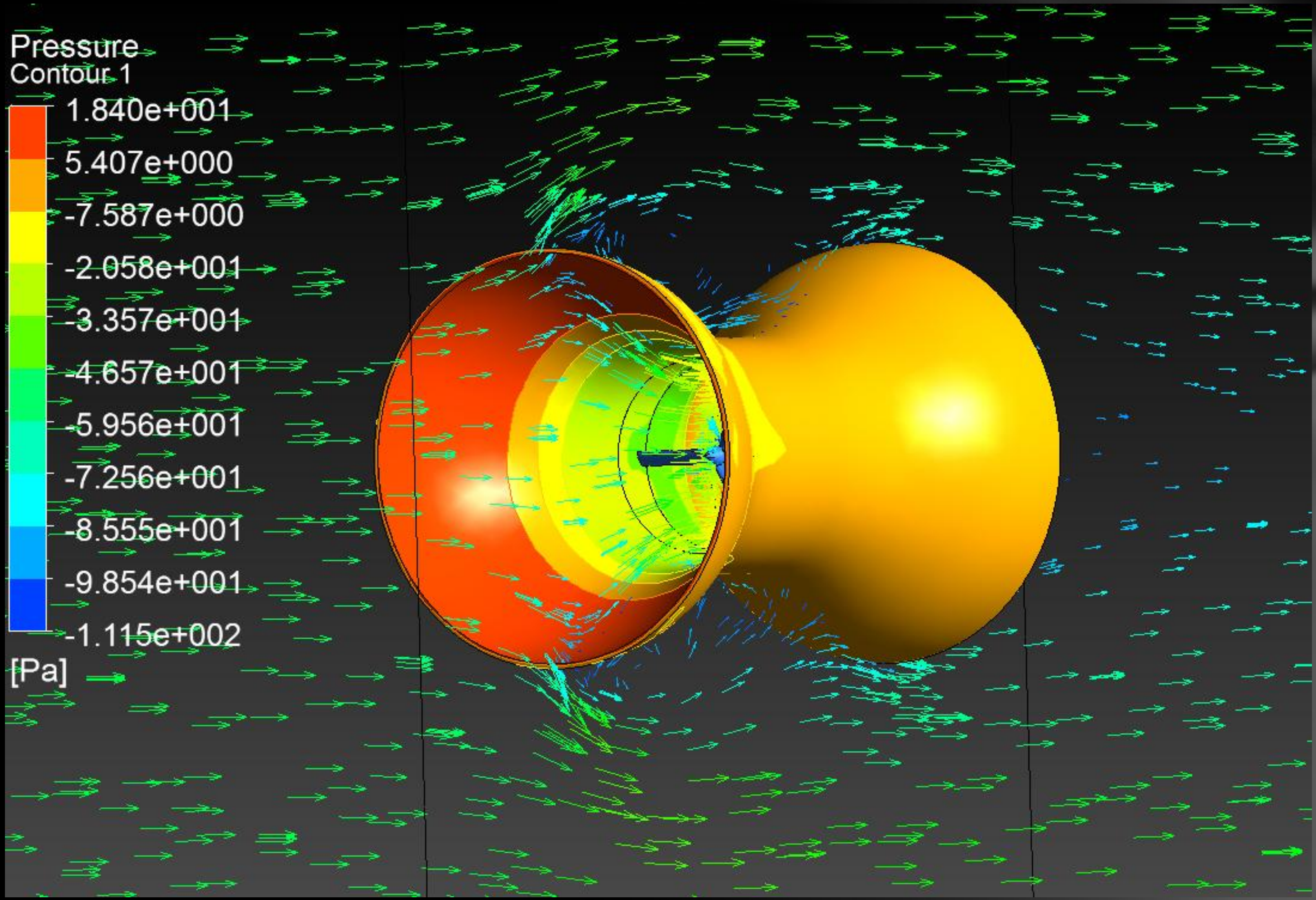




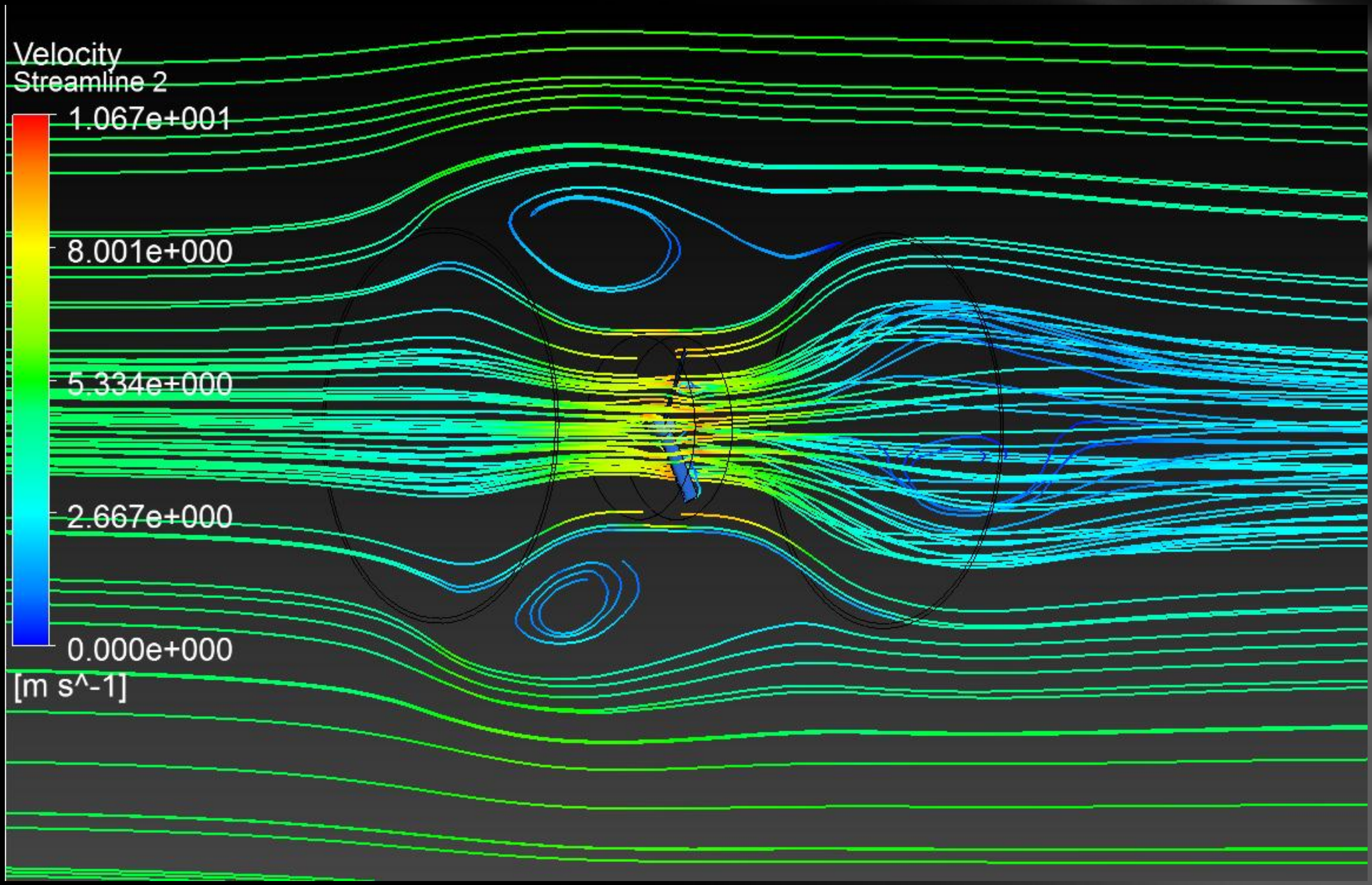




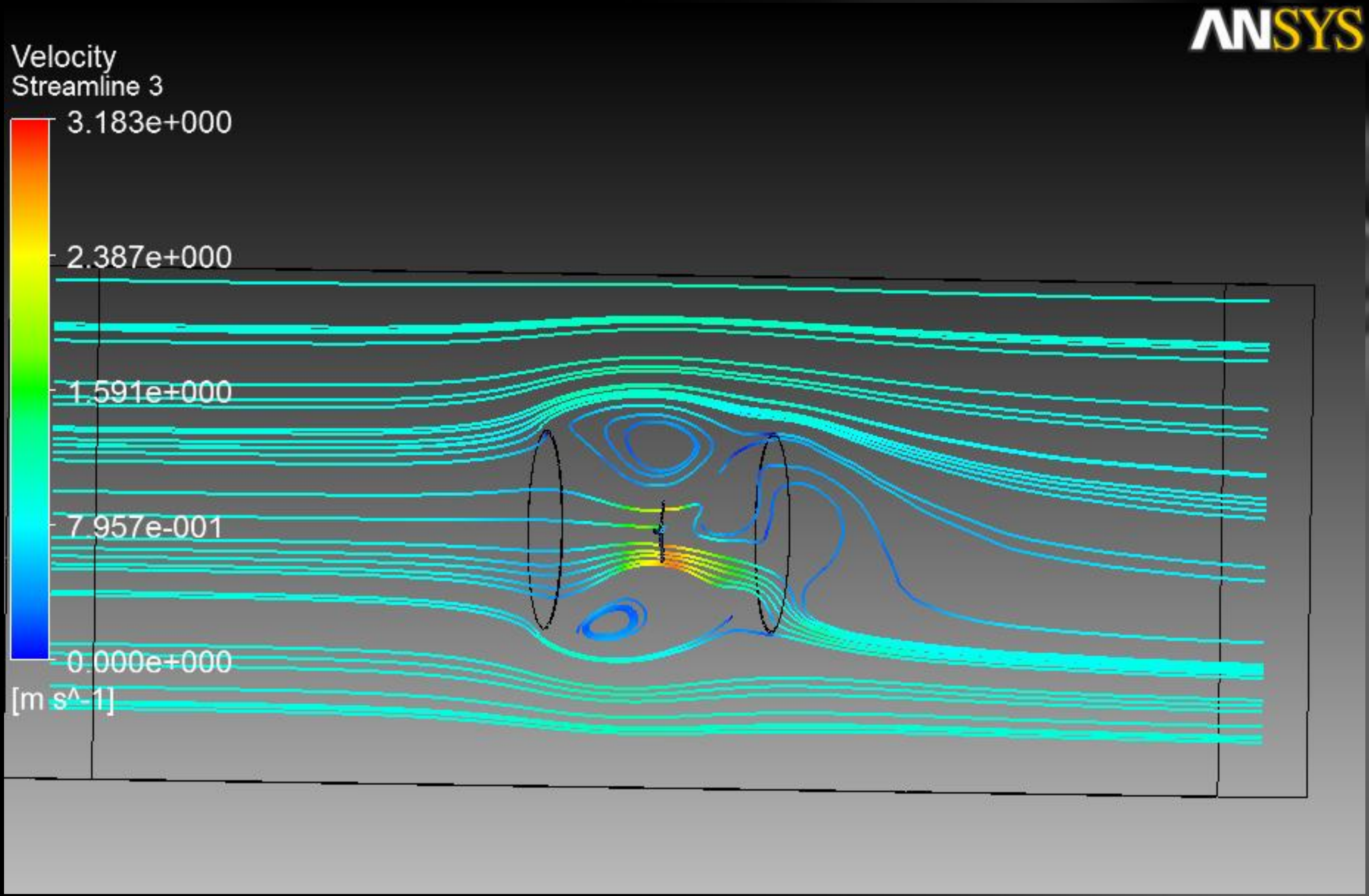
STEADY STATE RESULTS



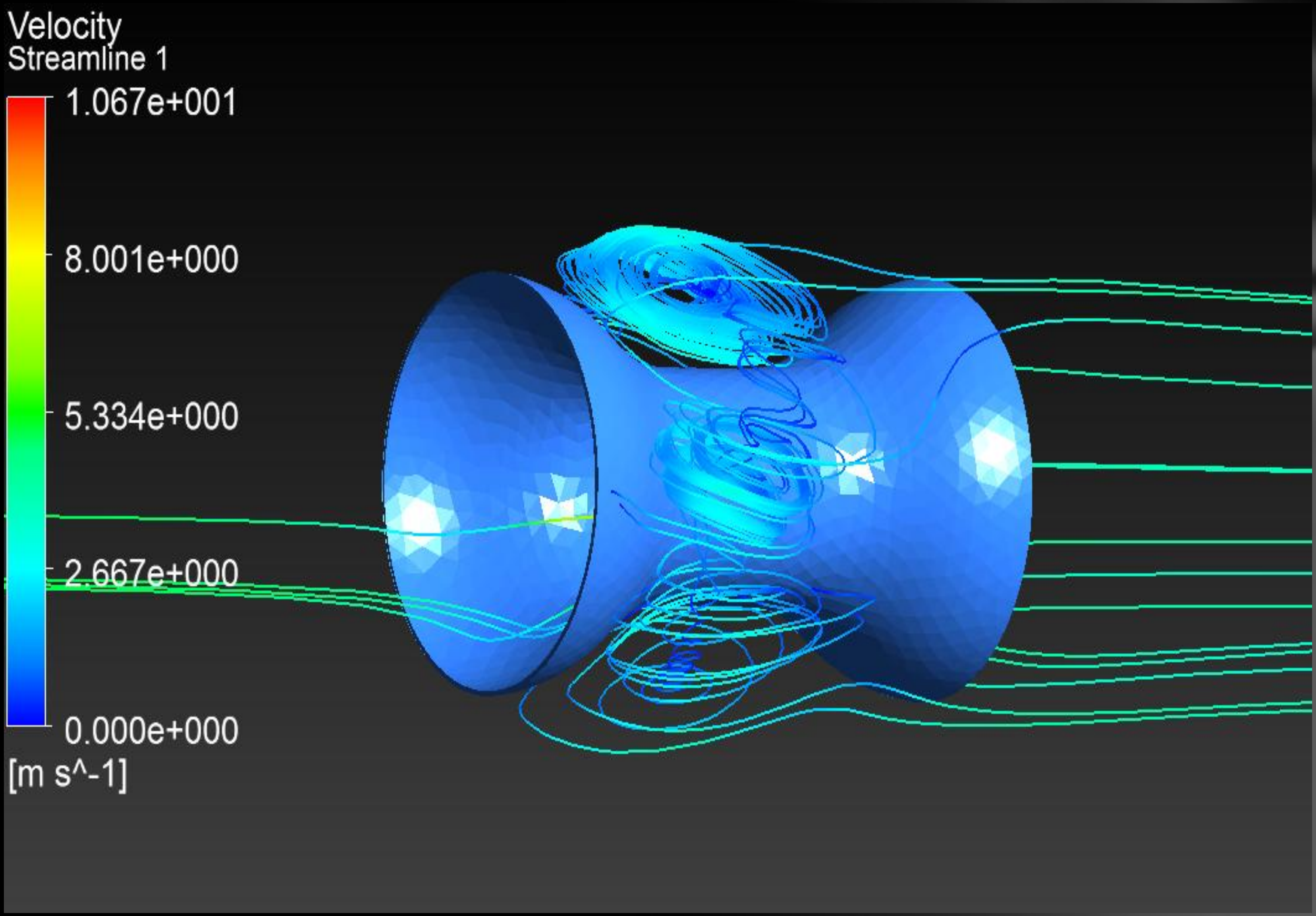
STEADY STATE RESULTS



STEADY STATE RESULTS



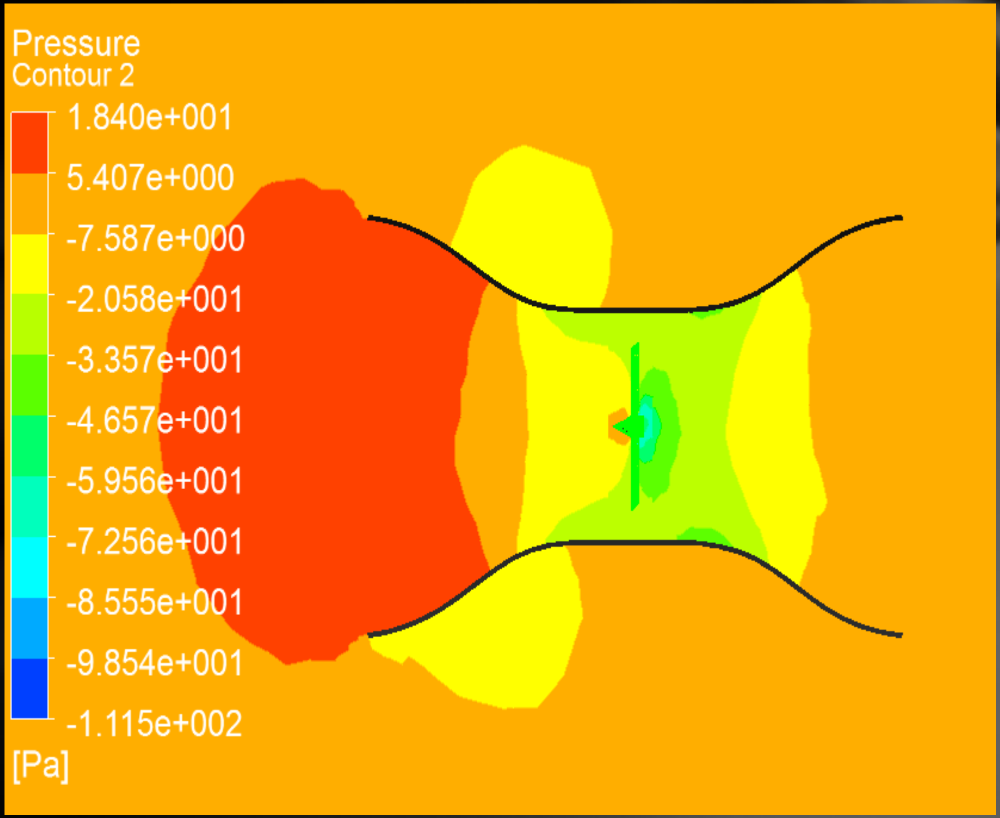
STEADY STATE RESULTS



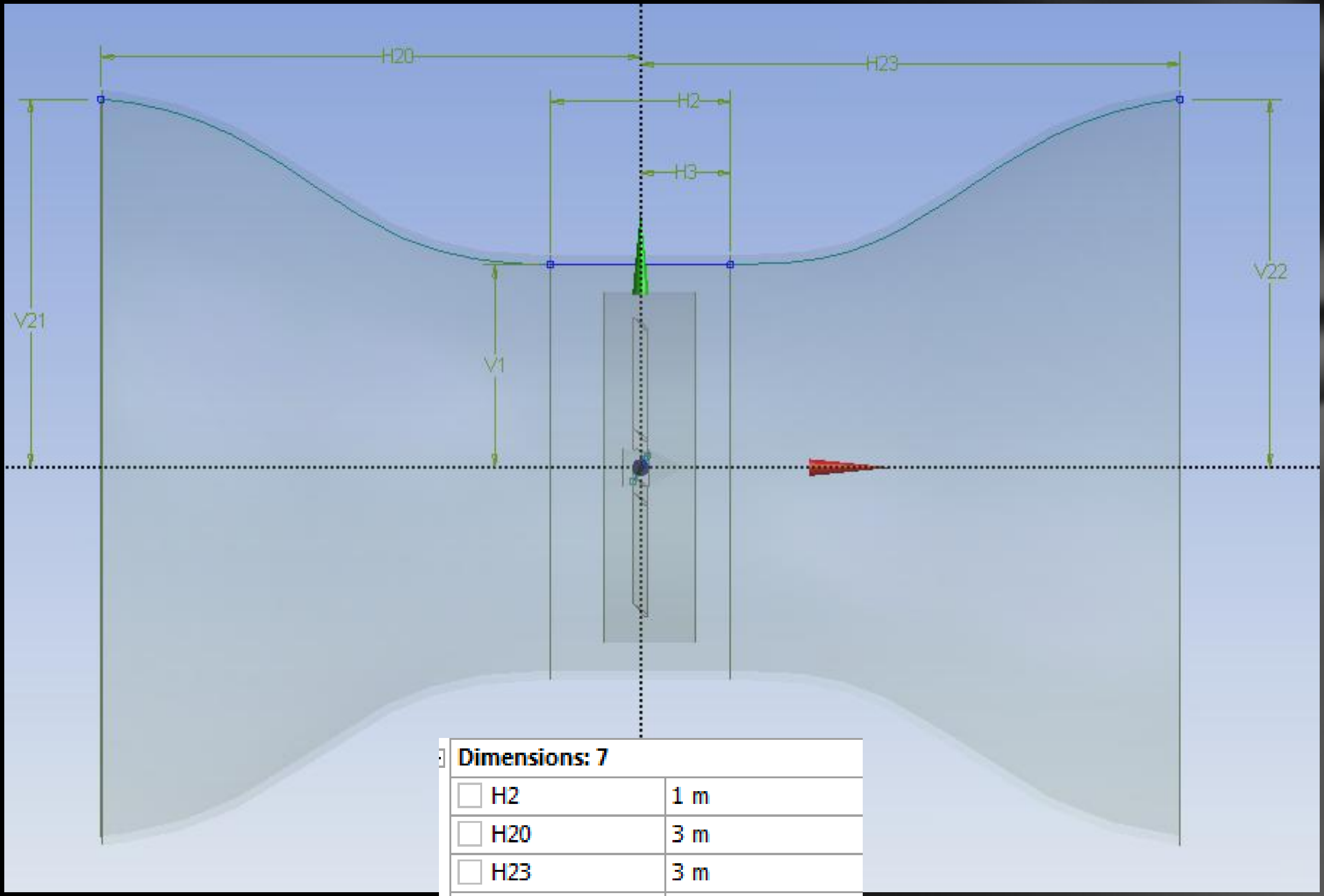
STEADY STATE RESULTS

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.



STEADY STATE RESULTS

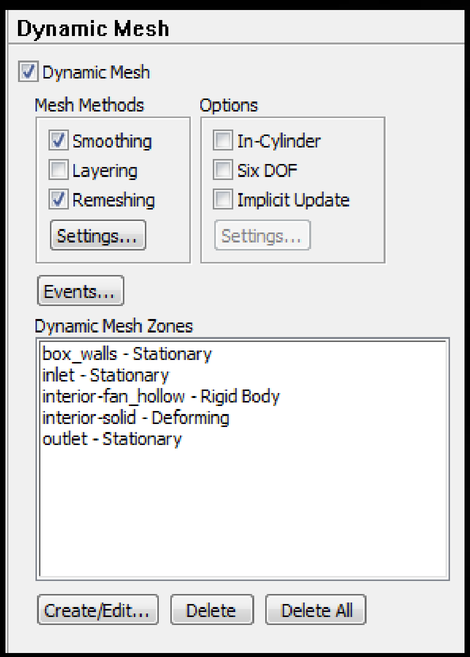


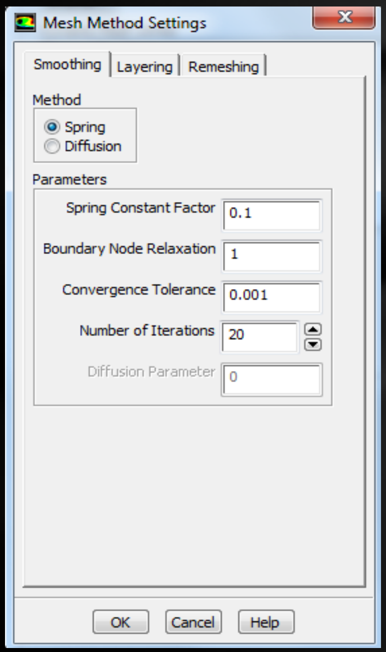
Dimensions: 7	
H2	1 m
H20	3 m
H23	3 m
H3	0.5 m
V1	1.1 m
V21	2 m
V22	2 m

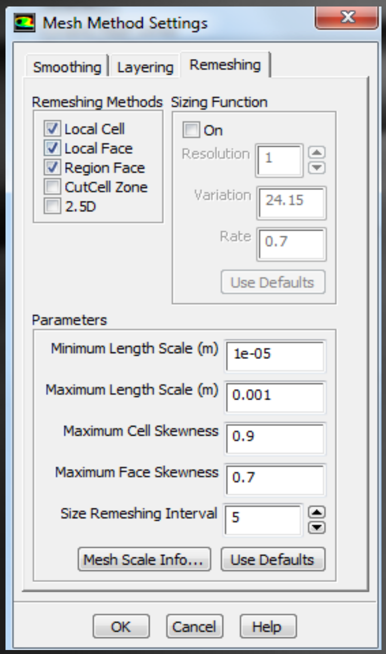
Transient Analysis

