# Vertical Axis Wind Turbine (Part 1) - Physics Setup

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

## **Physics Setup**

## Launch Fluent

To launch FLUENT, double click Setup under Project Schematic.

Check the option "Double Precision". This will make the simulation more accurate, since Fluent will use twice as many bits in the calculations.

Dimension	Options	
2D	Double Precision	
🔘 3D	Processing Options	
Display Options	<ul> <li>Serial</li> </ul>	
<ul> <li>Display Mesh After Reading</li> </ul>	O Parallel	
🖌 Embed Graphics Windows		
🖌 Workbench Color Scheme		
Do not show this panel again		
🛃 Show More Options		

You can leave the rest as Default. Select Parallel if desired. Now press OK to launch FLUENT.

## **Check Mesh**

It is always a good practice to check the mesh, especially if you are importing a mesh that was not created by you. It is not rare to get matching errors; particularly with meshes that have multiple Cell-Zones like our mesh.

In order to check the mesh, you can click on Mesh > Check, which is located under the upper tab next to File.



You can also note that when Fluent is first launched, an option to check the mesh is available under "General". You can see this highlighted in the above figure.

Lastly, you can type "mesh/check" and hit Enter in the messages box.

## Create mesh interfaces

As briefly discussed before, we need to start by creating the interfaces between the different zones of the mesh.

To account for the rotation of the mesh, we cannot simply create one single mesh. Instead, we had to create sub regions or Zones of meshing.



The mesh created is "non-conformal", i.e. the nodes do not match across the interface, so we need to tell Fluent that the adjacent cell across that interface share information.

To make the "Interfaces" option appear, first go to "Boundary Conditions". Select the first Zone on the list, "blade\_bot\_in," and change the type to "Interface". Click OK in the popped window to keep the default name.

A:	VAWT - Moving Frame of Reference Fluent@Julio-PC	[2d, dp, pbns, lam] [ANSYS Academic Teach
<u>F</u> ile <u>M</u> esh D <u>e</u> fine <u>S</u> olve <u>A</u> dapt S <u>u</u>	rface <u>D</u> isplay <u>R</u> eport Para <u>l</u> lel <u>V</u> iew <u>H</u> elp	
📕 📄 🕶 🖌 🚽 🚳 🞯 👘 🖓	£ ⊕ ∥ @ 洗 ‼ + 🗆 + 🕼 + 🔳 + 😫 ≈ /	N 😝 MOI
Setup General General Cell Zone Conditions Cell Zone Conditions Cell Zone Conditions Cell Zone Conditions Cell Zone Conditions Cell Zone Conditions Solution Methods Solution Methods Solution Methods Solution Initialization Calculation Activities Monitors Calculation Activities Run Calculation Calculation Calculation Calculation Calculation Pots Cell Zone Conditions Cell Zone Conditions Calculation Activities Calculation Ca	Boundary Conditions         Zone         blade_bot_in         blade_cright_in         blade_right_outer         blade_top_in         blade_top_outer         faffeld1         faffeld2         interior-blade_top         interior-blade_top         wall_blade_right         interior-blade_top         wall_blade_top         wall_blade_right         Phase         mixture         wall_blade_right         Phase         Type         mixture         valis         cexhaust-fan         opietwert         Parameters         Opintake-fan         Pressure-far-field         pressure-far-field         pressure-far-field         pressure-far-field         pressure-far-field         pressure-far-field         pressure-field         pressure-field         velocity-inlet         wall	1: Mesh

Note that the "Mesh Interfaces" option appears on the main menu to the left.

Go ahead and set the other interfaces Boundary Conditions. There is no quick way to do that as Fluent does not allow you to Copy Interfaces Boundary Conditions. So, proceed as before and assign the following Zones to Interface:

blade\_bot\_in

blade\_bot\_outer

blade\_right\_in

blade\_right\_outer

blade\_top\_in

blade\_top\_outer

hub\_inner

hub\_outer

Now we're ready to finally create the interfaces!

In our case, each pair of Interface Boundary Condition will be an Interface Zone. So, there will be 4 zones in total.

Highlight "Mesh Interfaces" and click "Create/Edit..."

