

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

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](#)

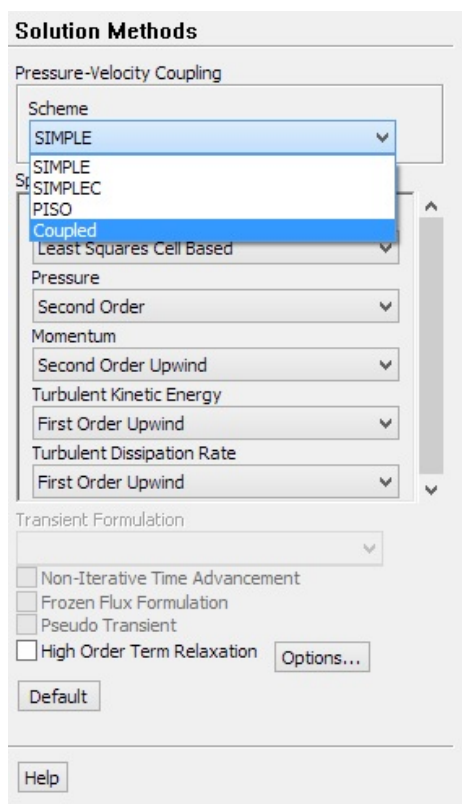
[Comments](#)

## Numerical Solution

### Solution Methods

Here we will specify various settings of the method used to solve. Each option change the order or accuracy of the calculation of each possible variable. On simple problems the default setting usually work pretty well. But once you start to make you own simulations of more complex problems, you soon will understand the importance of selecting the correct method here.

For this problem, just change the first option, "*Scheme*" to "*Coupled*". Leave the rest as default.



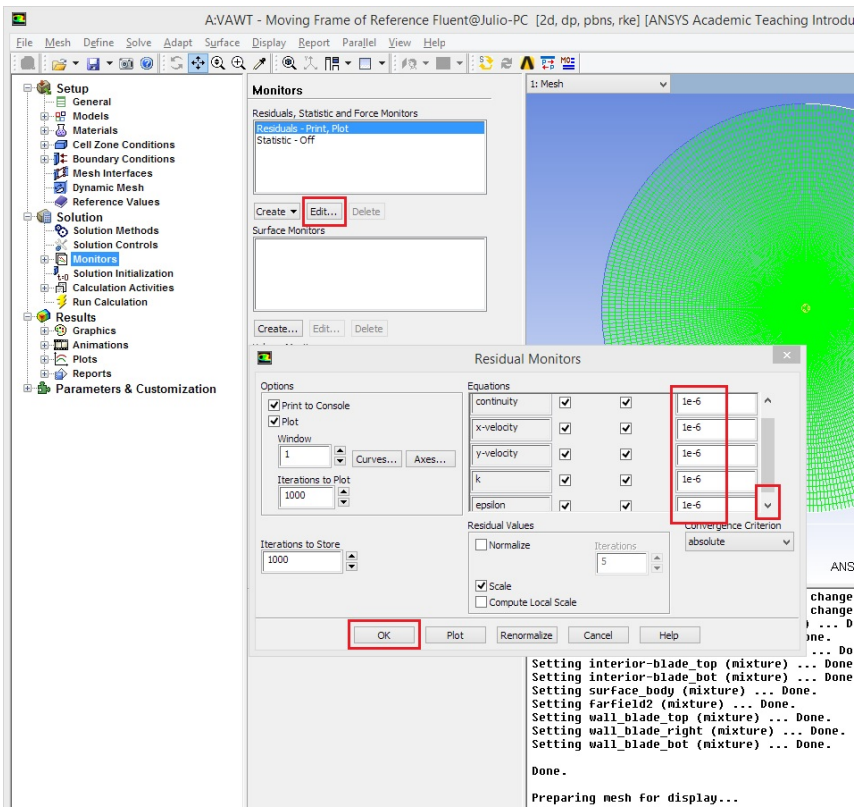
### Monitors

Here is where we define parameters to help checking for convergence. Residuals, the first one already embedded, is a measure of the linearization error. We define some threshold value and when the Residuals are below that value, the solution has converged.

The smaller the Residual criteria, less error is expected, however considerable more simulation time will be require.