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.

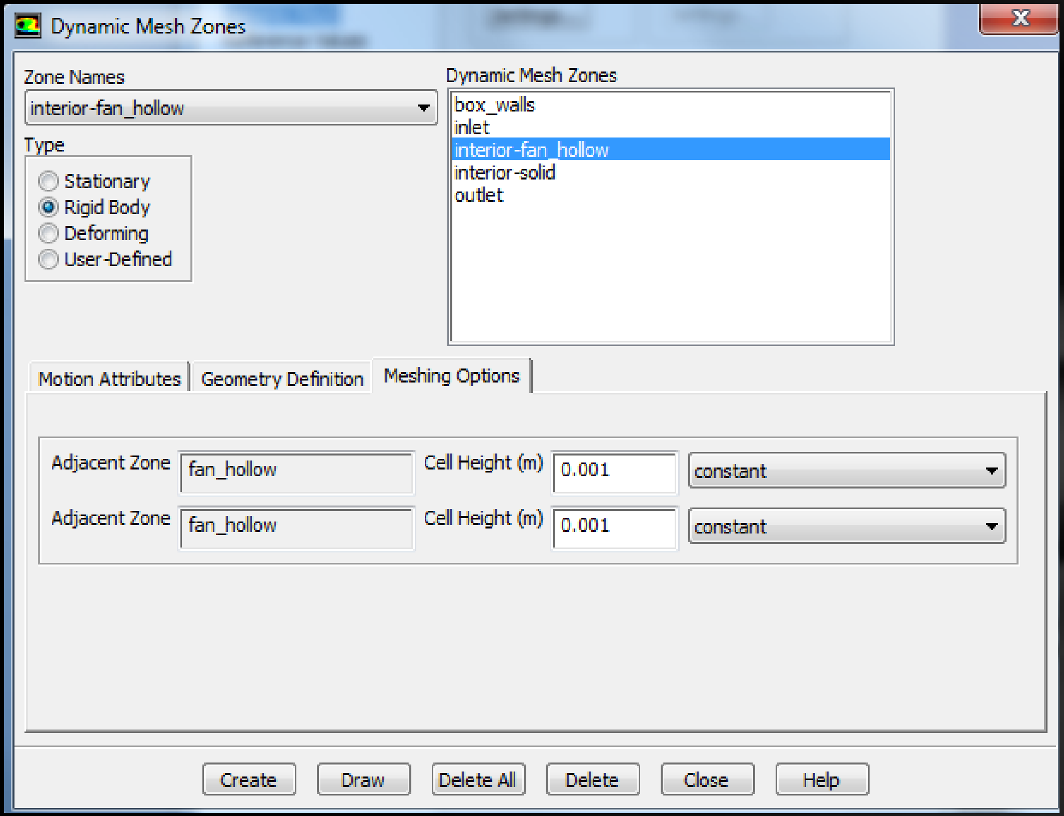
In the general setting we change the **time parameter to: Transient.**

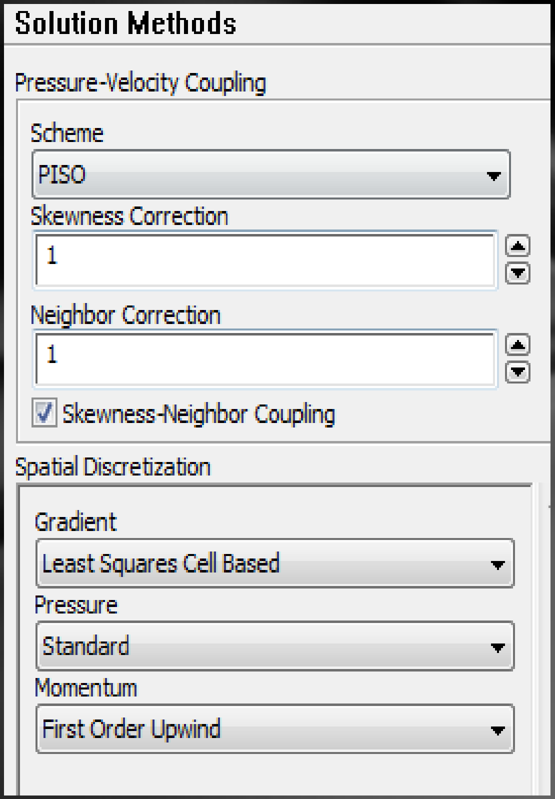






Transient Analysis





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

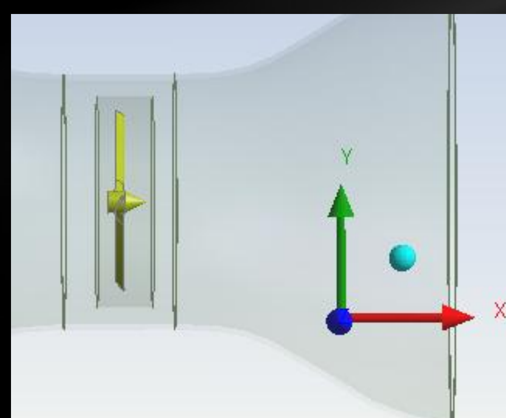
UDF

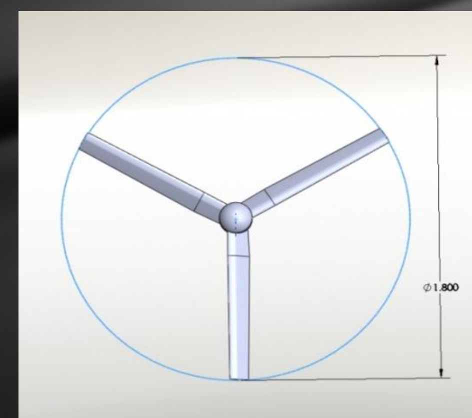
For our simulation we will use the Define_CG_Motion function in order to formulate the rotation of the blade. The function requires six arguments, time step, linear velocity, angular velocity, time and time step. We will define the initial linear velocity and angular velocity to zero.

