Vertical Axis Wind Turbine (Part 1) - Verification & Validation

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
```

Verification & Validation

Check mass flow

It's always good to check the mass flow rate after CFD simulation. The solver tries to keep it satisfied, but sometimes a representative imbalance is obtain, showing that something has to be done in order to get more accurate results.

To do this, we will use FLUENT. Open FLUENT from Solution.

Highlight "Reports" in the left box. Then select "Fluxes" and click "Set Up ... ".

2	A:VAWT - Mo	Noving Frame of Reference Fluent@Julio-PC [2d, dp, pbns, rke] [ANSYS Academic Teaching Introduc	tory]				
<u>File Mesh Define Solve Adapt Surface D</u>	isplay <u>R</u> eport Para <u>l</u> lel <u>V</u> iew <u>H</u>	Help					
💼 📴 🔻 🚽 🕶 🞯 🎼 🛟 🔂 🕀 🌶	* 🔍 洗 唱 • 🗆 • 🕅 •	▼ ■ ▼ ● ○ ○ ○ ○ ○ ○ ○ ○ ○ ○ ○ ○ ○ ○ ○ ○ ○ ○					
E- Setup	Reports	1: Mesh v	1: Mesh v				
	Reports Fluxes Forces Projected Areas - Unavailable	E Flux Reports ×					
Hesh Interfaces	Volume Integrals	Options Roundaries E Results					
Oynamic Mesh Reference Values Solution Solution Methods Solution Initialization Osolution Initialization Results Graphics Results Reports Plats Plats Plats	Discrete Phase: Sample Histogram Summary - Unavailable Heat Exchanger - Unavailable Set Up Parameters Help	Dudola bidde_botinterior-1-1 Didde_botinterior-1-1 bidde_bot_inde_wall-bidde_bot_onuer Didde_bot.inde_wall-bidde_bot_onuer bidde_bot.outer bidde_bot.outer bidde_tot.outer bidde_tot.outer <					
		Compute Write Close Help					
		Mesh					
		Note: zone-surface: cannot create surface from sliding interface zo Creating empty surface. Note: zone-surface: cannot create surface from sliding interface zo	ne . ne				

Note that "Mass flow rate" is already selected.

Here you can play around to see the mass balance through each boundaries. Since most of our boundaries are single circular element, it is expected that the imbalance at **each boundary alone be zero**. We will check the mass flow rate though two boundaries.

First, the external boundaries. In the "*Flux Reports*" window, locate and highlight "*farfield1*" and "*farfield2*" and hit "*Compute*". The mass flow rate though that boundary is now printed in the command window.

2	A:VAWT - N	loving Frame of	Reference Fluent@Julio-PC	[2d, dp, pbns, rke] [ANSYS	Academic Teachi	ing Introductory]	×
<u>File Mesh Define Solve Adapt Surface D</u>	<u>D</u> isplay <u>R</u> eport Para <u>l</u> lel <u>V</u> iew	<u>H</u> elp					
💼 📴 🕶 🖬 🖉 🎯 🖉 🔂 🖉	/ [@ 洗 開 - 🗆 - [12]	• 🔳 • 📕 R	∧ ☶ ■				
Setup	Reports		1: Mesh v				
B B Models	Reports						ANSYS
Materials Cell Zone Conditions	Fluxes Forces				2000 HILLING		Academic
Boundary Conditions	Projected Areas - Unavailable Surface Integrals			Flux Reports			
Dynamic Mesh	Volume Integrals Discrete Phase:	Options	Boundar	es	E = Results		
Reference Values	Sample Histogram	Mass Flow I Total Heat	late blade_r blade_r	ght-side2-wall-blade_right_outer ght_in	^		
Solution Methods	Summary - Unavailable Heat Exchanger - Unavailable	O Radiation H	eat Transfer Rate blade_t blade_t	ght_outer op-interior-1-1	_		
Solution Controls		Boundary Types	<pre>blade_t</pre>	op-side1-wall-blade_top_in op-side2-wall-blade_top_outer			
Solution Initialization		axis exhaust-fan	hlade_t	op_in			
Run Calculation		fan	farfield farfield				
e 🔗 Results		Boundary Name	Pattern hub-sid	1-wall-hub_inner			
Animations	Set In Darameters		Match hub_inr	er er	~		
H→ C→ Plots H→ D→ Reports	occopiii rometerom				<	•	
🖲 🏇 Parameters & Customization	Help	Save Output P	rameter		Net Results ((kg/s)	
					7.399905e-	-09	
			Compute	Write Close Help]	¥	
			Mesh				May 02, 2018 ANSYS Fluent Release 16.2 (2d, dp, pbns, ike)
			1				^
			Surface group = inter	ior-surface_body, id =	= 18, name = in	terior-surface_body	
			Surface group = inter	ior-surface_body, id =	= 18, name = in	terior-surface_body	
			Surface group = inter	ior-surface_body, id =	= 18, name = in	terior-surface_body	
			Surface group = inter	ior-surface_body, id =	= 18, name = in	terior-surface_body	
			Surface group = inter	ior-surface_body, id =	= 18, name = in	terior-surface_body	
			Surface group = inter	ior-surface_body, id =	- 18, name - in	terior-surface_body	
			Mas	s Flow Rate	(kg/s)		
				farfield1	73.500001		
				tart10102	-73.500001		
				Net 7.3	3999847e-89		
			<				×

Note that 73.5kg/s get into out domain and virtually everything leaves. The imbalance is 7.4e-9kg/s which is negligible considering the total amount of flux in the system. Nice!