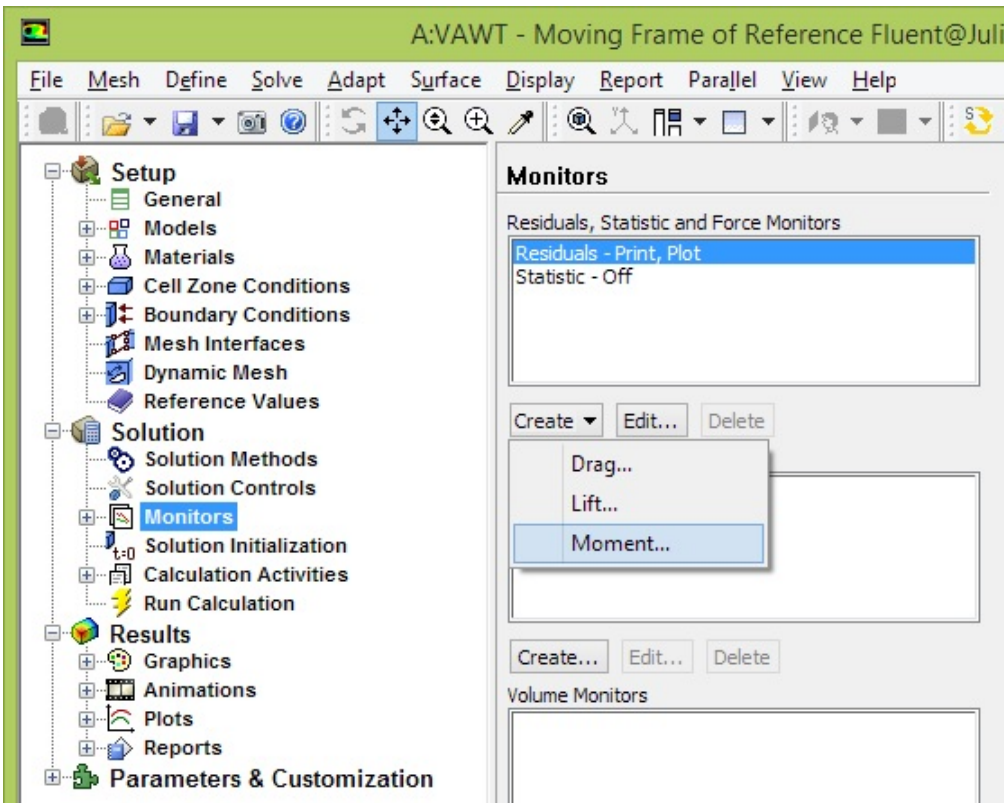
Since this is a simple problem, we can set the Residuals to be smaller than the 0.001 default value. For that, highlight the first line in the first "*Monitors*" box, "*Residuals - Print, Plot*", then click "*Edit...*".

The right most box containing the 0.001 number is the threshold we just discussed. There are 5 of them, continuity, x-velocity, y-velocity, k and epsilon (you might have to scroll down to see the last one). Change all values to "1e-6". (Note, "Ctrl C" does not work here...). Click Ok.

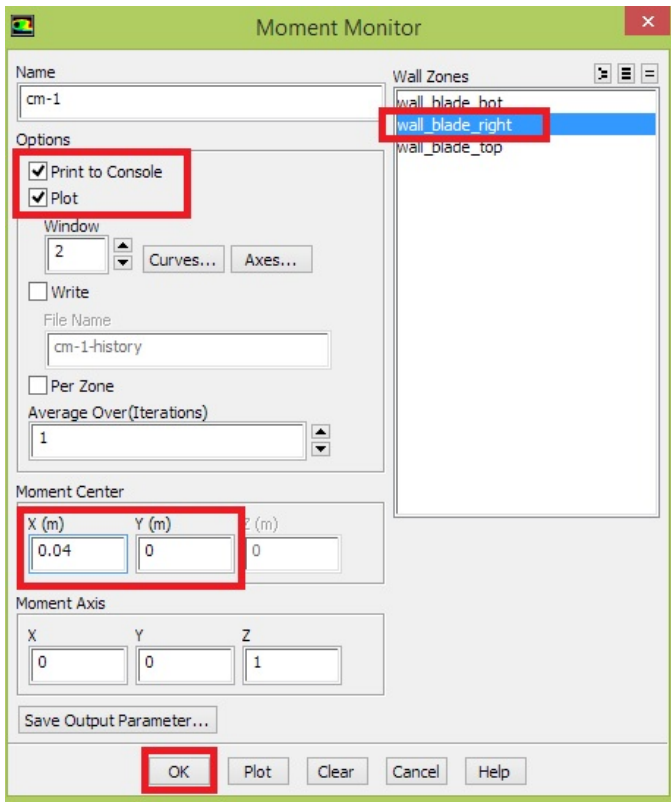


Now, let's create a monitor for Moment.