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.

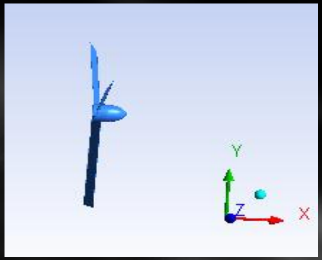
Details View

Analysis Tools	
Analysis Tool	Entity Information
Entity	Body
Body Type	Solid Body
Body Volume	0.011213 m^3
Body Area	0.9865 m^2



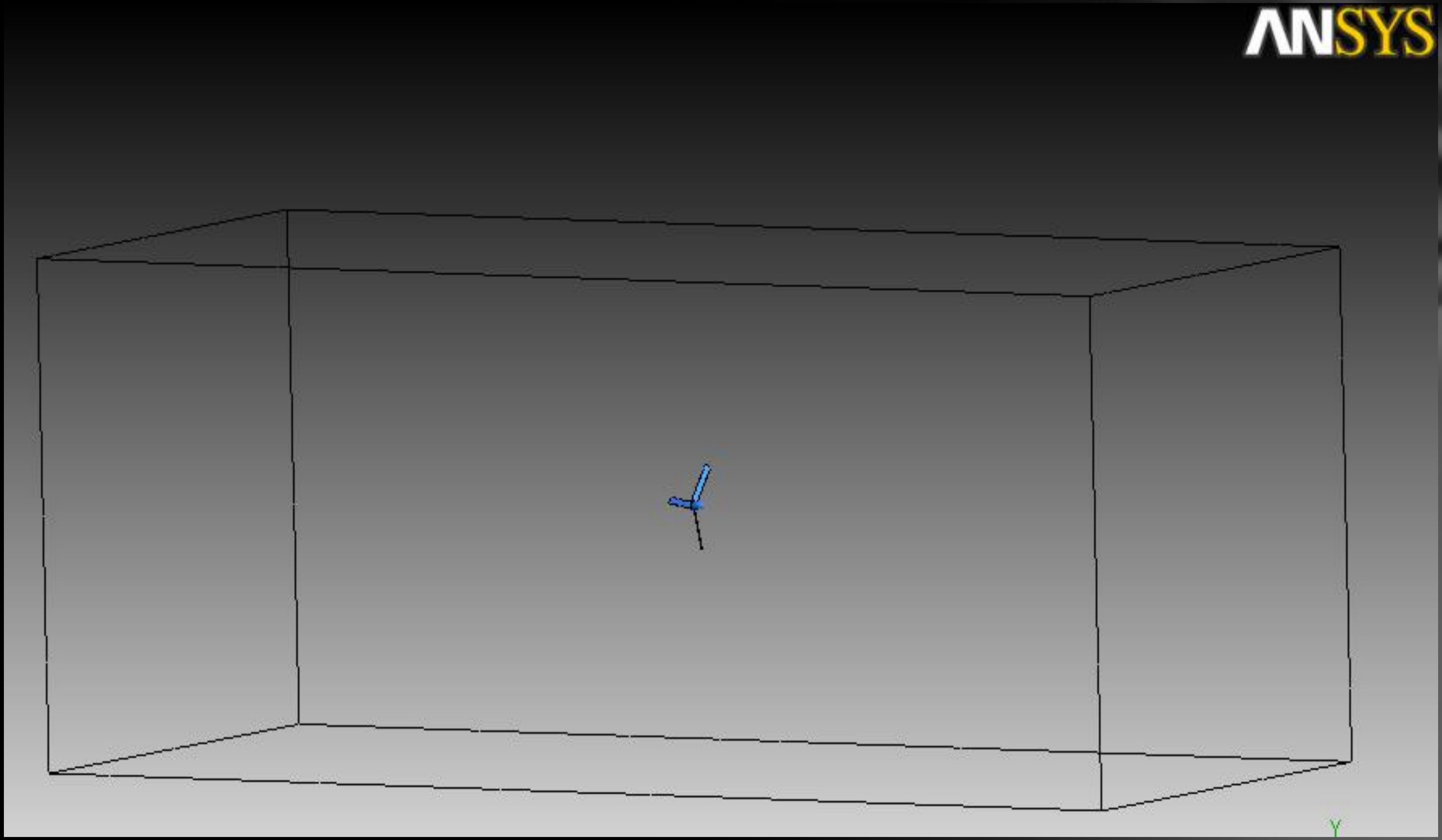


UDF

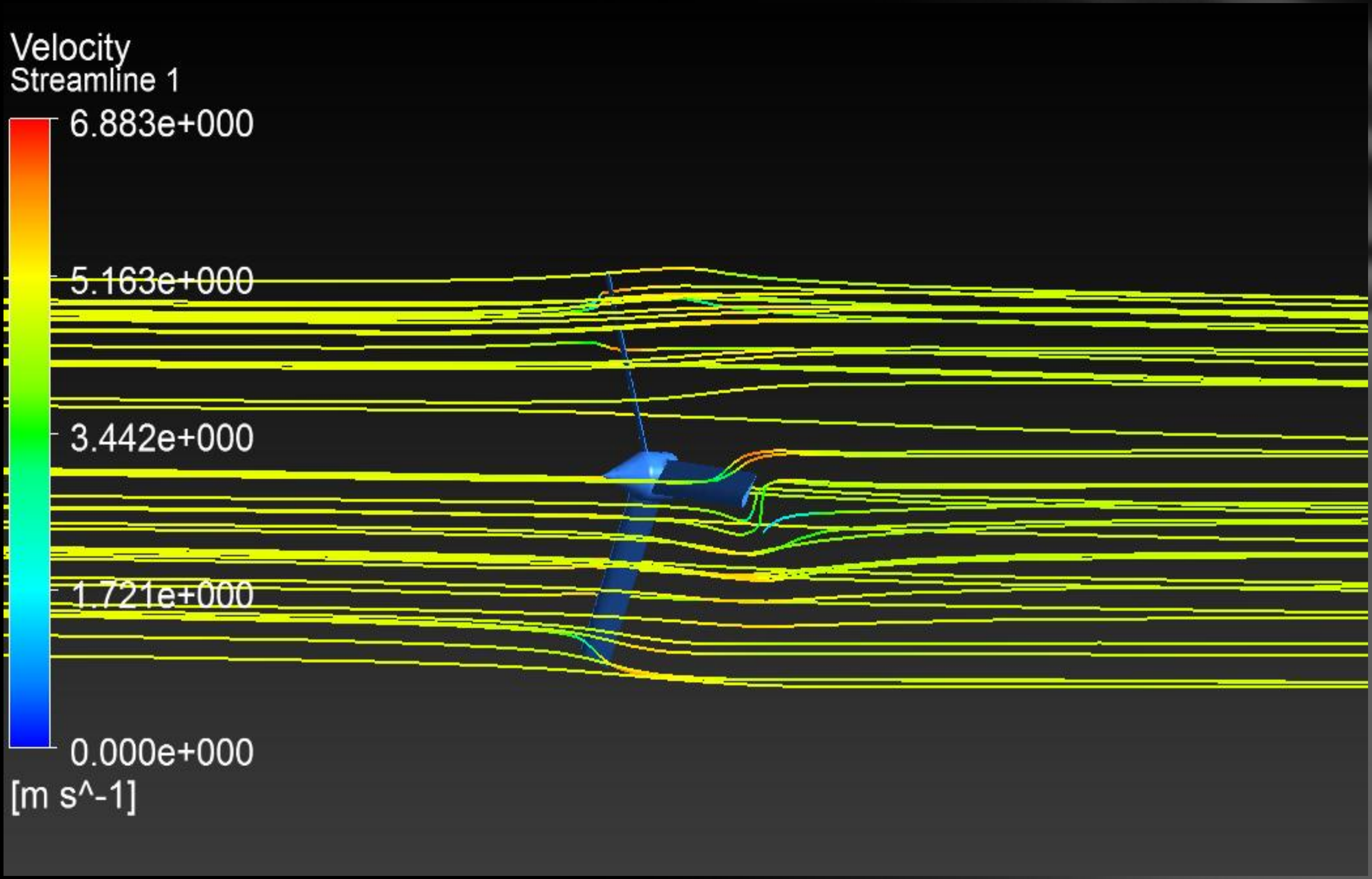



```
#include "udf.h"
static real Ux_prev = 0.0;

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
}
```

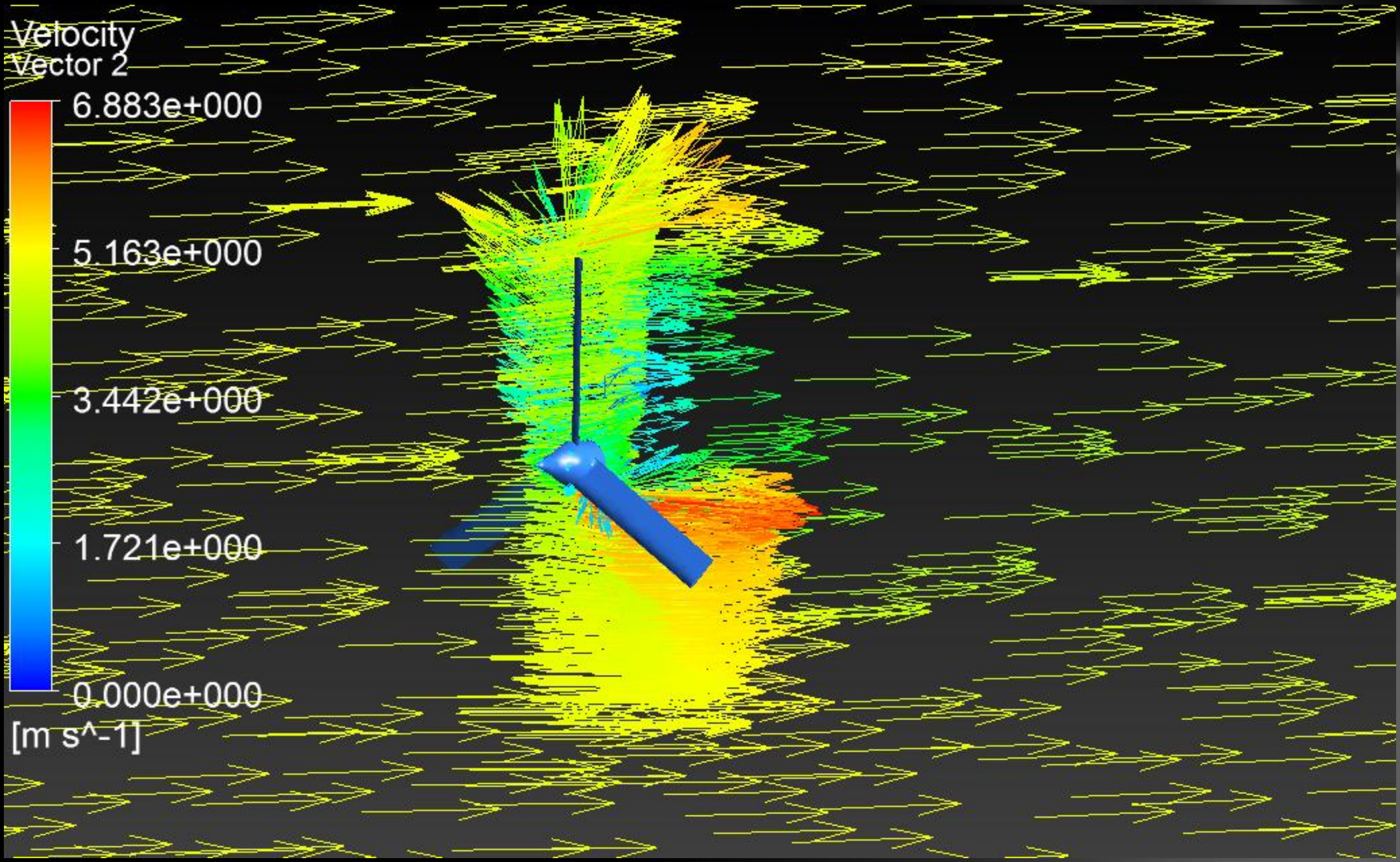
Case I – Wind Turbine Only (Converging-Diverging Nozzle removed)