In the first box, "Mesh Interface", write "blade\_bot". That's the name of the interface. Under Interface Zone 1, select *blade\_bot\_in*. For Interface Zone 2, select *blade\_bot\_outer*. Click "Create". The window will not close by itself so go ahead and close it.

lesh interiace	Interface Zone 1		Interface Zone 2	
blade_bot	blade_bot_in		blade_bot_outer	
	=			
	blade_bot_in	<b>^</b>	blade_bot_in	^
	blade_bot_outer		blade_bot_outer	
	blade_right_in		blade_right_in	
	blade_right_outer		blade_right_outer	
	place_top_in	*	plade_top_in	Y
nterface Options	Boundary Zone 1		Interface Wall Zone 1	
Periodic Boundary Condition	· · · · · · · · · · · · · · · · · · ·			
Periodic Repeats	Boundary Zone 2		Interface Wall Zone 2	
Coupled Wall				
Matching	I			
Mapped			Interface Interior Zone	
eriodic Boundary Condition				
Type Offset				
	Y (m) 0			
Rotational     X (m)				
Auto Compute Offset				
Iransiational     Rotational     Auto Compute Offset				
Iransiational     Rotational     Auto Compute Offset      Iapped     Enable Local Tolerance				
Iransiational     Rotational     Auto Compute Offset      Iapped     Enable Local Tolerance				
Iransiational     Rotational     Auto Compute Offset      Iapped      Enable Local Tolerance      1				
Franslational     Rotational     Auto Compute Offset      Auto Compute Offset      Enable Local Tolerance      1      Local Edge Length Factor				
Iranslational     Iranslational     X(m)     0     Auto Compute Offset     Aapped     Enable Local Tolerance     1     Local Edge Length Factor				

Setting wall blade bot (mixture) ... Done.

Repeat this and create the remaining 3 interface zones.

Mesh Interface	Interface Zone 1	Interface Zone 2
blade_bot	blade_bot_in	blade_bot_outer
blade_right	blade_right_in	blade_right_outer
blade_top	blade_top_in	blade_top_outer
hub	hub_inner	hub_outer

Be careful to select the correct in-outer pairs!!

Note: you do not have to close the Mesh Interface window all the time. After clicking "Create" you can go ahead and give the name for the new Interface and create it.

By the end of this step, your window should look like this.

<b>-</b>		A.VAWT - WOVING Flame OF Reference	e Fident@Julio-PC [2d, up, pblis, lattij [Alv:
<u>File M</u> esh D <u>e</u> fine <u>S</u> olve <u>A</u> dapt S <u>u</u>	rface <u>D</u> isplay <u>R</u> eport Para <u>l</u> lel <u>V</u> iew <u>H</u> elp		
💼 🛛 📻 🕶 🔙 🕶 🎯 🖉 🔂 🔂	Q @ ↗∥@ 次 懦▾ □▾∥救▾ ■▾∥?> ≈ i	∧ ☶ ≝	
File       Mesh       Define       Solve       Adapt       Su         File       Setup       General       Setup       Setup       Setup         General       Models       Setup       Setup       Setup       Setup         General       Models       Solution       Setup       Setup <td< th=""><th>rface Display Report Parallel View Help Q Q I I I I I I I I I I I I I I I I I I</th><th>A. VAW T - Moving frame of Kelerici</th><th></th></td<>	rface Display Report Parallel View Help Q Q I I I I I I I I I I I I I I I I I I	A. VAW T - Moving frame of Kelerici	
		Mesh Setting blade_bot_outer (mixt Setting blade_right_in (mixtu Setting blade_top_in (mixture	ure) Done. re) Done. ) Done.

## Set Solver informations

Now, resuming...

Select the first option under "Setup": "General".

Here we can change key information about the solver, whether it's steady or transient, planar or axisymmetric, etc.

For our problem, you can leave the default options i.e., Pressure-Based, Steady and Planar.

Lastly, click "Units" and change the "angular-velocity" unit to rpm.



## Set the Model

Here is where we tell FLUENT all the simplifications for the model it can assume. For instance, here you specify if the model is Inviscid, Laminar or Turbulent, if you should consider the Energy Equation (for supersonic flows), and other options.

For us, we will be considering turbulence, and we will use the k-epsilon Realizable model.

Highlight "Models" and double click the third item in the list, "Viscous - Laminar". Select "k-epsilon (2 eqn)". Change "k-epsilon Model" to Realizable. Retain the rest as default. Click Ok.



## Materials

We will use the default properties for air. Go ahead and check if they are correct.

Highlight "Materials" and double click "air" under "Fluid".

