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

Author: Julio Sampaio Gabriel de Pieri, Cornell University

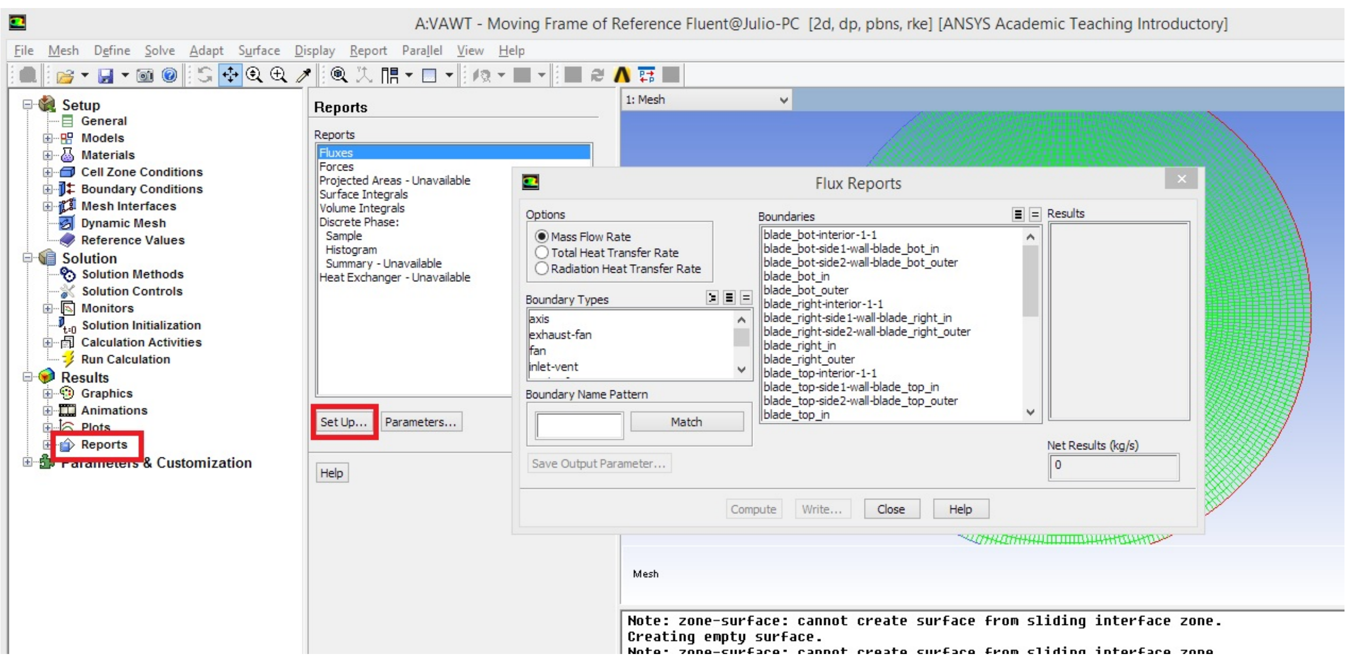
Verification & Validation

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***.

