# Vertical Axis Wind Turbine (Part 1) - Numerical Results

Author: Julio Sampaio Gabriel de Pieri, Cornell University

```
Problem Specification

1. Pre-Analysis & Start-Up

2. Geometry

3. Mesh

4. Physics Setup

5. Numerical Solution

6. Numerical Results

7. Verification & Validation

Exercises

Comments
```

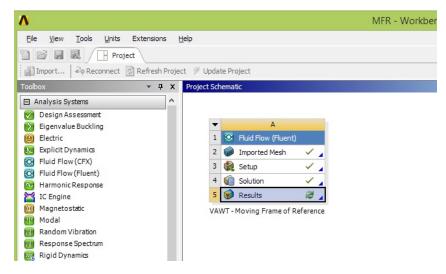
# Numerical Results

We can use either FLUENT or CFD-Post as post-processing tool. CFD-Post is preferable as it is more user friendly and gives you more freedom.

### **Velocity Contours**

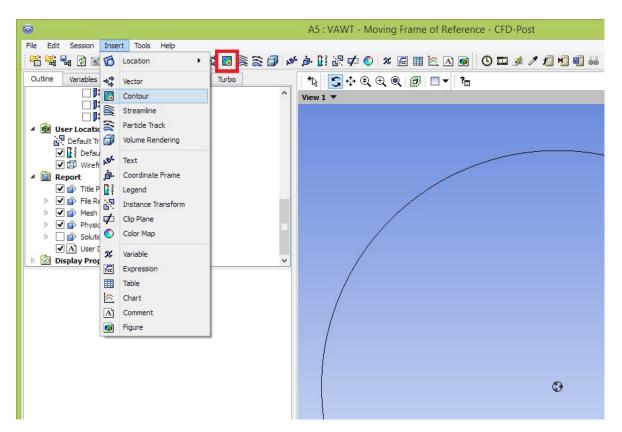
To plot the velocity contours we will use CFD-Post. You can now close FLUENT.

In Workbench, under Project Schematic, double click Results. This will launch CFD-Post



CFD-Post usually opens with an isometric view of the part. Since we're in a 2D analysis, this is not very useful for us. So, the first thing to do is click on the Z axis arrow (bottom-right corner of graphics window) to change the view.

Now insert a Contour. Click on the Contour icon (or Insert > Contour)



Name it "Velocity contour". A new box will appear on the left side of the screen. Summary of what to do:

#### Domains: leave default ("All Domains")

Locations: click on the three dots "..." on the side. select all names with "symmetry 1" in it. Hold the Ctrl key for that. You will select 5 zones in total, see figure.

	Title Page				E Hode_bot E Hode_b E Hode_b E Hode_b E Hode_b	Prame of Refe of symmetry 1 of symmetry 2 of jn	×
letaris of Veloc	Properties and Defaults Ry contour Lobels Renter View		v		i≇ biede_n I≢ biede_n		_
Domains Locations	Al Coneire blade_bot symmetry 1		-	- 1		de_right op symmetry 1 op symmetry 2	
Karlable Range	Pressure	•			<ul> <li>biede_to</li> <li>val_bie</li> <li>fuid surface</li> </ul>	søjn dejtep jacdy	
Mer Mex		utinesn utinesn				l face_body synn	
of Cantours 11. Advanced Properties					F fuid aut F hub_aut		
					E binds ja E binds ja E binds ja E binds ja E binds ja E binds ja E binds ja	ght_water sp_owter er mnetry 1	_
					DK.	Carrod	
				_	-		

Variable: change is to "Velocity".

Range: leave as "Global".

# of Contours: change to around 51.

Click Apply.

You can zoom into the hub, drawing a box with the right mouse button.

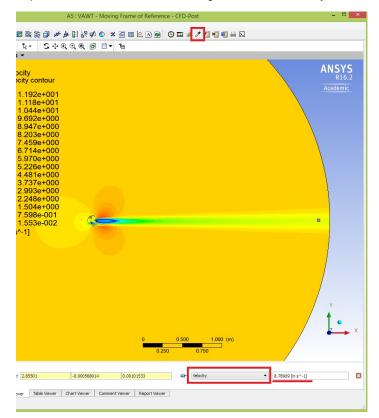
	A5 : VAWT - Moving Frame of Reference - CFD-Post	- 0
Edit Session Insert Tools Help	ミ 河 か 身 野 沢 中 ◎ ≈ ◎ 目 ≥ △ ● ◎ □ ≠ / □ 1 ● ■ ⇒ Σ	
e Variables Expressions Calculators Turbo	N S 🕂 Q Q Q @ □ - 1a	
□ <b>]</b> ‡ inner symmetry 1	^ View1 ▼	
inner symmetry 2		ANCV
User Locations and Plots	Velocity	ANSY
V R Velocity contour	Velocity contour	
Default Transform	Velocity contour	
Default Legend View 1	1,192e+001	
V 🗇 Wireframe		
Report	1.118e+001	
🕑 🍙 Title Page	1.044e+001	
Fle Report	9.692e+000	
Mesh Report     Physics Report	8,947e+000	
Solution Report	8.203e+000	
Solution Report		
—	✓ 7.459e+000	
of Velocity contour	6.714e+000	
netry Labels Render View	5.970e+000	
ns All Domains	5.226e+000	
blade_bot symmetry 1,blade_right symmetry 1,blac	4.481e+000 3.737e+000	
able Velocity		
	2.248e+000	
ge Global ·		
0.015529 [m s^-		
12.4208 [m s^-	7.598e-001	
	■ 1.553e-002	
Contours 51	[m s^-1]	
anced Properties		
enceu Properoes		
		Υ
		1.
		•
	0 0.150	0.300 (m)
	0.075 0.225	
ply Reset I	faults	
//eset	3D Viewer Table Viewer Chart Viewer Comment Viewer Report Viewer	

Note that it's very clear the effect of when the blade is perpendicular to the flow: a huge recirculation bubble is made. This will negatively affect other turbines placed downstream of this one. This is a very important thing to consider when designing an array of VAWTs.