Density: 1.225 kg/m3

Viscosity: 1.7894e-05 kg/m-s

Materials Cell Zone Conditions Cell Zone Conditions Boundary Conditions Dynamic Mesh Reference Values Solution Methods Solution Controls Solution Initialization Calculation Activities Results Properties Properties Properties Properties Properties Properties Density (kg/m3) constant L225 Viscosity (kg/m3) constant Viscosity (kg/m3) constant Visco	Setup	Material	S		1: Mesh	¥	
Solution   Solution Methods   Solution Controls   Monitors   Solution Initialization   Chemical Formula   Chemical Formula   Chemical Formula   Chemical Formula   Fluent Fluid Materials   Results   Parameters & Customization   Help	General P Models Materials Cell Zone Conditions Cell Zone Conditions Mesh Interfaces	Materials Fluid air Solid aluminu	m				
Solution       Name       Order Materials Dy         Solution Controls       iar       fluid       Order Materials Dy         Animations       Chemical Formula       Fluent Fluid Materials       Order Materials Dy         Results       Results       Fluent Fluid Materials       Fluent Pluid Materials         Pois       Properties       Density (kg/m3)       Constant       V Edit         Parameters & Customizatio       Leep time       Constant       V Edit         Help       Liz25       Liz25       Liz25	Reference Values		2		Create/Edit Materials		×
Anometers & Customizatio     Properties     Pr	Solution Methods		Name		Material Type	U	Order Materials by
Results Results Graphics Plots Porperties Properties User-Defined Database Properties Viscosity (kg/m3) Constant Create/t Help Help Itemport Constant Itemport Itemport Create/t Help Itemport The point of the point	Solution Initialization		Chemical Formula		Fluent Fluid Materials		Chemical Formula
Piots       Peroperties         Parameters & Customizatio       Create/c         Viscosity (kg/m3)       constant         Li225       Viscosity (kg/m3)         Viscosity (kg/m3)       constant         Li225       Viscosity (kg/m3)         Viscosity (kg/m3)       constant         Li255       Viscosity (kg/m3)         Viscosity (kg/m3)       constant         Viscosity (kg/m3)       constant         Li255       Viscosity (kg/m3)         Viscosity (kg/m3)       constant         Viscosity (kg/m3)       constant     <	Results				Air Mixture	v	User-Defined Database
Parameters & Customizatio       Density (kg/m3)       constant       v       Edit         Create/E       Viscosity (kg/m4)       constant       v       Edit         Help       Help       1.7894e-05       1.7894e-05	Animations		Properties		•		
Create/E Viscosity (kg/m-5) constant v Edit Help	⊢☆ Reports ▶ Parameters & Customizatio		Density (kg/m3)	constant	♥ Edit		
1.7394e-05		Create/E	Viscosity (kg/m-s)	constant	♥ Edit		
		Help		1.7894e-05			
v					v		

## **Cell Zone Conditions**

Here we specify to fluent the material of each meshed zone (usually correct by default, unless we create a new material).

Also, we set this if we are solving a moving mesh problem or moving frame of reference problem (our case!). We can also set the centroid of each zone and the angular velocity of that zone here.

Highlight "Cell Zone Conditions". Note that our problem is made out of 5 zones:

- blade\_bot: corresponds to the mesh around the bottom blade
- blade\_right: similar as above, but for the right blade
- blade\_top: similar as above, but for the top blade
- fluid-surface\_body: corresponds to the big circular mesh around the main geometry of the turbine.
- inner: corresponds to the zone between the blades inside the hub.

These are the same zones described previously when creating the mesh interfaces. We will have to edit parameters for each of them.

#### fluid-surface\_body

Highlight fluid-surface\_body and click "Edit ... ".

No options should be selected here, and the material must be set to air. This is the default. You can verify that and click Ok.

#### inner

Highlight *inner* and click "*Edit..*". Here we will set that the inner portion of the hub is spinning, adding more terms to the equations to account for local acceleration, even though the mesh is not actually moving.

Right under "Material Name", select the first option "Frame Motion". We want to specify the angular velocity and the origin about which the mesh turning.

From the geometry, the centroid is at the global origin (0,0). Verify that that is inputted.

From the problem statement, the turbine is spinning at 40rpm, so go ahead and input 40 to "Rotational Velocity". This is the **absolute** velocity. Note that "ab solute" is selected under "Relative Specification".

The rest you can keep as default. Click Ok.