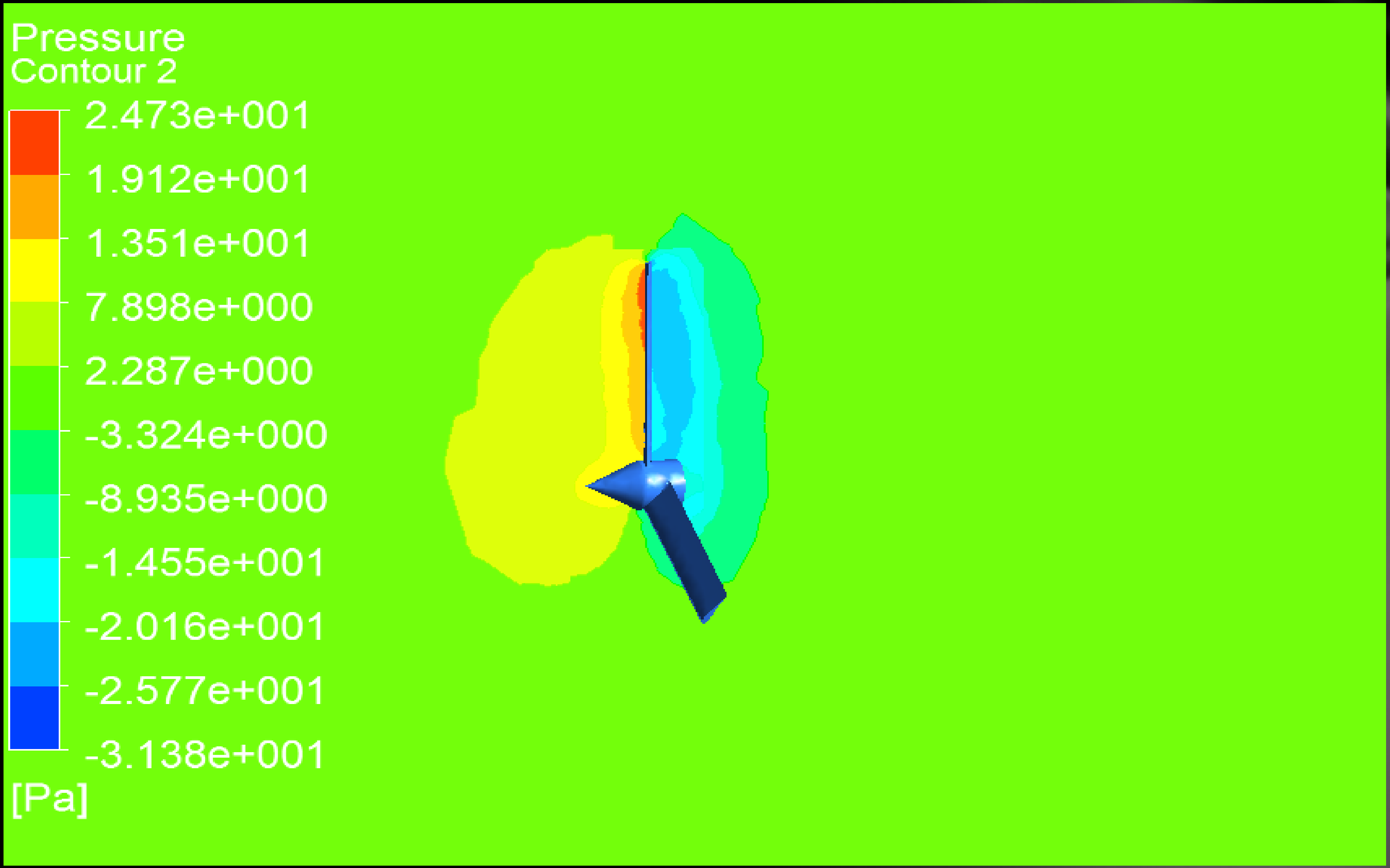
<u>Case I – Wind Turbine Only (Converging-Diverging Nozzle removed)</u>



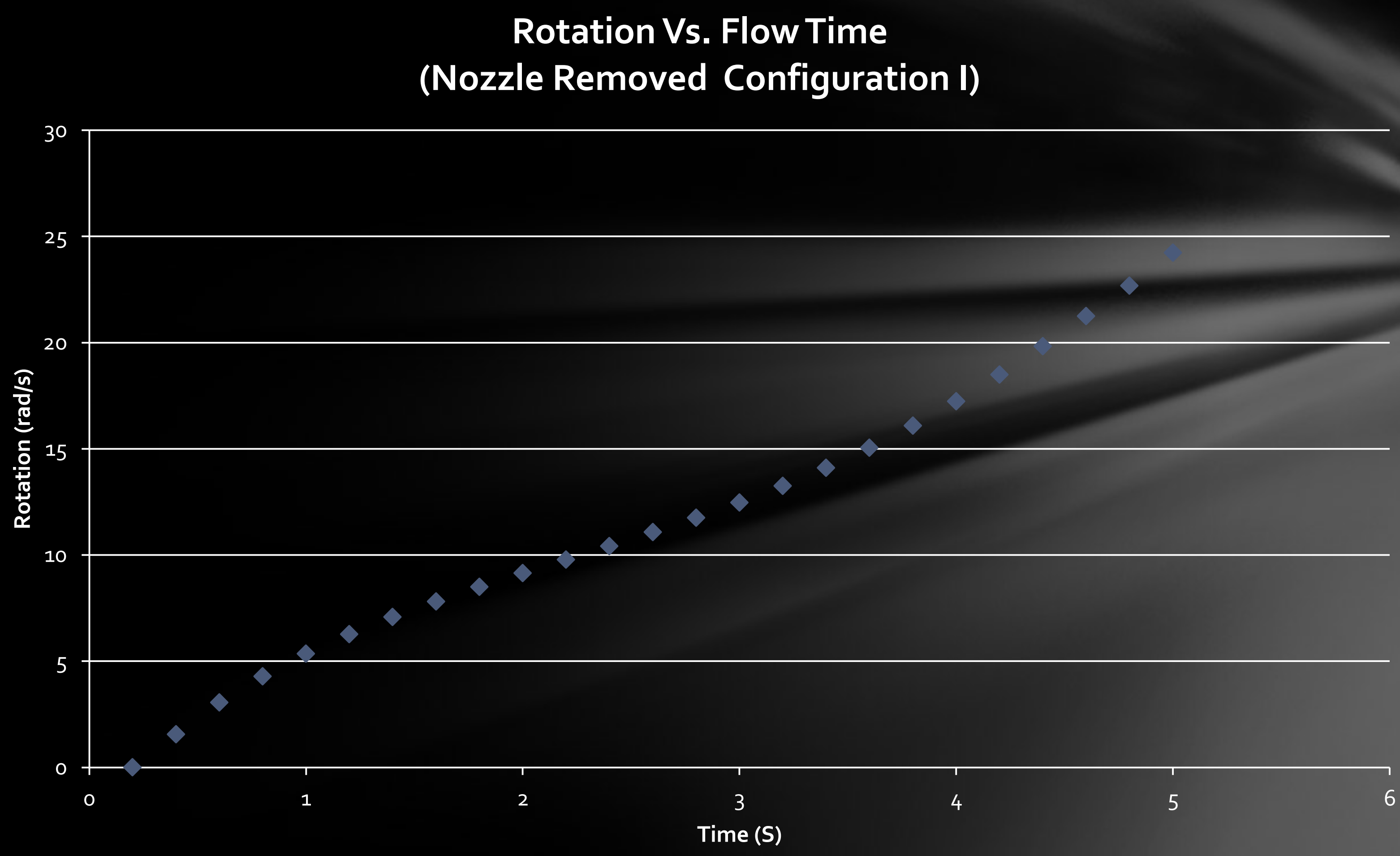
Case I – Wind Turbine Only (Converging-Diverging Nozzle removed)



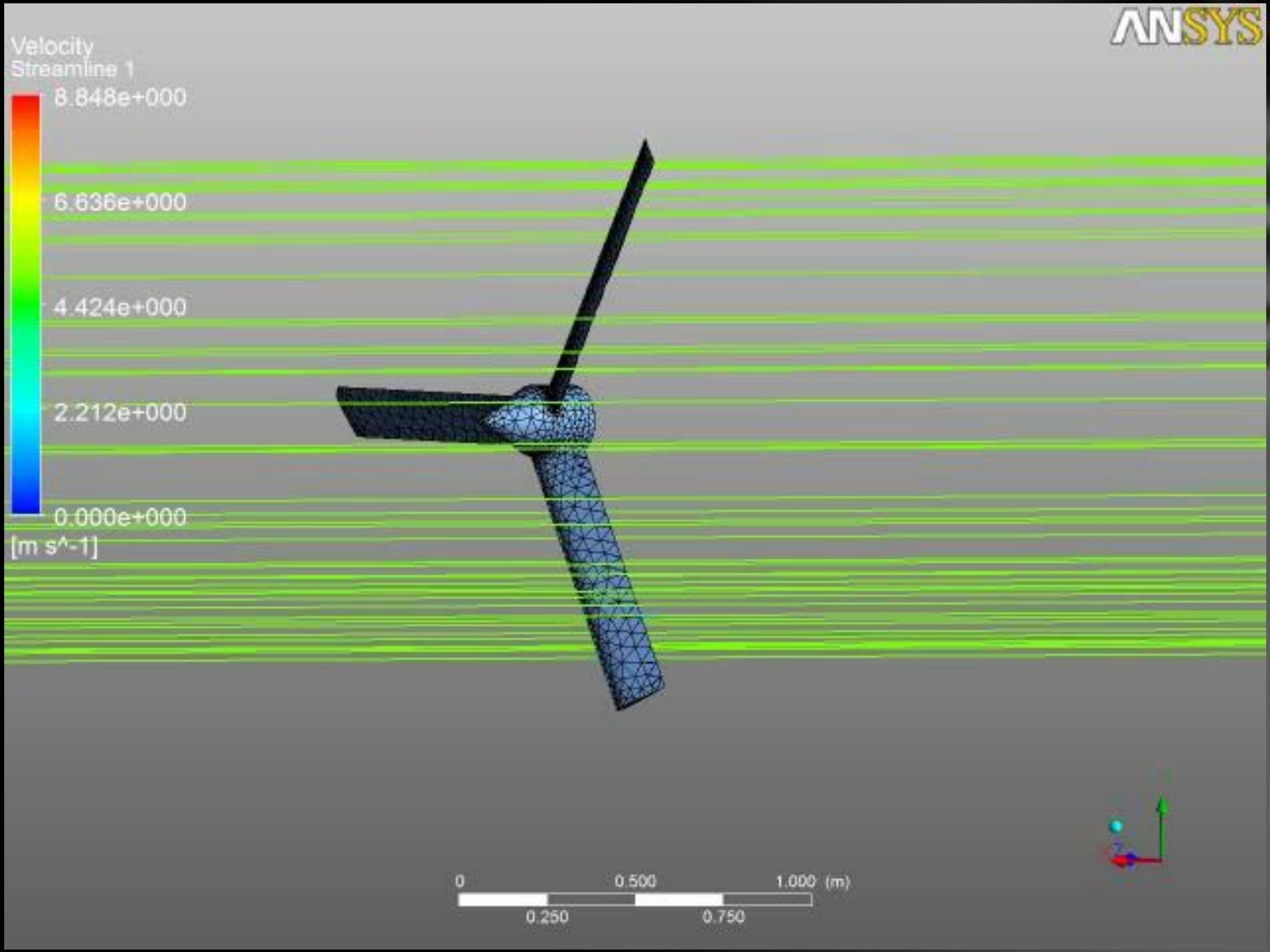
Case I – Wind Turbine Only (Converging-Diverging Nozzle removed)



Case I – Nozzle Removed



Demonstration of Turbine Rotation



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

<u>Power Output</u>: 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 (1 0 0)

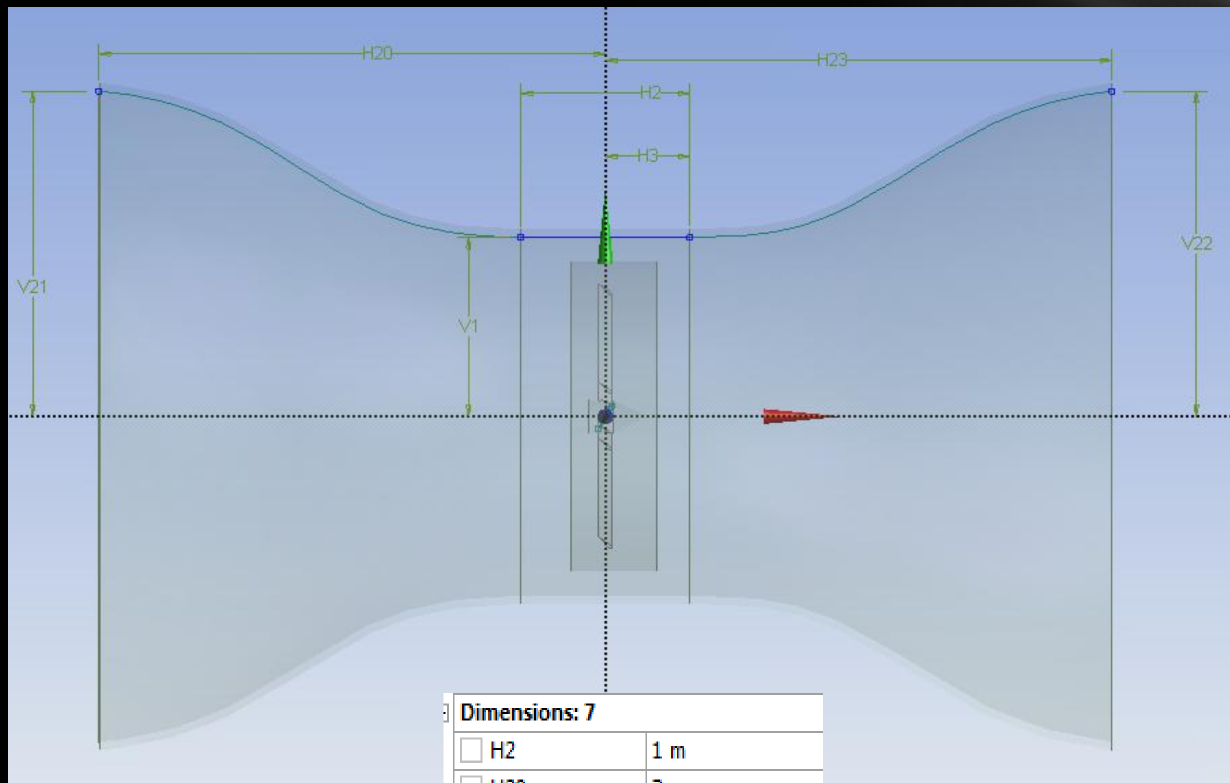
Zone	Pressure	Viscous	Total
turbine	-0.22938613	0.00020540987	-0.2291807