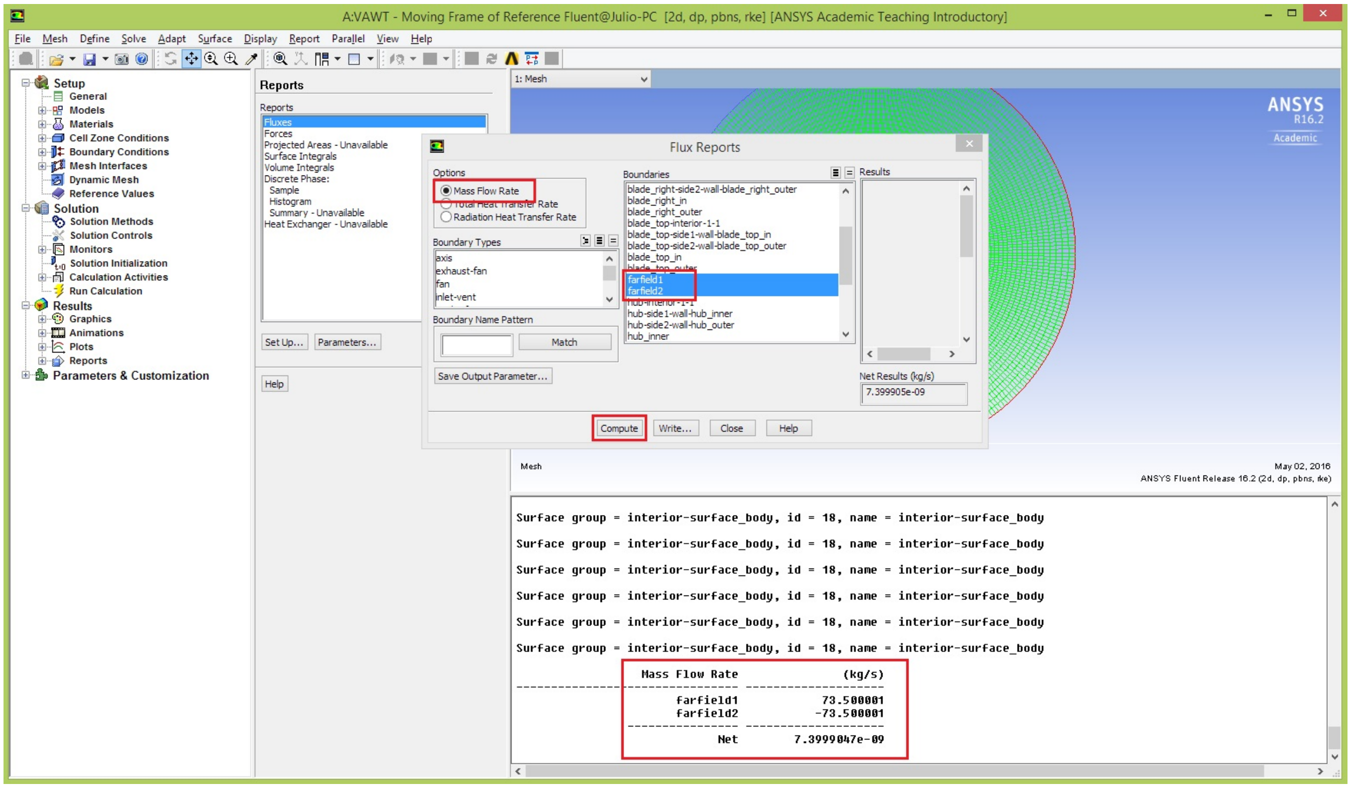
Highlight "*Reports*" in the left box. Then select "*Fluxes*" and click "*Set Up...*".



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.