. <u>                                    </u>	Q Q / : Q ⊼ ‼ + □ + : /2 + ■ + : 2 ≈ Λ ;; *	
Setup       Image: Setup of the	Cell Zone Conditions	
Solution Methods	E Fluid	×
Solution Initialization Calculation Activities Calculation Results Caphics Calculation Cal	Immer     Material Name air     V     Edit       Material Name air     V     Edit       Material Name air     V     Edit       Material Name     Laminar Zone     Source Terms       Phase     Porous Zone     Fixed Values       Mathwesh Motion     Porous Zone     3D Fan Zone     Embedded LES       Reference Frame     Mesh Motion     Porous Zone     3D Fan Zone     Embedded LES       Relative Specification     UDF       Relative To Cell Zone absolute     Zone Motion Function     none       Porous F     Station-Axis Origin     X (m) 0     constant       Y (m) 0     constant     V     Translational Velocity	s Fixed Values Multiphase
	Copy To Mesh Motion	Fluent Release 16.2
	No zone with name blade_right No zone with name blade_right No zone with name blade_top=	bs in (type wall) outer (type wall terior) (mixtur ght_in (type wal t-side2-wall-blade_right_outer (type t-interior) (nixtur ide1-wall-blade_top_in (type wall)

#### blades

Highlight *blade\_top* and click "*Edit...*".

Again, select "Frame Motion" option. Now, in order to make the blades rotate and keep the same relative position to the hub, select the "inner" Cell Zone under "Relative Specification". The relative velocity will be zero.

We need to do some basic geometry to find the rotating center. The blades are located 0.04m from the center of the turbine. The top blade is 120 degrees from the (positive) x-axis. So:

x=0.04\*cos(120)=-0.02m

y=0.04\*sin(120)=0.034641m

Go ahead and input that for X and Y of "Rotation-Axis Origin (Relative)".

💼 📴 🕶 🔒 🕶 🚳 🞯 🖄 🔂 🧐	2 ① / ◎ 九 腊 - 🗆 - 🍂 - 🔳 - 💱	2 2 A 🖽 🎬	
🖻 🍓 Setup	Cell Zone Conditions	1: Mesh v	
General     Beneral     Beneral     Beneral     Materials     Materials     Gallzone Conditions     Beneral Conditions     Beneral Interfaces     Sommic Mesh     Reference Values     Sefference Values	Zone Diade pot blade ropt blade ropt blade ropt hud surfac_body mner		
Solution Methods			
Monitors	2	Fluid	
Calculation Activities	Zone Name		
Run Calculation	blade_top		
Graphics	Material Name		
Animations     Plots	Frame Motion		
Reports	Fixed Values		
🗄 🎰 Parameters & Customizatic	Porous Zone		
	Reference Frame Mesh Motion Porous Zone 3D Fan 2	one   Embedded LES   Reaction   Source Terms   Fixed Values   Multiphase	
	Relative Specification	UDF	
	Relative To Cell Zone inner	V Zone Motion Function none V	
	Potation Avia Origin (Delative)		
	X (m) -0.02 constant		
	x (n)		
	0.034641 constant V	ANOVO EL	unt Deleges 16-3 (9d
	Rotational Velocity (Relative)	Translational Velocity (Relative)	ient Release 10.2 (20
	Speed (rpm)	X (m/s)	
	Constant		
	Copy To Mesh Motion	Y (m/s) 0 constant v zone IDs	•••
	1	e bot out	(type wall) (m) ter (type wall)
		ype inter	rior) (mixture),
		ade_right	t_in (type wall) t outer (tupe wa
	OK	Cancel Help (type int	terior) (mixture
		P_top_in	(type wall) (mi ter (tupe wall)
		No zone with name blade top-interior-1-1 (tupe inter	rior) (misturo)
		No zone with name hub-sīdei-wall-hub_inner (type wa No zone with name hub-sīde2-wall-hub_outer (type wal No zone with name hub-interior-1-1 (type interior) (	<pre>(mixture), s (mixture), s (mixture), s (mixture), skipp</pre>
		No zone with name hub-side1-wall-hub inner (type wal No zone with name hub-side2-wall-hub_outer (type wal No zone with name hub-interior-1-1 (type interior) ( Done.	(mixture), s ll) (mixture), s ll) (mixture), s (mixture), skipp

Do the same for the right and bottom blades. Select "Frame Motion" and make it relative to the "inner" cell zone. The needed values for centroid and angular velocity are summarized below.

Zone name	Centroid (X,Y)	Angular velocity (RPM)
blade_top	(-0.02, 0.034641)	0
blade_bot	(-0.02, -0.034641)	0
blade_right	(0.04, 0)	0

#### Don't forget to select the relative motion!

It would be good to go back and check each cell zone to avoid messy errors in the future!