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

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

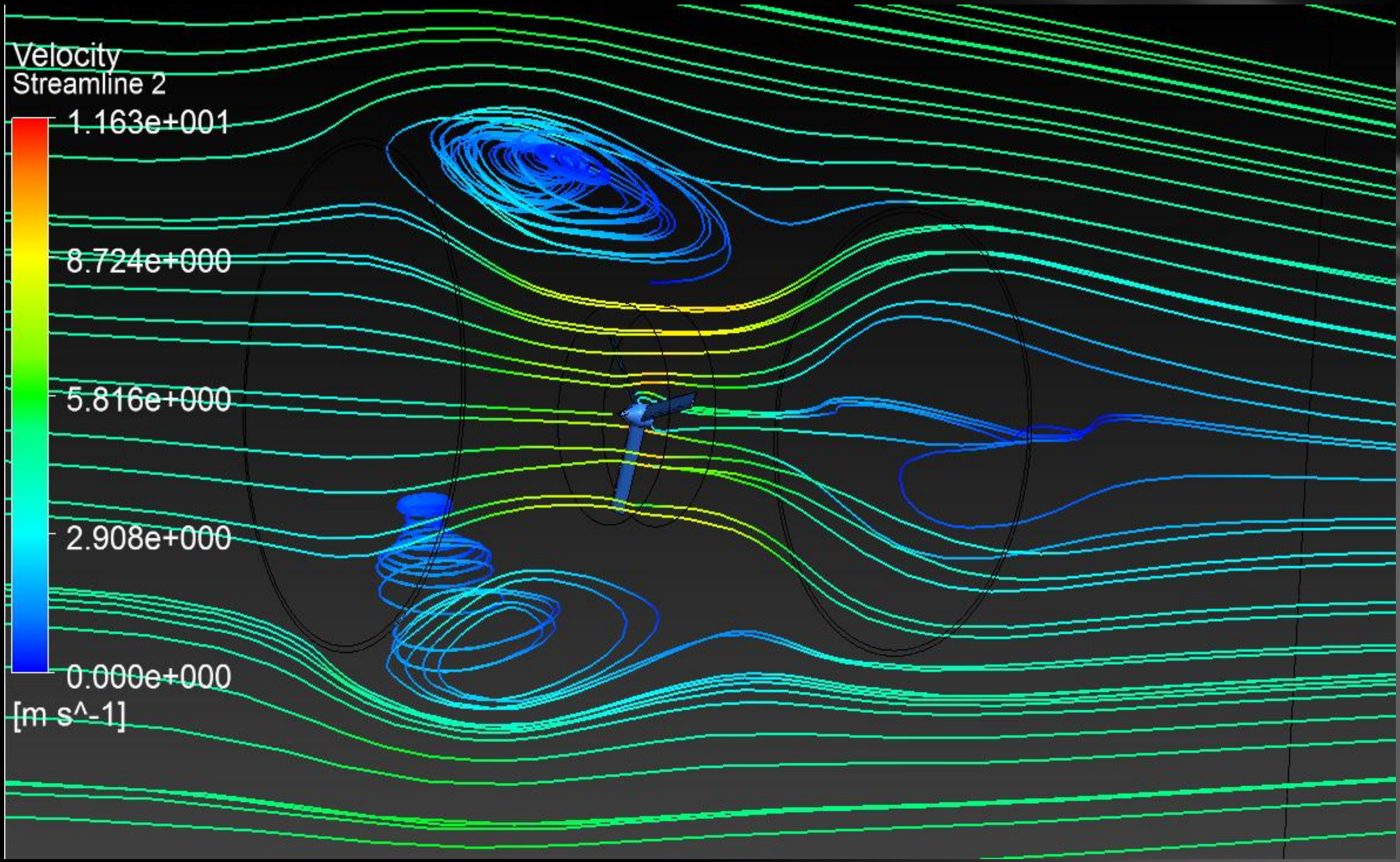
$$Power\ Captured = \frac{P_{generated}}{P_{avaliable}} x 100\% = 2.85\%$$

Case II – Initial Design

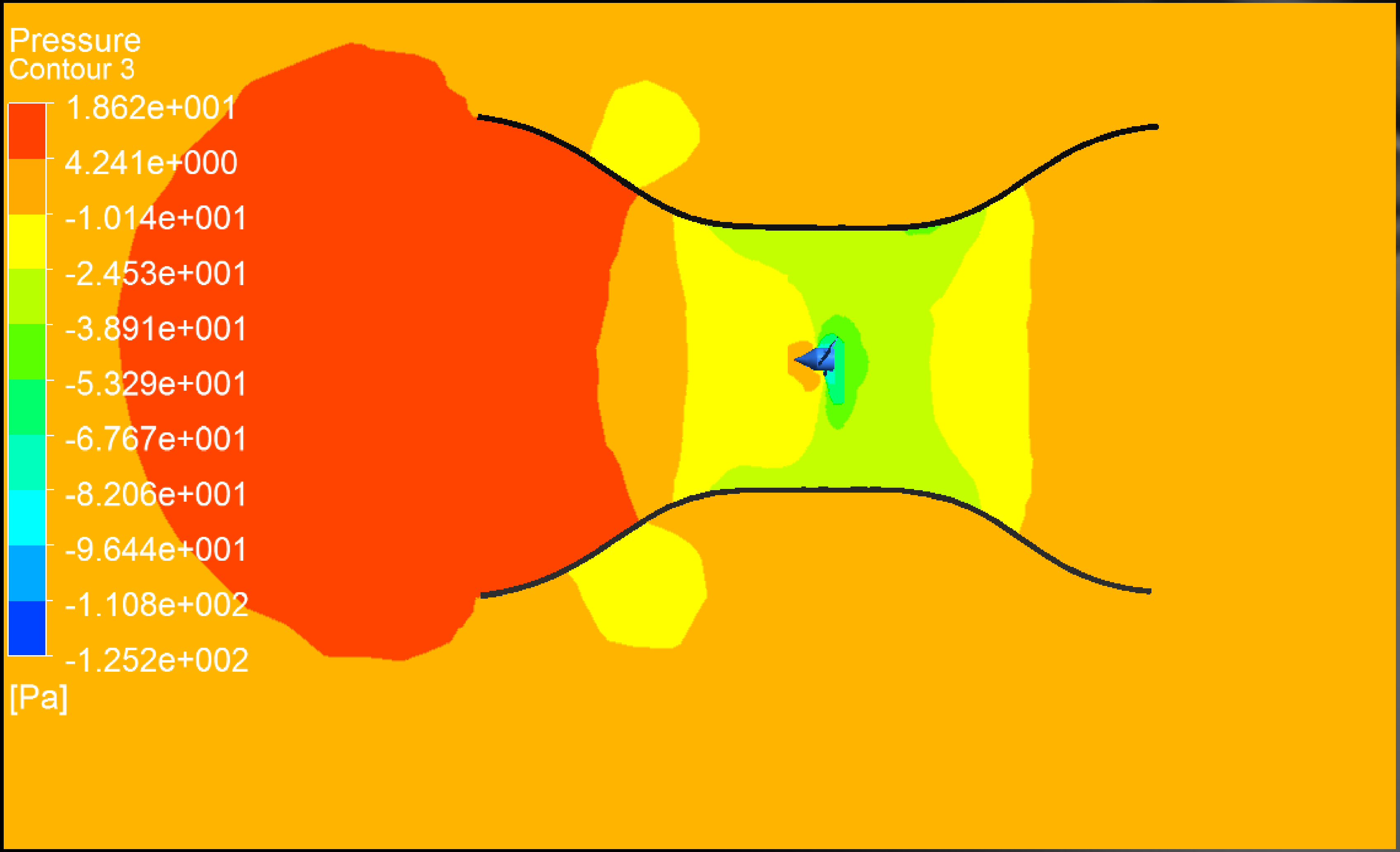


Dimensions: 7	
H2	1 m
H20	3 m
H23	3 m
H3	0.5 m
V1	1.1 m
V21	2 m
V22	2 m

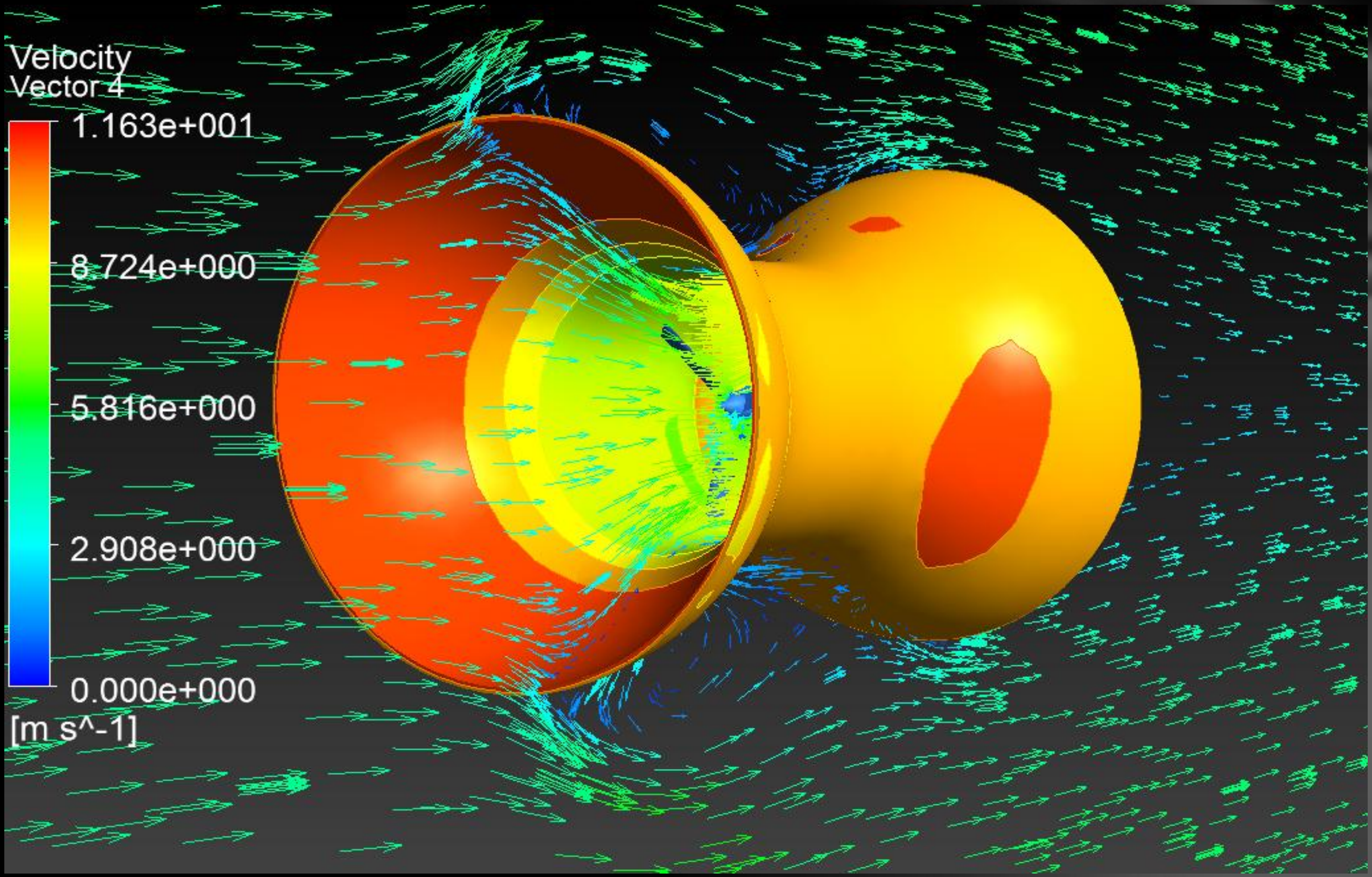
Case II – Initial Design



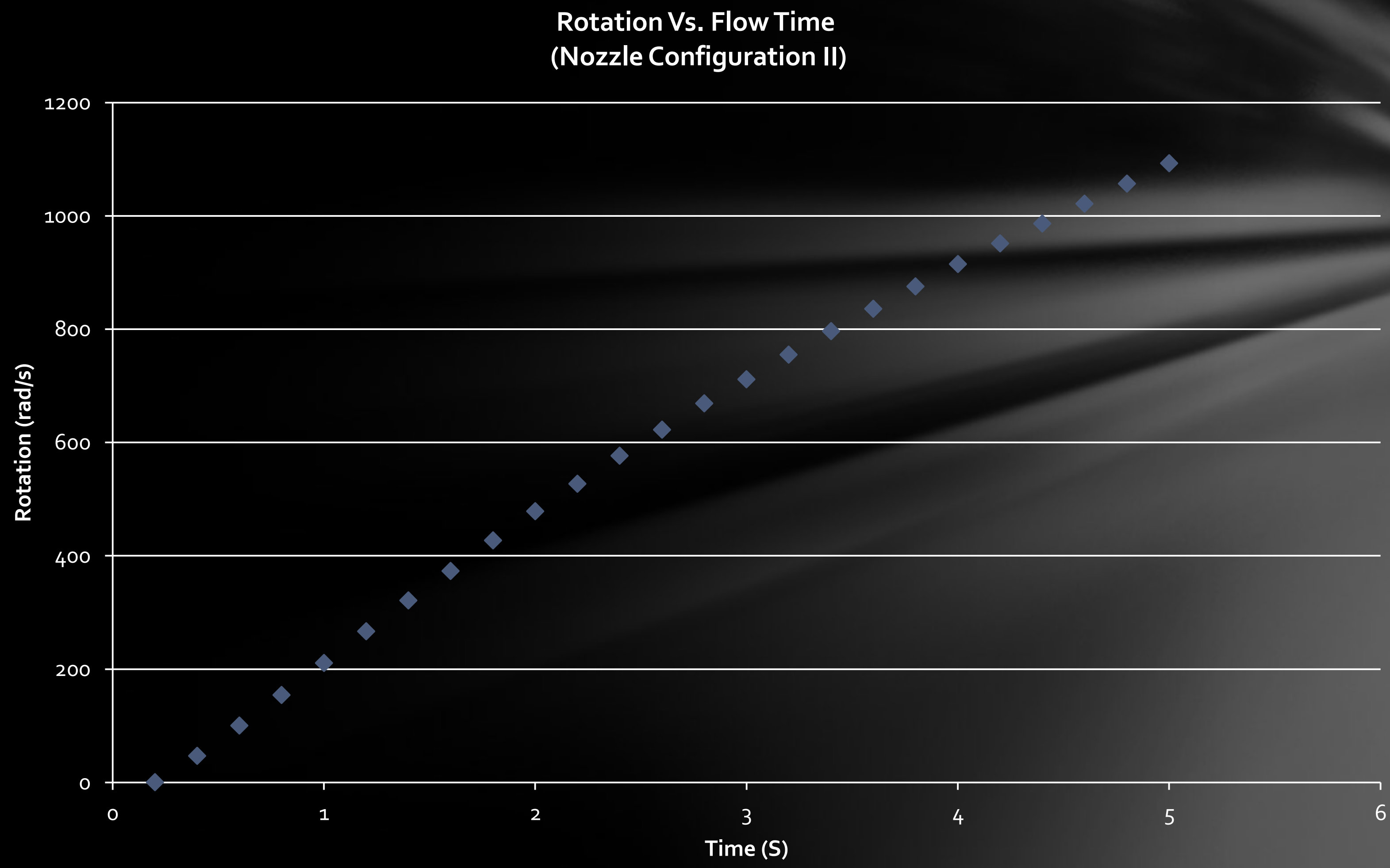
Case II – Initial Design



Case II – Initial Design



Case II – Initial Design



Case II – Initial Design

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 (1 0 0)