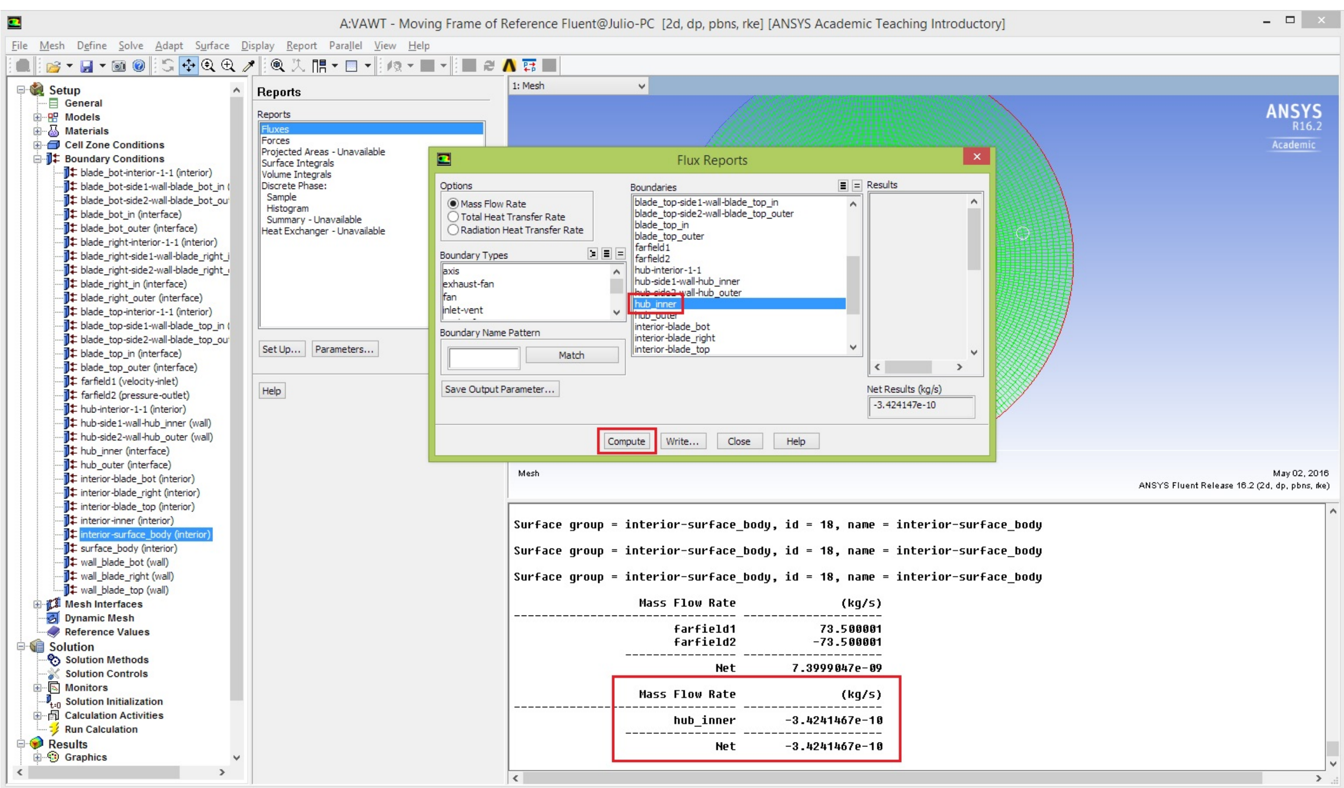
First, the external boundaries. In the "*Flux Reports*" window, locate and highlight "*farfield1*" and "*farfield2*" and hit "*Compute*". The mass flow rate though that boundary is now printed in the command window.



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.



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)

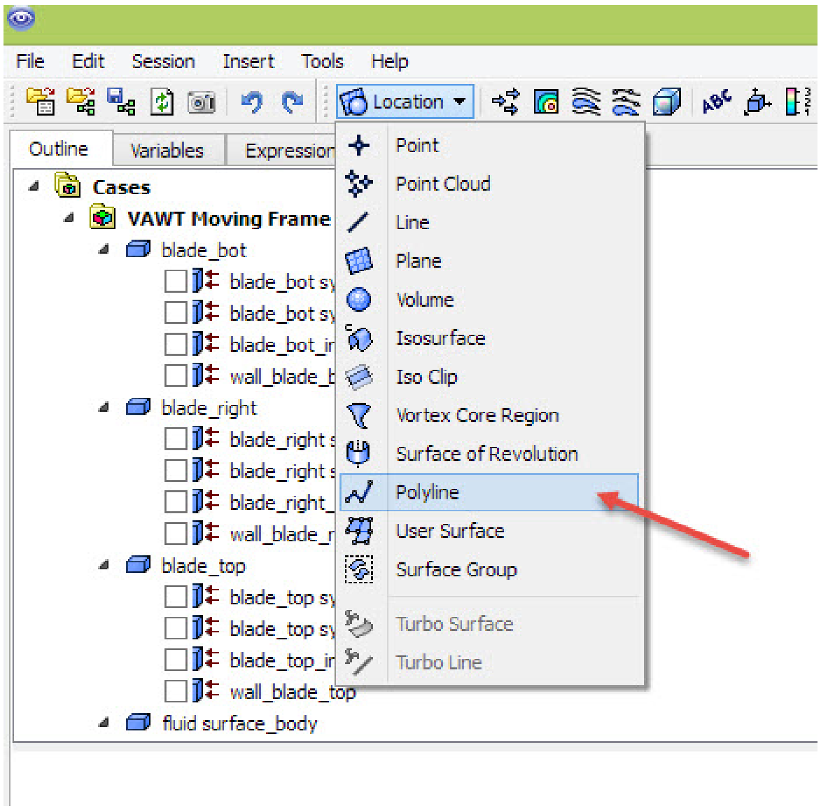
In practice, this is extracted directly from the Boundary Conditions, since we will essentially check that the velocity at the wall is zero. Therefore the purpose of this check is more to verify if we had correctly inputted the mathematical model into Fluent.