Click "Create > Moment...".



Under "Options", check the boxes to "Print to Console" and "Plot". Under "Wall Zones", select "wall\_blade\_right". Change the "Moment Center" to X=0.04m and Y=0. Click Ok.



Similarly, create a similar monitor for "*wall\_blade\_top*". Remember to change the Moment Center to  $X=-0.02\text{m}$  and  $Y=0.034641\text{m}$ .

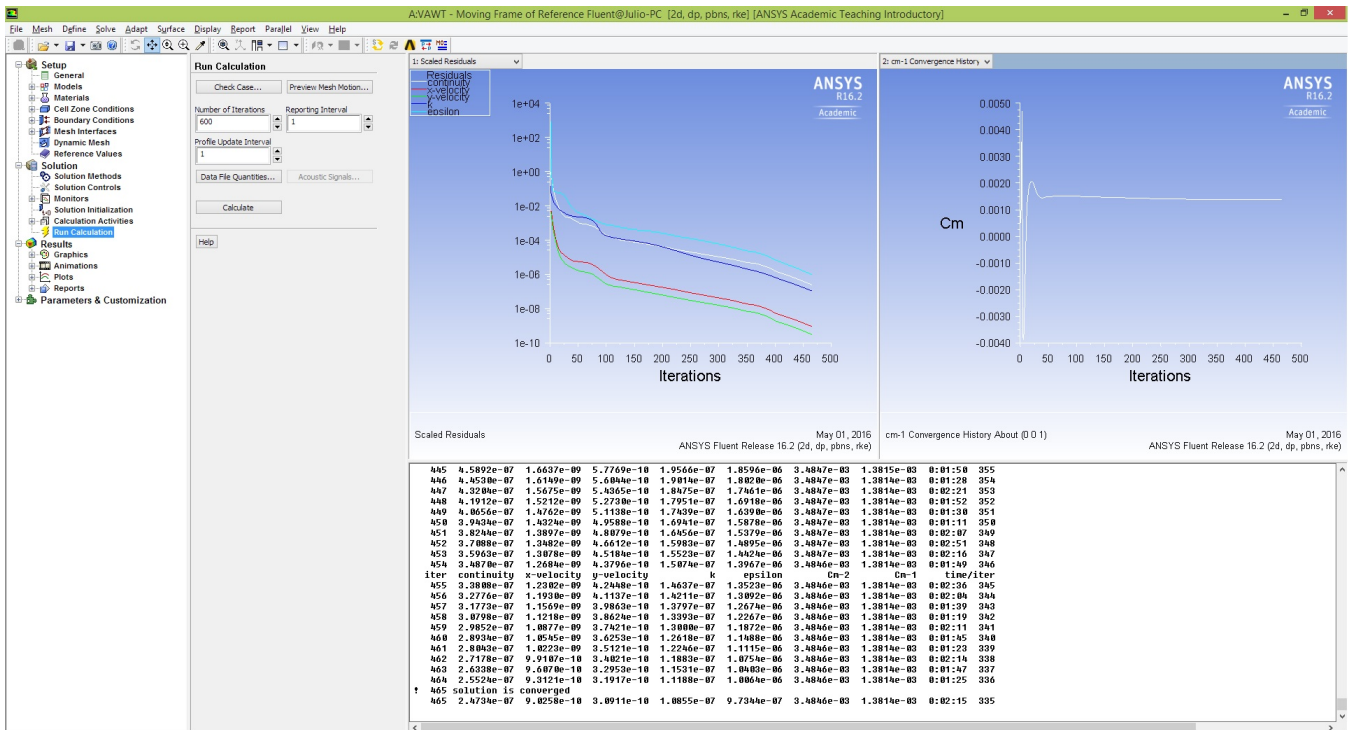
## Initialize the solution and run the calculation.

Highlight "*Solution Initialization*". Keep the "*Hybrid Initialization*" selected and click Initialize.

Highlight "*Run Calculation*". Change the Number of Iterations to 600 and click Calculate.