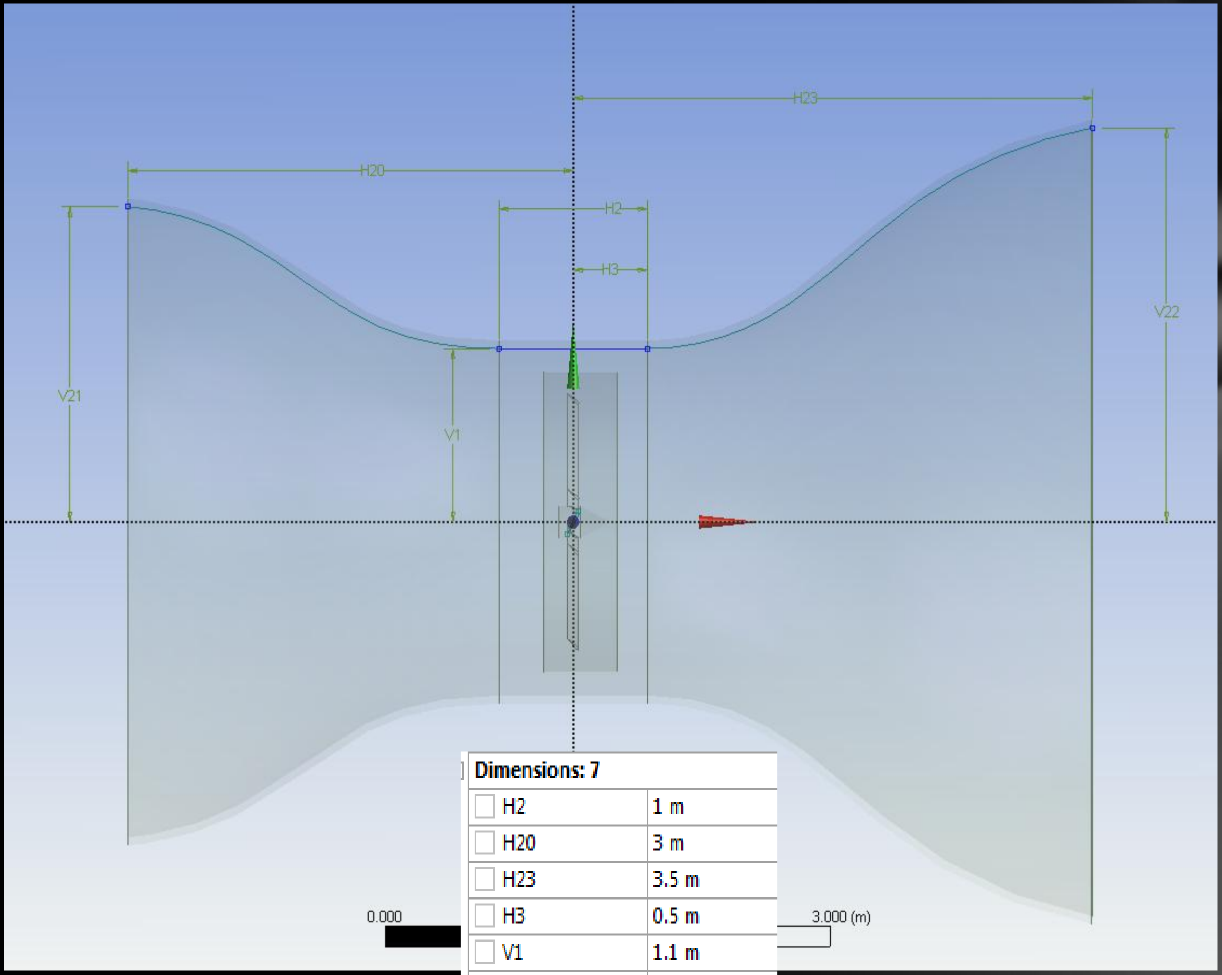
Zone	Pressure	Viscous	Total
turbine	-0.51939837	0.00021137946	-0.51918699

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

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

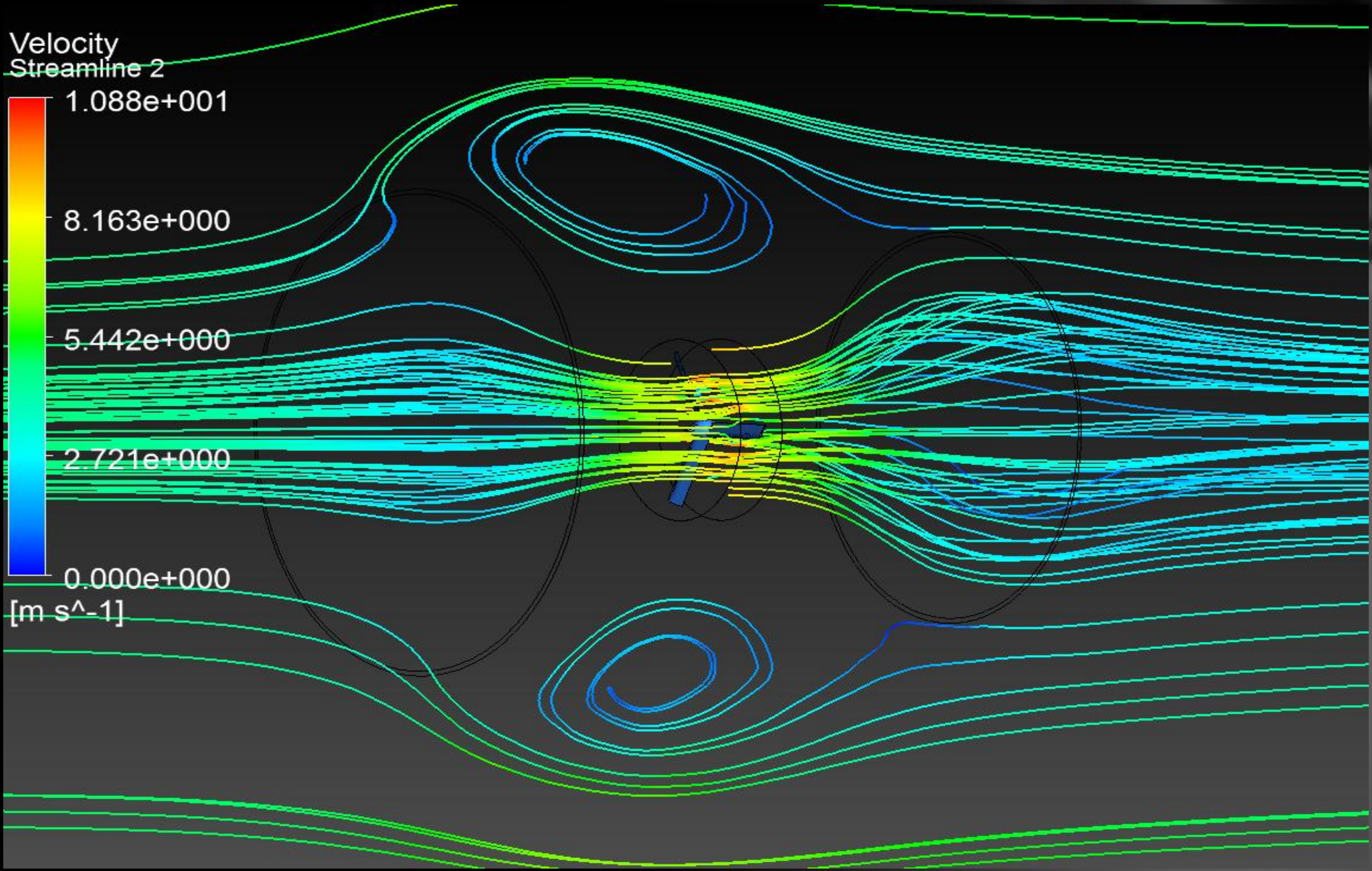
$$Power\ Captured = \frac{P_{generated}}{P_{avaliable}} x 100\% = 26\%$$

Case III –Increased in Inlet diameter and length of the converging Nozzle

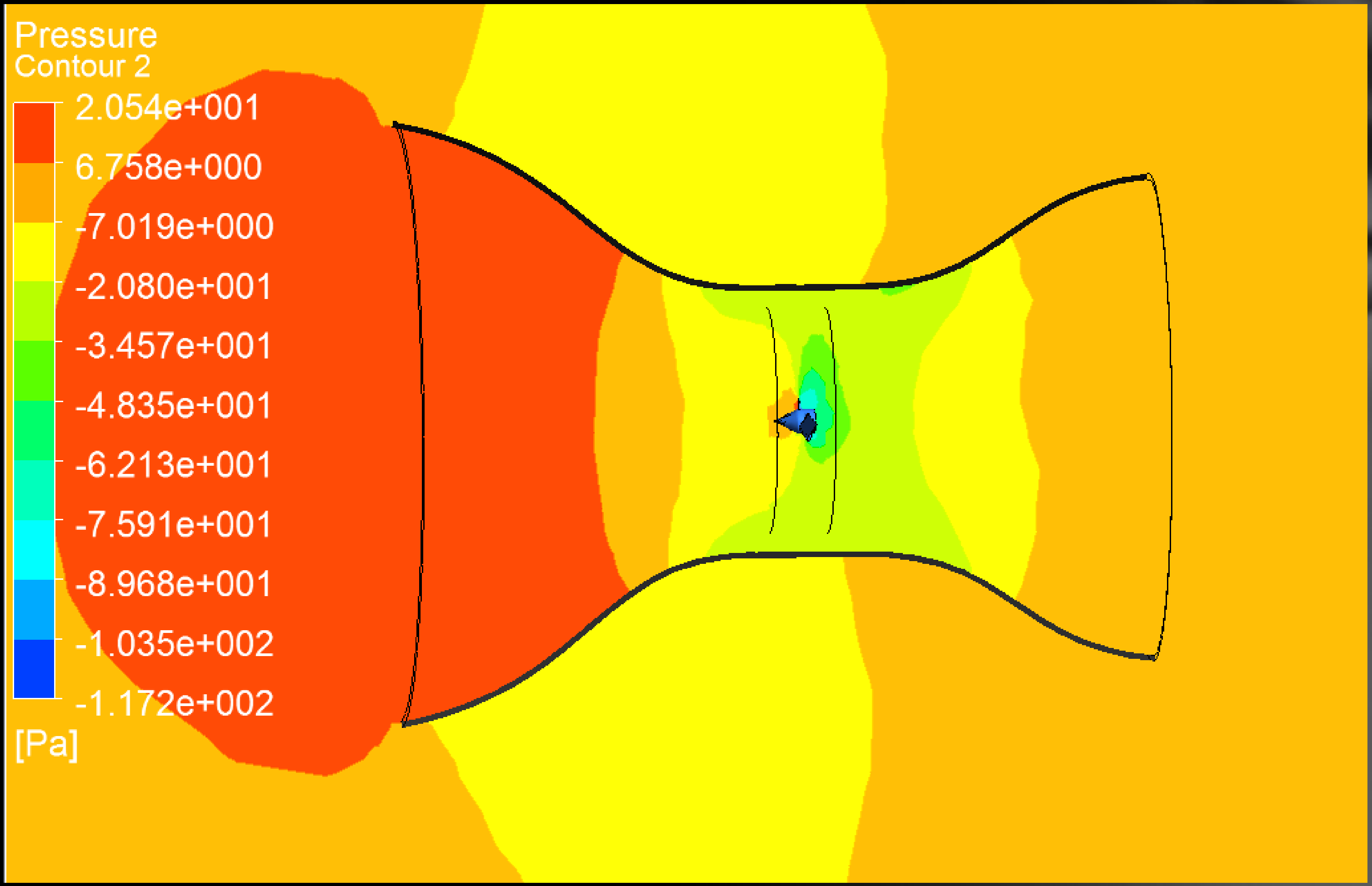


Dimensions: 7	
H2	1 m
H20	3 m
H23	3.5 m
H3	0.5 m
V1	1.1 m
V21	2 m
V22	2.5 m

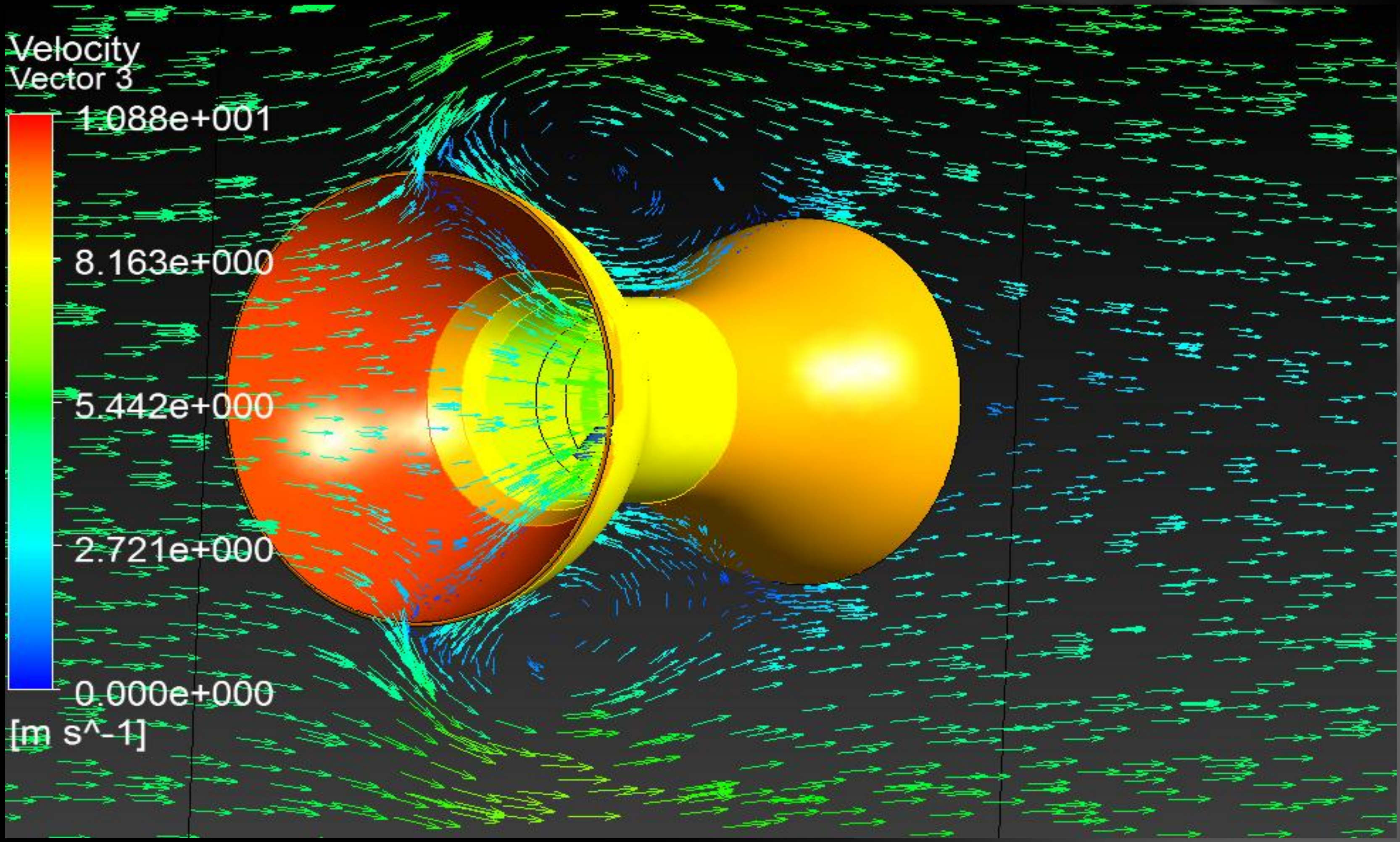
Case III –Increased in Inlet diameter and length of the converging Nozzle