Now, let's check the imbalance inside the hub. For that we only have one boundary completely circling the zone, *hub_inner* (note that *hub_outer* is essentially the same boundary).

Proceed similarly as before and check the mass flow through the *hub_inner* boundary. You could also select *hub_outer* and the result would be almost identical.

	A:VAWT - Mo	oving Frame of	Reference Fluent@Jul	lio-PC [2d, dp, pbns, rke] [ANSYS Academ	nic Teaching Introdu	ictory]	_ 🗆 🗙
<u>File Mesh Define Solve Adapt Surface D</u>		lelp						
💼 📴 🕶 🖬 🕶 🎯 🕲 🕾 🖓 🔍 🕀 🖉	🖊 🔍 洗 開・ 🗖 ・ 🖗 ・	. .	∧ ☶ ■					
B Setup	Peporte		1: Mesh	×				
General	Topono							ANCVC
Models	Reports							ANJIJ 16.2
Materials	Fluxes							110.2
Cell Zone Conditions	Projected Areas - Unavailable			Elun Demente			×	Academic
blade bot interior. 1.1 (interior)	Surface Integrals	-		Flux Reports				
1 blade bot-side1-wall-blade bot in (Discrete Phase:	Options		Boundaries		Results	LESSES A	
t blade_bot-side2-wall-blade_bot_ou	Sample	Mass Flow	Rate	blade top-side1-wall-blade to	op in			
	Histogram Summary - Linavailable	O Total Hea	t Transfer Rate	blade_top-side2-wall-blade_to	op_outer			
→ D‡ blade_bot_outer (interface)	Heat Exchanger - Unavailable	Radiation	Heat Transfer Rate	blade_top_in				
blade_right-interior-1-1 (interior)				farfield 1				
blade_right-side1-wal-blade_right_i		Boundary Type	es Alle	farfield2				
Diade_right-side2-wai-blade_right_i		axis	^	hub-interior-1-1 hub-side 1-wall-hub inner			開始目的目	
→ blade_right_in (interface)		fan		hub_cide2_wall-hub_outer				
blade_ingit_dotter (interiocc)		inlet-vent		hub_inner				
t blade top-side1-wall-blade top in (interior-blade bot				
		Boundary Nam	e Pattern	interior-blade_right				
jt blade_top_in (interface)	Set Up Parameters		Match	[interior-blade_top	•		✓ 000000000000000000000000000000000000	
blade_top_outer (interface)						< >		
	Help	Save Output	Parameter			Net Results (kg/s)		
tarfield2 (pressure-outlet)	1 Cap					-3,424147e-10	1	
hub aide 1 wall bub interior)						1		
tub-side2-wall-hub_outer (wall)								
tub inner (interface)			Cor	mpute Write Close	Help			
thub_outer (interface)		and the second se						
			Mesh					May 02, 2016
								Ans to Fident Release 10.2 (20, up, pbils, ike)
·····]‡ interior-blade_top (interior)								^
□]‡ interior-inner (interior)			Surface group =	interior-surface_bo	dy, id = 18, na	ame = interior-su	rface_body	
interior-surface_body (interior)								
t wall blade bot (wall)			Surface group =	interior-surface_bo	dy, 1d = 18, na	ame = interior-su	rtace_body	
1 wall blade right (wall)			Surface group =	interior-surface bo	du. id = 18. na	ame = interior-su	rface bodu	
twall_blade_top (wall)			ourrace group	incerior barrage_be	ay, 10 10, 10	ine incerior se		
🕀 🚺 Mesh Interfaces				Mass Flow Rate	(kg	J/S)		
				Caucialda	70 500	004		
Reference Values				farfield2	-73 500	3001		
Solution								
Solution Controls				Net	7.3999047e	2-09		
Monitors			Г Г	H 53 8/				
Solution Initialization			l	mass Flow Kate	(kg	J/S)		
Calculation Activities				huh inner	-3.42414670	-10		
Run Calculation								
E 💓 Results				Net	-3.4241467e	2-10		
ui u								~
· · · · · · · · · · · · · · · · · · ·			<					ان ‹

Remember to deselect farfield1 and farfield2 before hitting compute!