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.

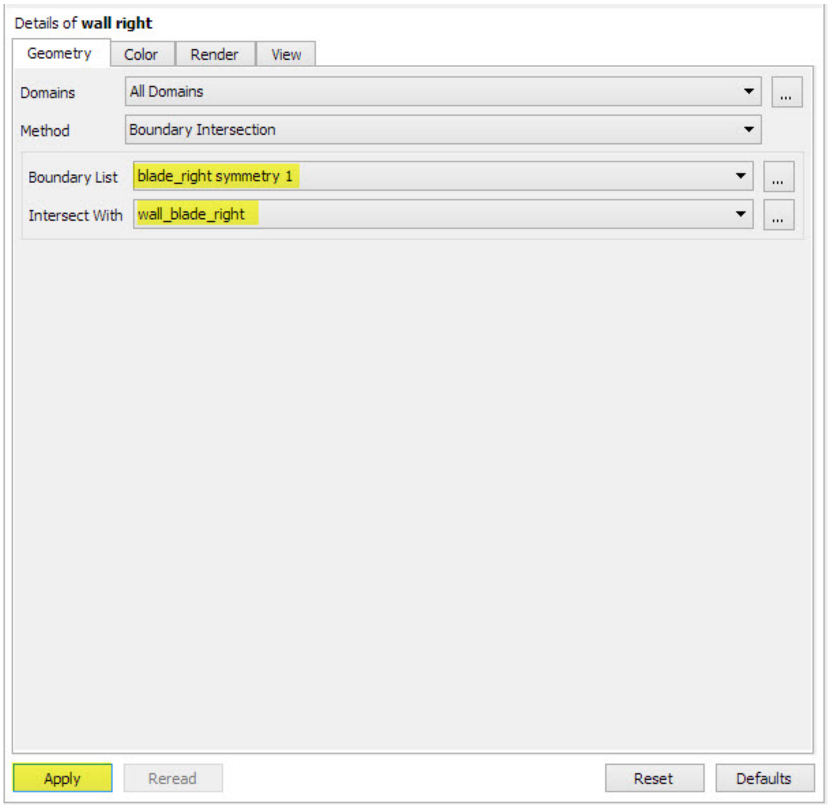
One can plot the velocity vectors and read off the legend. However this is quite imprecise.

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*.