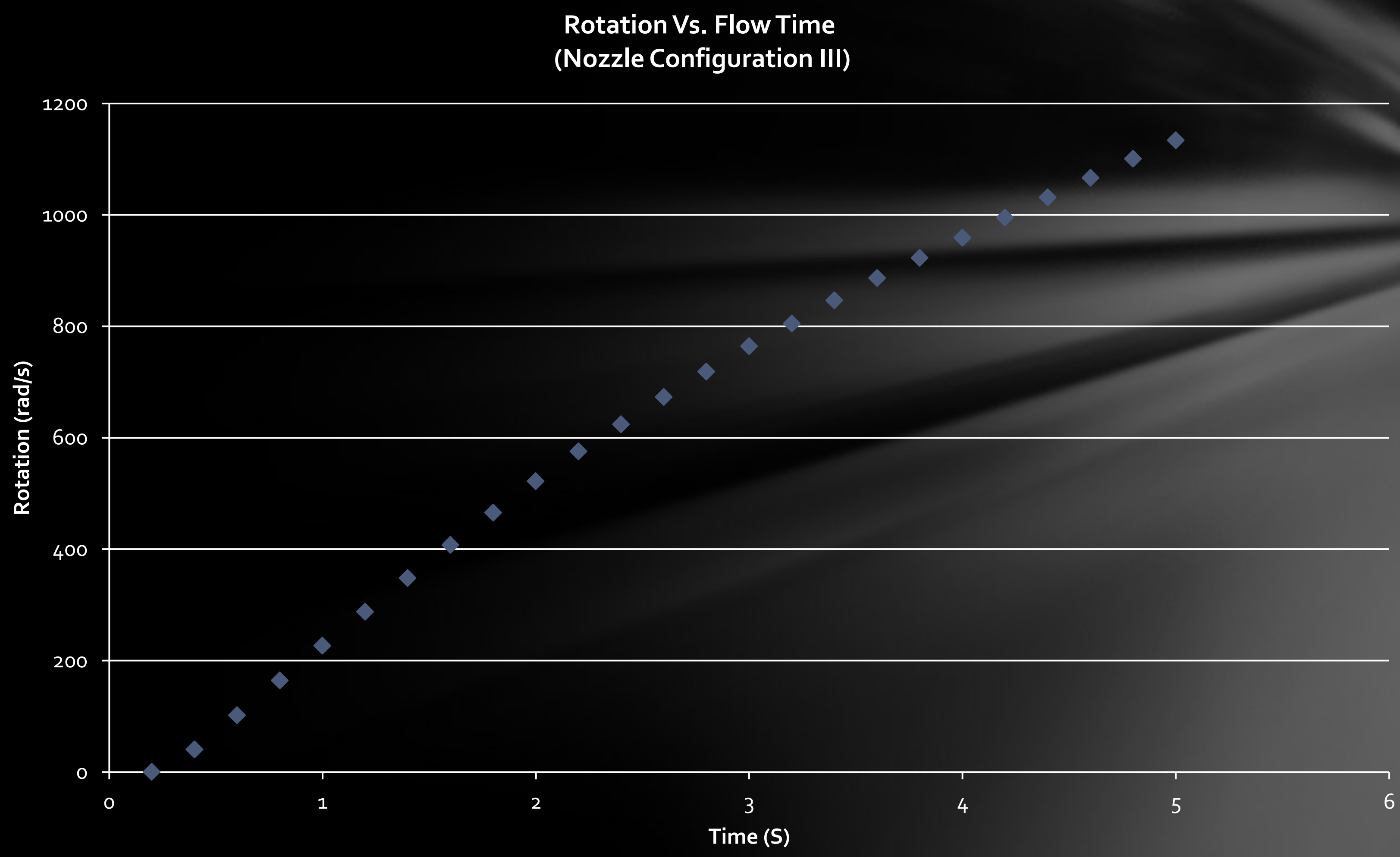
Case III –Increased in Inlet diameter and length of the converging Nozzle



Case III –Increased in Inlet diameter and length of the converging Nozzle



Case III –Increased in Inlet diameter and length of the converging Nozzle



Case III –Increased in Inlet diameter and length of the converging Nozzle

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 (1 0 0)

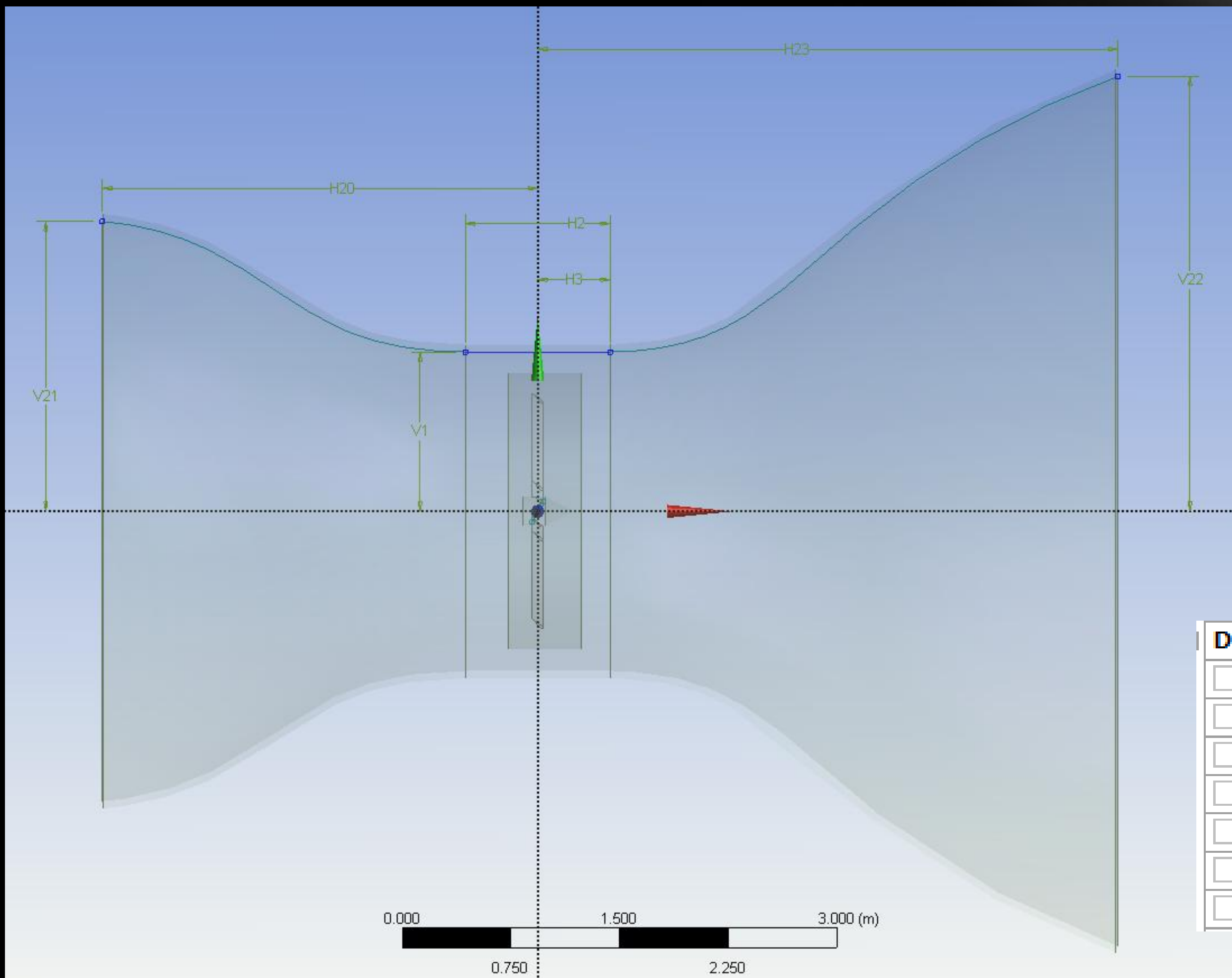
Zone	Pressure	Viscous	Total
turbine	-0.39420759	-9.7518579e-05	-0.39430511

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

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

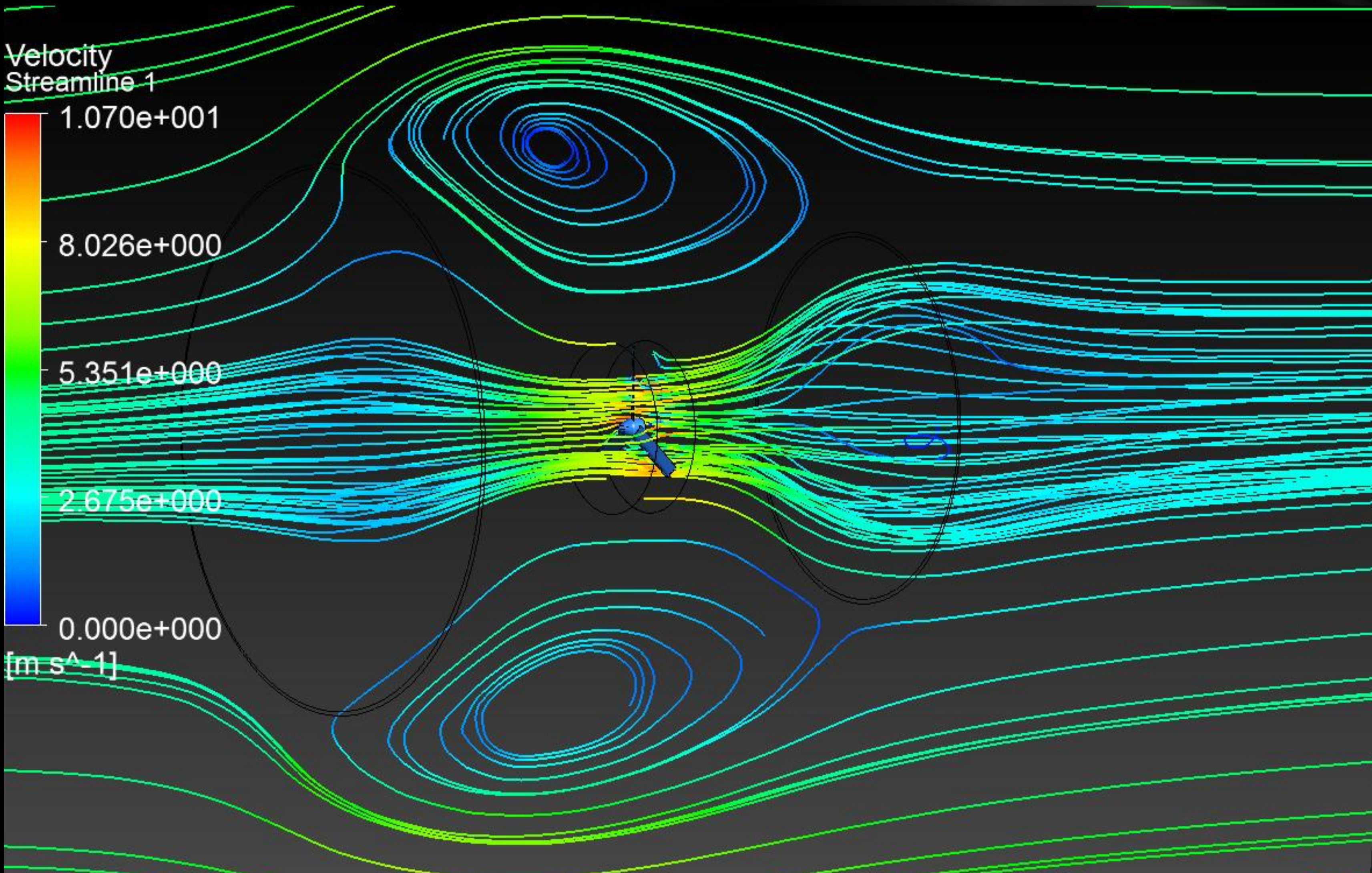
$$Power\ Captured = \frac{P_{generated}}{P_{avaliable}} x100\% = 52.8\%$$