You should get an imbalance of 3.42e-10kg/s which is essentially zero. Cool!

Tip speed ratio (TSR)

In practice, this is extracted directly from the Boundary Conditions, since we will essentially check that the velocity at the wall is zero. Therefore the purpose of this check is more to verify if we had correctly inputted the mathematical model into Fluent.

To calculate the TSR we first need to extract the velocity from CFD-Post. Since our reference is the value of velocity at r=0.04m, we need to find some way to extract the velocity of fluid particles in touch with the blade at that particular location.

One can plot the velocity vectors and read off the legend. However this is quite imprecise.

One of the ways to do this trick is to plot the velocity distribution along the X coordinate for the whole surface of the **right** blade, and then extract the value at x=0.04m. Since the "wall" entity is a closed line, the plot should also be circular. As the blade is rectangular, we should expect abrupt change in velocity very close to the maximum and minimum X. Let's do it!

First thing to do is to create a Polyline over the wall of the right blade. Select Location > Polyline.



Name it "wall right" and for "Method" select "Boundary Intersection". For "Boundary List" select "blade_right symmetry 1" and for "Intersection With", select " wall_blade_right". Click Apply.

Geometry	Color F	Render	View		
Domains	All Domain	s			•
lethod	Boundary	Intersect	on		-
Boundary List	blade_rig	ght symme	try 1		•
Intersect With	wall_blac	le_right			•

Next, insert a chart (Insert > Chart). Name it "Veloc at blade". Under "Data Series" tab, change the Location to the created "wall right".

Under "X Axis" tab, change the Variable to "X".

Under "Y Axis" tab, change the Variable to "Velocity in Stn Frame v". This is the velocity in the Stationary frame of reference (our interest. CFD Post uses the variable Velocity as relative to the rotating frames). We are taking only the y component because we know that the velocity of the blade should be only in the y direction at that location. Click Apply.

The chart should look like this. The point of interested is marked by the dashed lines. Also notice that at the edges of the plot there is an abrupt change in velocity, as expected. The "closed loop" plot expect is in fact happening, but the curve collapsed into a single line. You can she the curves separated if you choose "*Velocity in Stn Frame*" as Y *Variable* instead



The point is slightly above the halfway between 0.165 and 0.170. Recall from pre-analysis that we calculated the expected value as 0.1676m/s. The value is virtually the same, indicating that we might have inputted the right mathematical model into the tool.

But we're not done! To calculate the TSR we still have to perform one short step. Recall that TSR=veloc blade/veloc wind, so all we have to do is divide the calculated velocity by 10m/s, the wind speed.

So, our TSR is 0.01676.

Angular velocity in Steady state

One could have noticed that the moment coefficient (Cm) is not zero, which means that there are some extra torque applied on the turbine. That is in fact true, and we could even use this Cm to calculate the power coefficient (Cp) for this setup.

It's possible to find at which velocity a VAWT like that would spin in case we applied the very same conditions as of this tutorial. At that angular velocity, there would be no extra torque applied on the turbine: if it were to spin faster, a negative torque would slow it down, if it were to spin slower, a positive torque would spin it up.

To find that velocity using the Moving Frame of Reference approach, you can vary the angular velocity (under "*Cell zone conditions*", open the "*inner*" condition and change its angular velocity) until Cm is approximately zero.

If you do that, you will find an angular velocity of about 80RPM.

Mesh Refinement

One can also perform a mesh refinement study. Duplicate your project, import the new refined mesh (found here), "Clear Generated Data" for Results cell, refresh the project and launch Fluent. Remember to use Parallel (2, 3 or 4 cores, no more than that) if available. Also, compute the initialization from *farfield*. Run the simulation for about 2000 iterations. This is the new velocity contours.



The (absolute value of) mass imbalance between farfield1 and farfield2 also change, dropping to about 1.48e-9. This is a reduction of exactly 5 times! Yay.

Note: this new mesh uses completely different meshing techniques. The ideal would be using the very same method as before. This new mesh is more "professional made" than the previous one.

Go to Exercises

Go to all FLUENT Learning Modules