We are now ready to set the other boundary conditions!

## **Boundary Conditions**

Remember that we had already set some Boundary Conditions, before doing the Mesh Interfaces. However we still need to tell FLUENT what is wall, and specify some pressures and velocities.

#### Velocity at the inlet

First, specify the velocity at the inlet.

Locate and highlight "farfield1". Change its Type to "velocity-inlet".

File Mesh Define Solve Adapt S	A:VAWT - Moving Frame of Reference Fluent@Ju
💼 📴 - 🕞 - 🗃 🞯 🖾 💠	Q Q ↗ @ 次 !!! ▾ □ ▾ !! !? ▾ ■ ▾ !! \$}
Setup General Cell Zone Conditions Cell Z	Boundary Conditions Zone Diade_bot_in Diade_bot_outer Diade_right_in Diade_right_outer Diade_right_outer Diade_top_outer Interior-biade_bot interior-biade_top interior-inner interior-biade_top wall_biade_bot wall_biade_right Edit Ciexhaust-fan interface Display Mesh Per Maxue Velocity-inlet Velocity

Click "Edit...". In "Velocity Specification Method", change to "Components". Set "X-Velocity (m/s)" to 10.

Under "Turbulence", change the "Specification Method" to "Intensity and Length Scale". Set the "Turbulence Intensity (%)" to 5 and the "Turbulent Length Scale (m)" to 1.

2	Velocity Inlet		×
Zone Name farfield1			
Momentum Thermal Radiation Specie	s DPM Multiphase U	DS	_
Velocity Specification Method	Components		~
Reference Frame	Absolute		~
Supersonic/Initial Gauge Pressure (pascal)	0	constant	-
X-Velocity (m/s)	10	constant	-
Y-Velocity (m/s)	0	constant	-
Turbulence		۰ 	_
Specification Methed	Intensity and Length Scale	~	
	Turbulent Intensity (%	(6) 5 F	1
	Turbulent Length Scale (r	n) 1	]
OK	Cancel Help		

It's very important to change the Specification Method for the Turbulence to "Intensity and Length Scale", otherwise your model won't converge.

#### Pressure at the outlet

∕∖∖

Locate and highlight "farfield2". Change its Type to "pressure-outlet".

A window will pop up. Leave "Gauge Pressure (pascal)" as default (zero). Apply the same turbulence conditions as describe above for Velocity at inlet boundary condition.



Note: is this window does not automatically pop up, click "Edit...", next to where you specified the Type of the boundary condition.

#### Wall

We need to tell FLUENT that the blades of the turbine are walls (i.e., no-slip condition, or no velocity normal or tangential to there), that are rotating together with the mesh around it.

To do that, select the "wall\_blade\_bot" zone. It should be already assigned to the Type "wall" (if not, do so). Click "Edit...".

Under "*Wall Motion*" change to "*Moving Wall*". On the new options that appeared, change the Motion to "*Rotational*". Now, specify the Rotation-Axis Origin with the same corresponding values as before. X=-0.02m, Y=-0.034641m. Click Ok.



Now, let's copy that to the other blades. Click "*Copy*", near to "*Edit...*". On the left list, select the boundary condition you just made, "*wall\_blade\_bot*". The two othe wall\_blade boundary condition appeared on the right column. Go ahead, click on both (the two must be highlighted), then click "*Copy*".

hub_outer interior-blade_pot interior-blade_right interior-blade_top interior-inner interior-surface_body surface_body wall_blade_bot wall_blade_right wall_blade_top			Copy Conditions	
			From Boundary Zone To Boundary Zones E = interior-blade_bot interior-blade_right interior-blade_top interior-inner	
Phase mixture	Type V wall	V ID 26	interior-surface_body wal_blade_bot wal_plade_not wal_blade_top	
Parameters Display Mesh	Operating Condition	itions	Copy Close Help	
Help				

Click Ok on the window that popped.

Lastly, we need to change the Origin of rotation of the copied wall boundary conditions. To do so, select each of them ("wall\_blade\_top" and "wall\_blade\_ri ght"), click "Edit..." and change the "Rotational-Axis Origin" to the respective centroid.

Boundary Zone	Centroid (X,Y)
wall_blade_top	(-0.02, 0.034641)
wall_blade_right	(0.04, 0)

Observe that the options selected for the first wall boundary condition were copied to each new one.

Now that we're done with Boundary Conditions, we're almost ready to run the simulation!

Save your project.

#### Go to Step 5: Numerical Solution

Go to all FLUENT Learning Modules