Case IV –Further Increased in Inlet diameter and length of the converging Nozzle

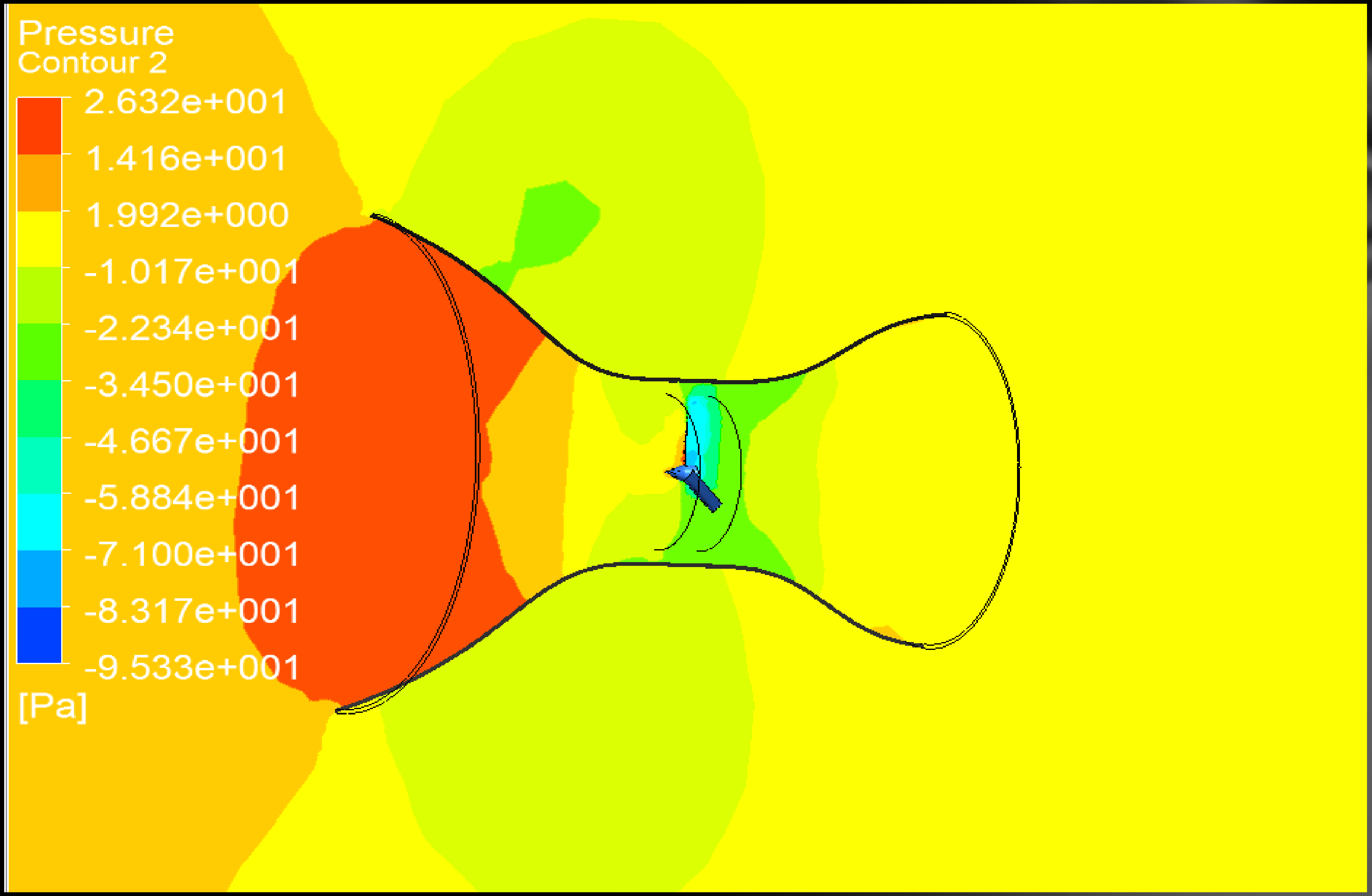


Dimensions: 7	
H2	1 m
H20	3 m
H23	4 m
H3	0.5 m
V1	1.1 m
V21	2 m
V22	3 m

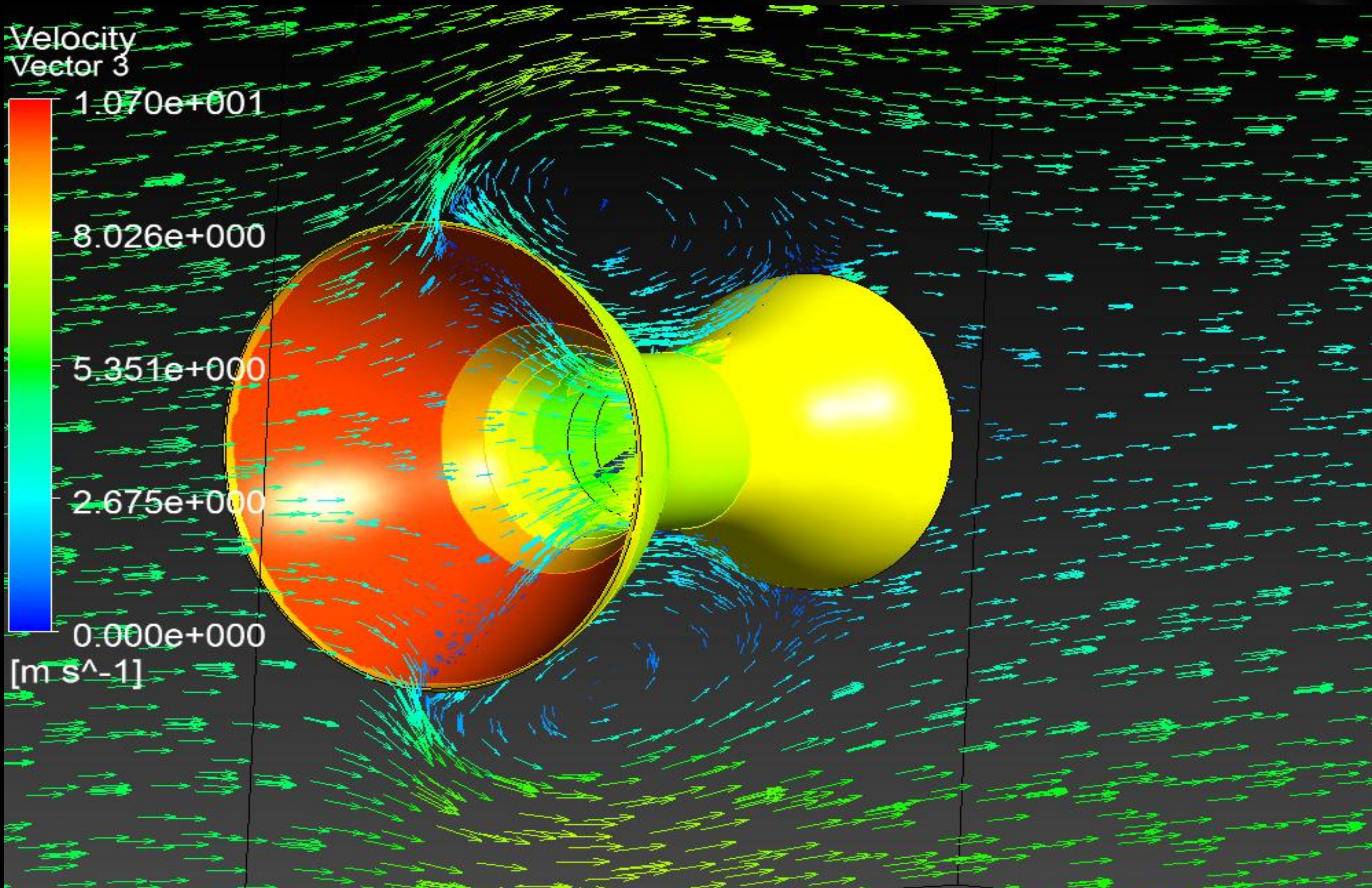
Case IV –Further Increased in Inlet diameter and length of the converging Nozzle



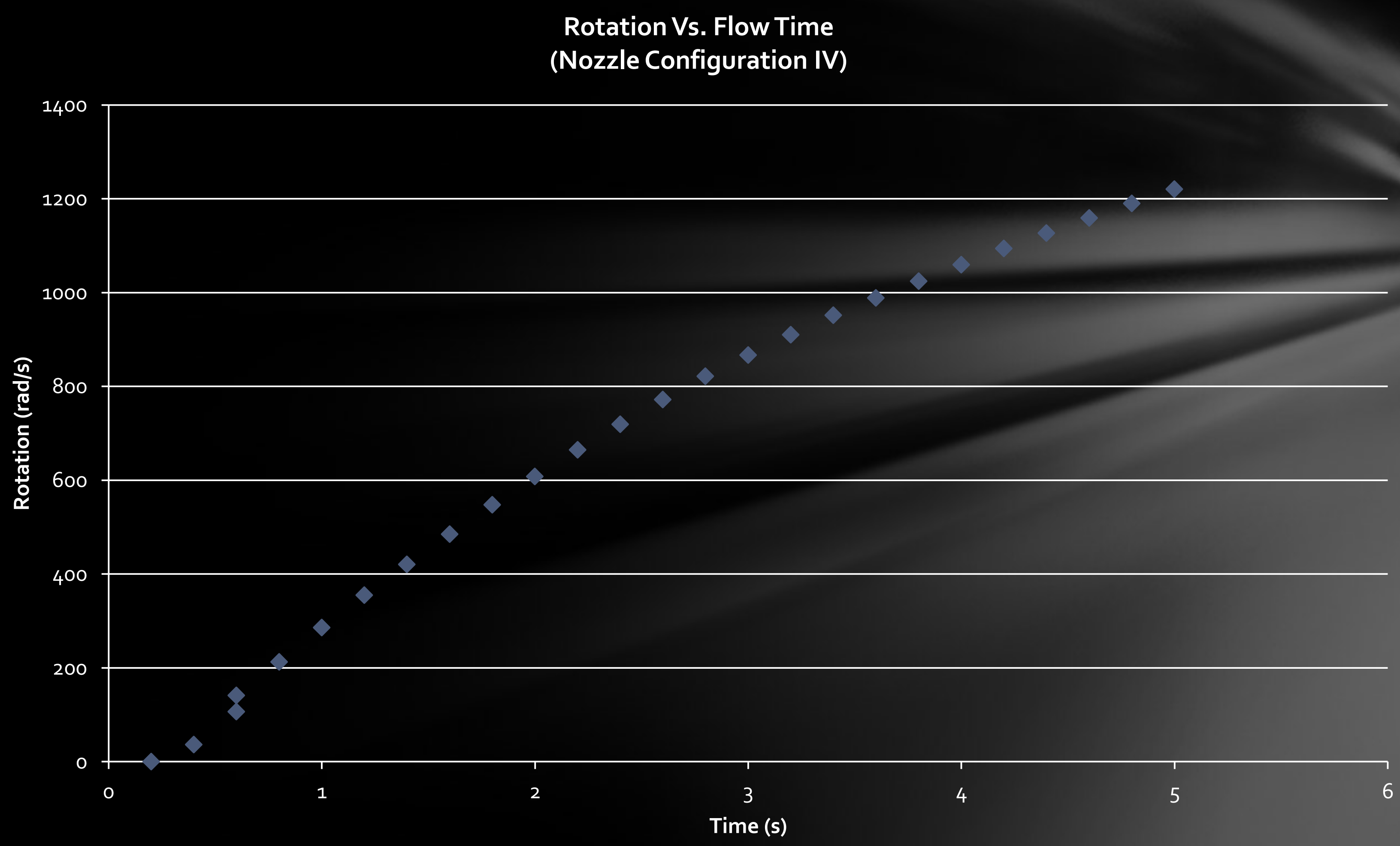
Case IV –Further Increased in Inlet diameter and length of the converging Nozzle



Case IV –Further Increased in Inlet diameter and length of the converging Nozzle



Case IV –Further Increased in Inlet diameter and length of the converging Nozzle



Case IV –Further Increased in Inlet diameter and length of the converging Nozzle

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 (1 0 0)

Zone	Pressure	Viscous	Total
turbine	-0.49006197	0.00020831742	-0.48985365

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

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

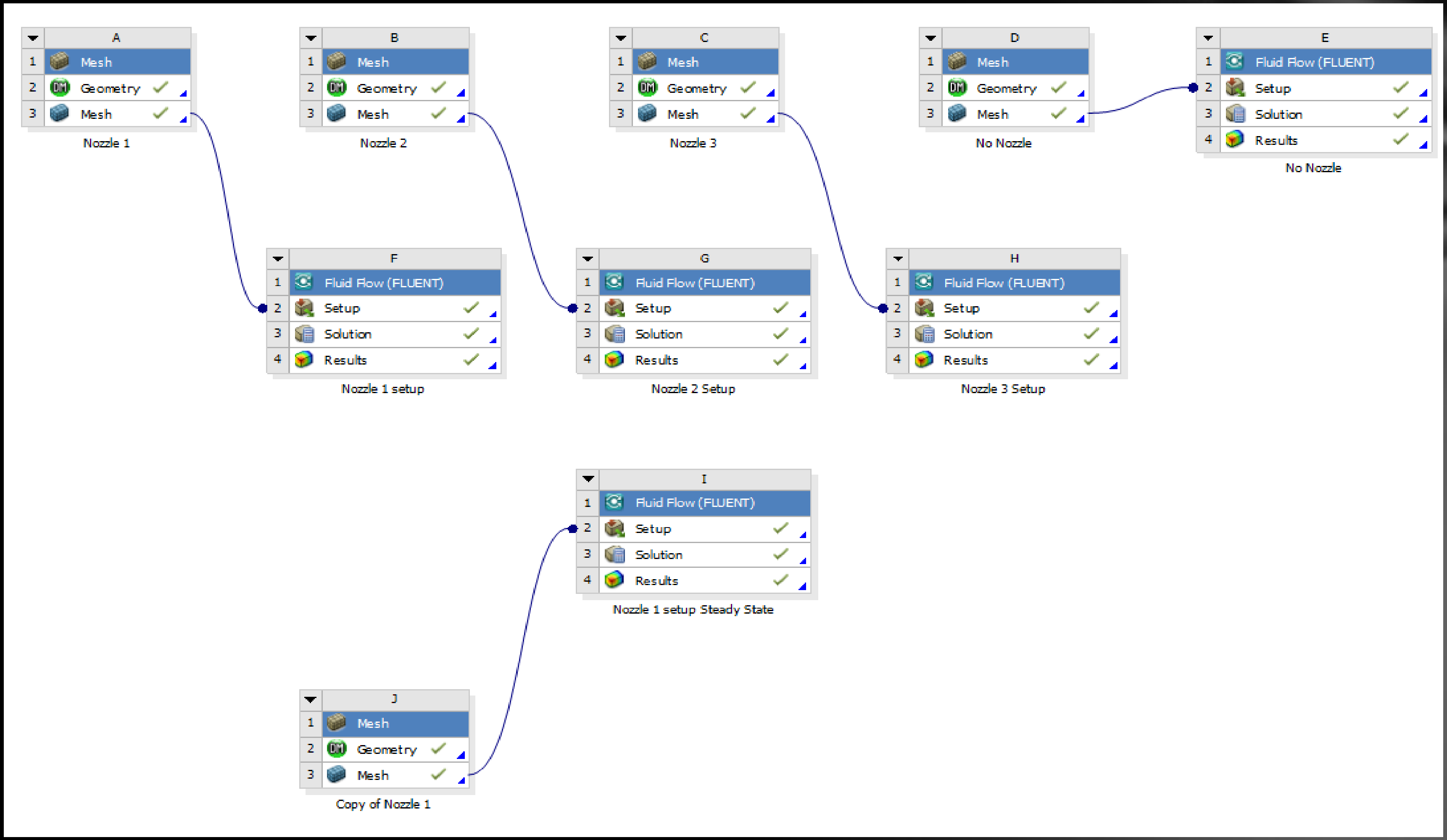
$$Power\ Captured = \frac{P_{generated}}{P_{avaliable}}x100\% = 76.80\%$$

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.

Setup	Power Available	Power Generated	Power 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.

References

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