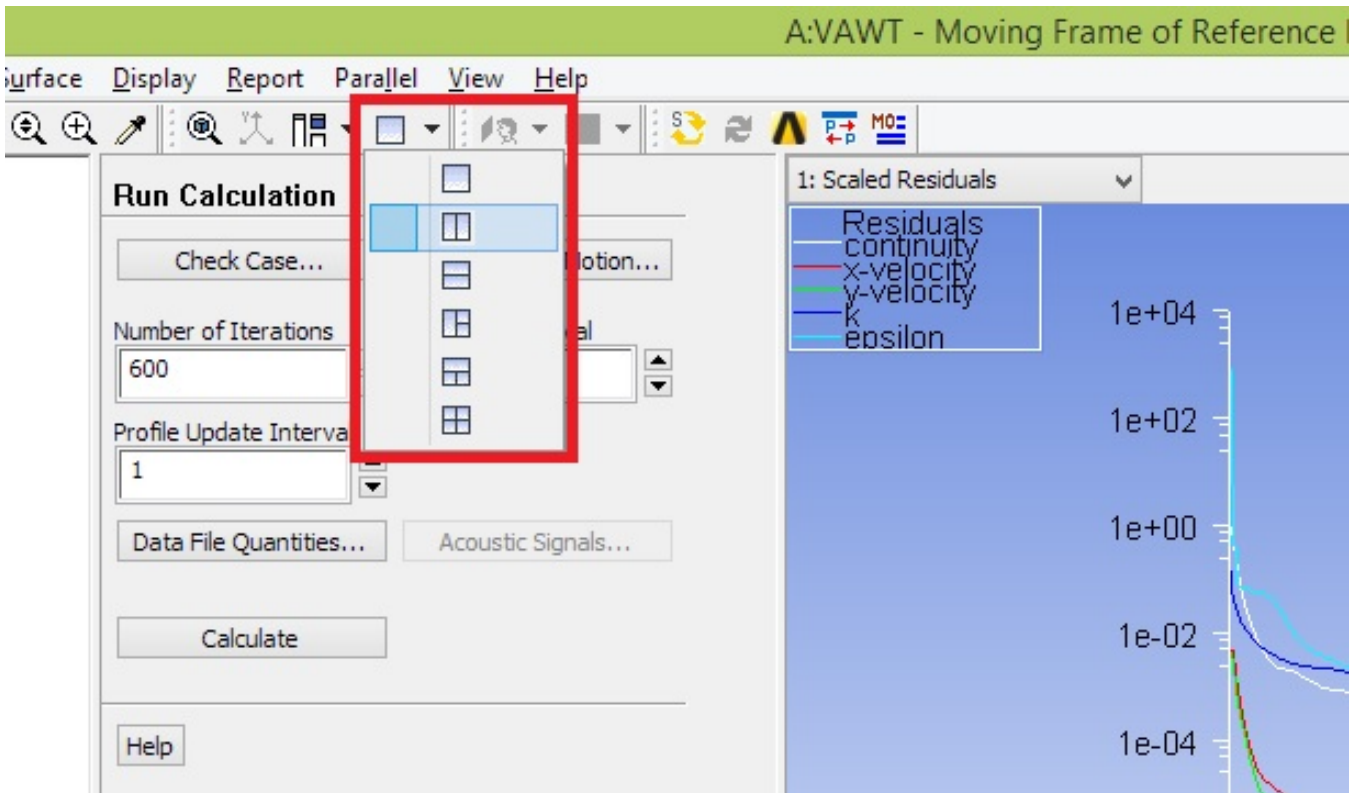
Note that on the first iterations, a message will be printed saying "Turbulent viscosity limited to viscosity ratio of  $1.000000\text{e}+05$  in *number* cells". Don't worry about this, as it will stop happening after 70 iterations or so.

The solution will converge in about 470 iterations. If it doesn't converge before 600 iterations, check that the Residuals are falling down and hit Calculate again for more 600 iterations.



You can change what is displayed by clicking on the drop-down menu right above the plot (upper-left corner). Select between Residuals or one of the two moment monitors created.

Note: if you want to display them side by side as the above picture, click on the "Arrange graphics window layout" and select the desired layout.



Save your project (if a window pop up, you can leave the default option selected and click Ok).

Next we will analyse the results in post-processing.

[Go to Step 6: Numerical Results](#)

[Go to all FLUENT Learning Modules](#)