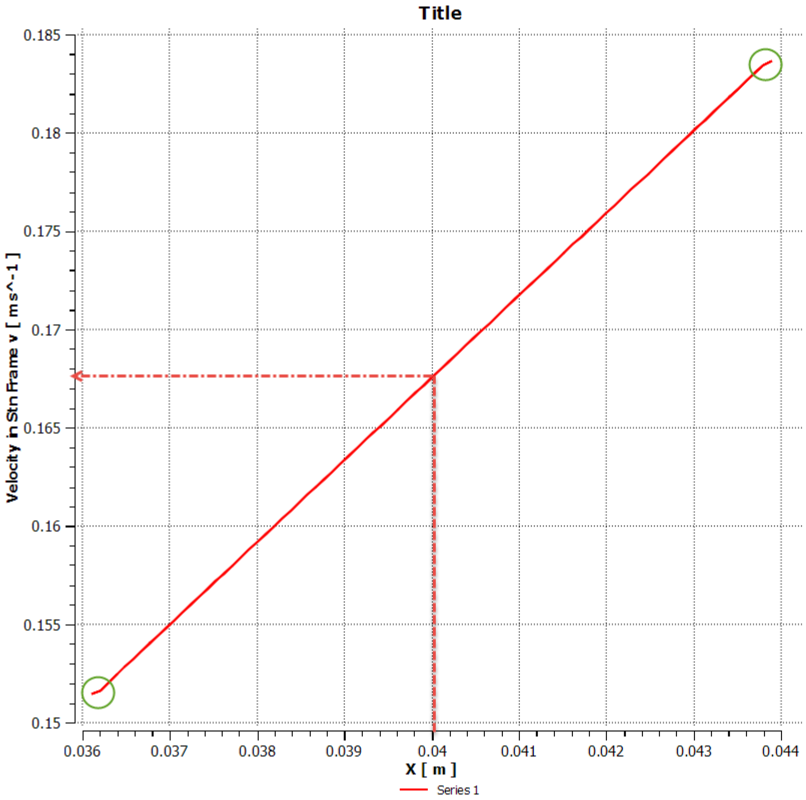


Next, insert a chart (*Insert* > *Chart*). Name it "Veloc at blade". Under "*Data Series*" tab, change the *Location* to the created "*wall right*".

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.

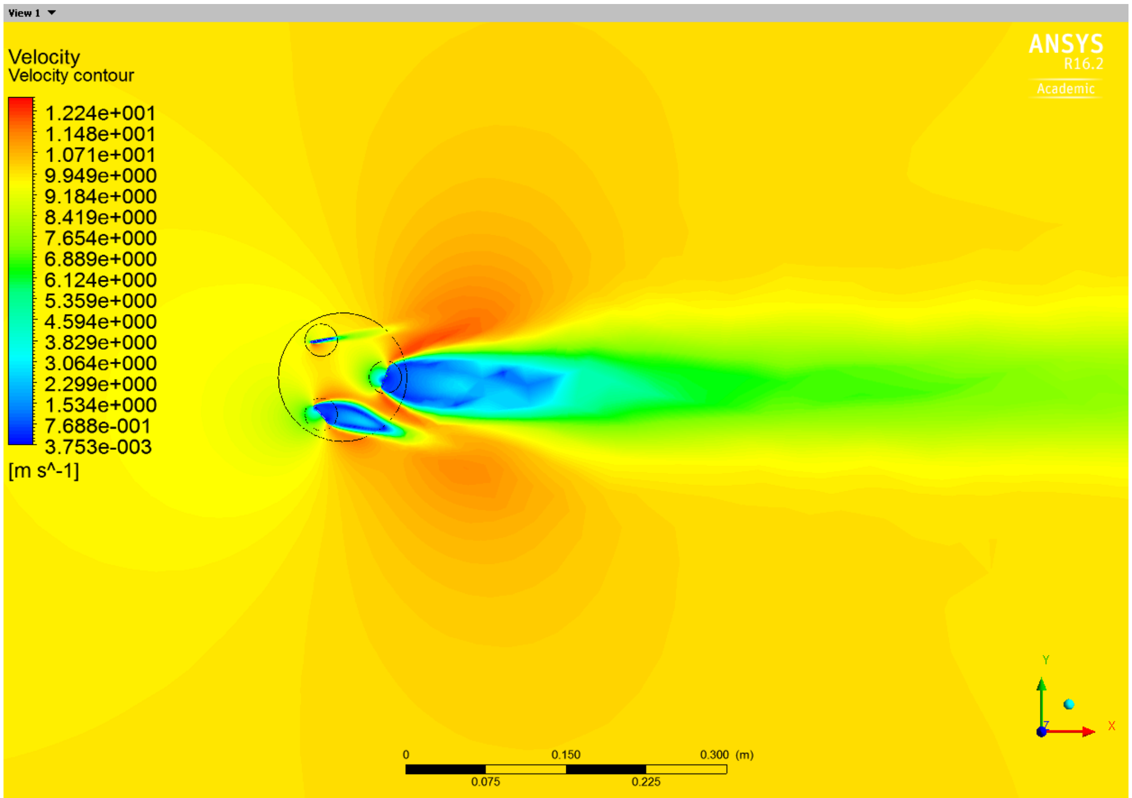
It's possible to find at which velocity a VAWT like that would spin in case we applied the very same conditions as of this tutorial. At that angular velocity, there would be no extra torque applied on the turbine: if it were to spin faster, a negative torque would slow it down, if it were to spin slower,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