This is a single snapshot of the spinning of the turbine, or a particular position. A transient analysis with a complete animation will be created in the future.

You can also zoom out and see that far downstream of the turbine, the flow has not yet recover its freestream condition. If you probe the velocity inside that slightly brighter area you will see that the velocity is around 9m/s (instead of 10m/s of the freestream).

To probe: Click on "Probe" icon. Then change the variable to "Velocity" and click on the screen where you wanna probe.



## Vorticity Contours

Another interesting thing to analyze is the Vorticity distribution downstream of the turbine. This strongly affects how you would distribute more turbines in case you are designing an array of them.

But to do that we have to tell Fluent to export Vorticity Magnitude to CFD-Post. To do that, go back to Workbench, and double click "Solution".

Under "Run Calculation", click on "Data File Quantities". Select "Vorticity Magnitude" from the list and click Ok.

	Run Calculation	1. PICSI1	•
	Check Case Preview M	esh Motion	
ns	Number of Iterations 600 Profile Update Interval	nterval	
	Data File Quantities Acousti	c Signals	
on es	Calculate		
	Di	ata File Quantities	×
omization	applications, select them below. Standard Quantities Pressure X Velocity Y Velocity Mass Flux Body Force Wall Velocity Original Wall Velocity Wall Shear Mach Number Boundary Heat Flux Boundary Rad Heat Flux Turbulent Kinetic Energy Turbulent Dissipation Rate Wall Yplus Wall Yplus Utau	Additional Quantities Additional Quantities Relative Y Velocity Relative Tangential Velocity Mesh X-Velocity Wesh Y-Velocity Velocity Angle Relative Velocity Angle Cell Reynolds Number Turbulent Kinetic Energy (k) Turbulent Intensity Turbulent Dissipation Rate (E Production of k Turbulent Viscosity	
	DPM-partition Density Laminar Viscosity Turbulent Viscosity	Effective Viscosity Turbulent Viscosity Ratio Wall Ystar	not cr
	DPM-partition Density Laminar Viscosity	Turbulent Viscosity Ratio Wall Ystar Wall Yplus Molecular Viscosity	not ci
	DPM-partition Density Laminar Viscosity Turbulent Viscosity 2nd Grad Bc Source	Turbulent Viscosity Ratio Wall Ystar Wall Yplus	
	DPM-partition Density Laminar Viscosity Turbulent Viscosity 2nd Grad Bc Source	Turbulent Viscosity Ratio Wall Ystar Wall Yplus Molecular Viscosity Wall Shear Stress	v not c

Run the simulation again so Fluent can retrieve the desired value. It will converge in 2 iterations. Close Fluent and Refresh the project.

Open CFD-Post and create a new contour. Name it "Vorticity Contour".

For "Locations", do the same procedure as for the velocity contours (click on "..." and select all "[...] symmetry 1" zones).

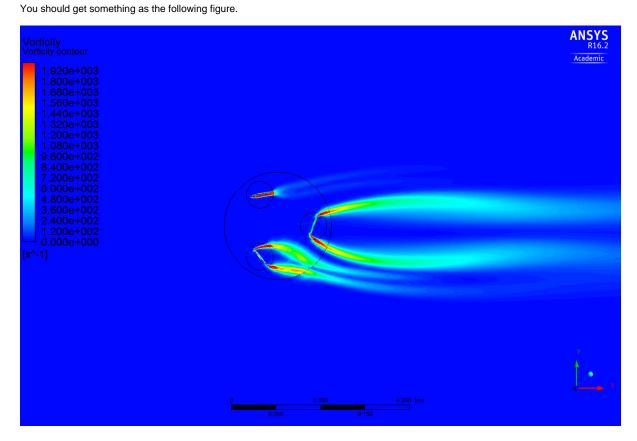
For "Variable", select "Vorticity".

For "Range", change it to "User Specified", and change the value for Min to 0, and for Max to 2000 [s^-1].

Change the number of contours to 101.

ometry	city contour	View	
ains	All Domains	The P	•
tions	blade_bot symmet	try 1,blade_right symmetry 1,blade_top symmetry 1,fluid surface_body sy	•
iable	Vorticity 👻		•
ange	User Specified		•
ı	0 [s^-1]		
c.	2000 [s^-1]		
Contours	101		-
vanced Pro	perties		Đ

#### . . . . . . . . . . .



You can see the vorticity emanting from the tips of the blades. If you ever seen a video from a flow over a plate perpendicular to the direction of the flow, you might remember the vorticies being generated at the tips of the plate.

Also note that for the blade that is almost aligned with the flow, the vorticity does not propagate much (is less intense).

One could also plot some cool pictures for pressure (you will notice some relationship with the velocity contours) and for Turbulent Kinetic Energy (you will notice some relationship with the vorticity contours). For the latter one, change the scale from Min=0 to Min=7 J/kg and enjoy a very beautiful turbulent kinetic energy distribution! Note how close the contour will be to the Vorticity one.

Go to Step 7: Verification & Validation

Go to all FLUENT Learning Modules