Vertical Axis Wind Turbine (Part 1) - Numerical Results

Author: Julio Sampaio Gabriel de Pieri, Cornell University

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.
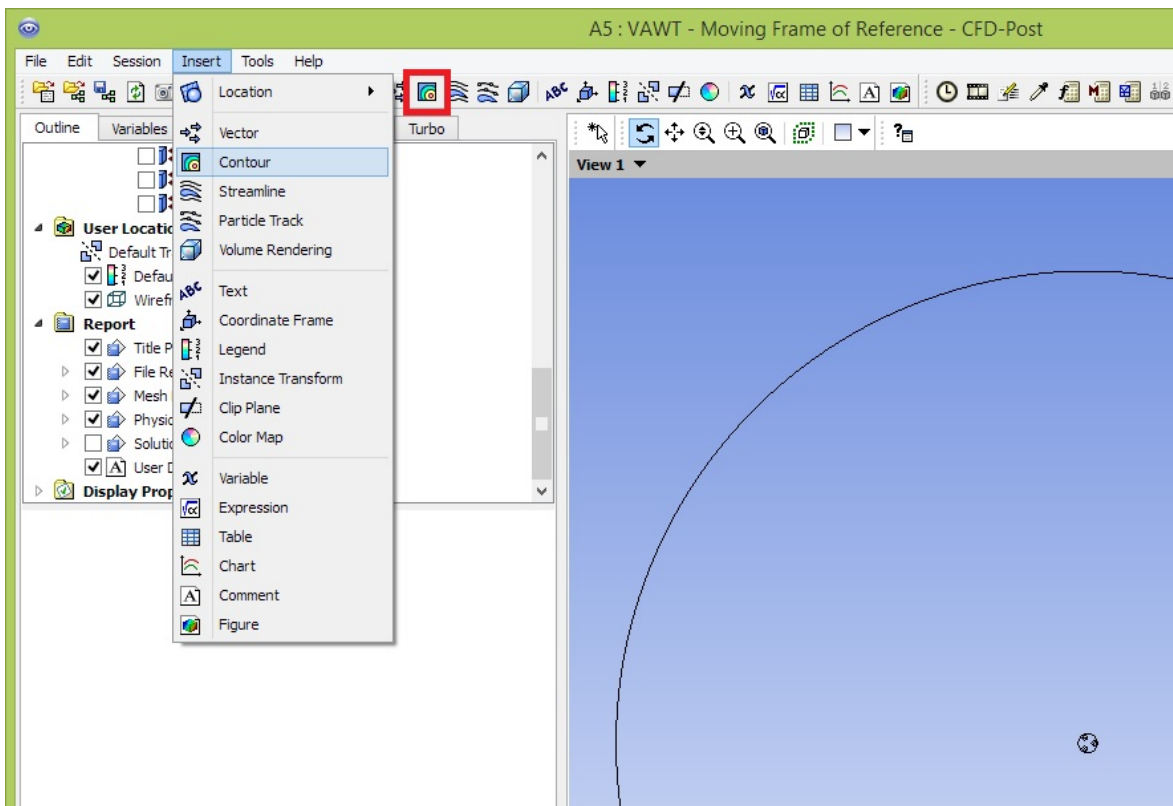
Velocity Contours

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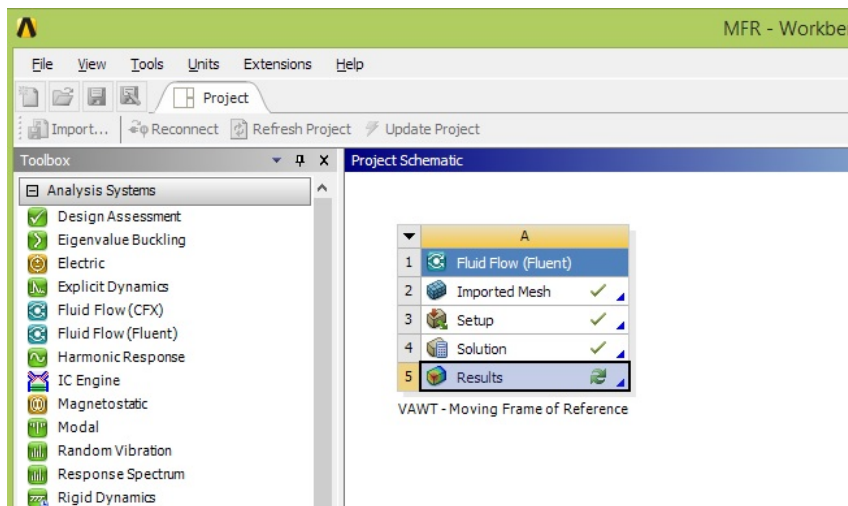
CFD-Post usually opens with an isometric view of the part. Since we're in a 2D analysis, this is not very useful for us. So, the first thing to do is click on the Z axis arrow (bottom-right corner of graphics window) to change the view.
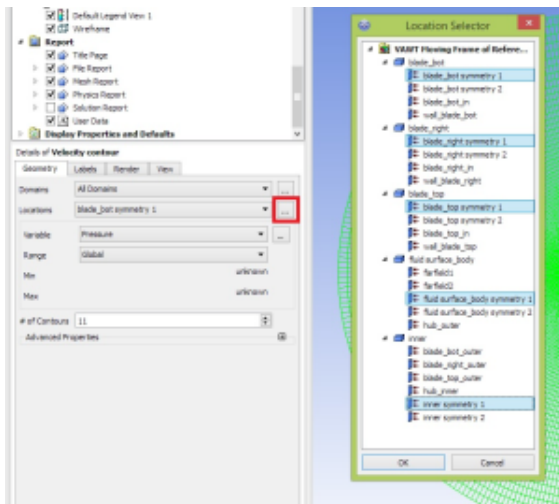
Now insert a Contour. Click on the *Contour* icon (or *Insert > Contour*)

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.
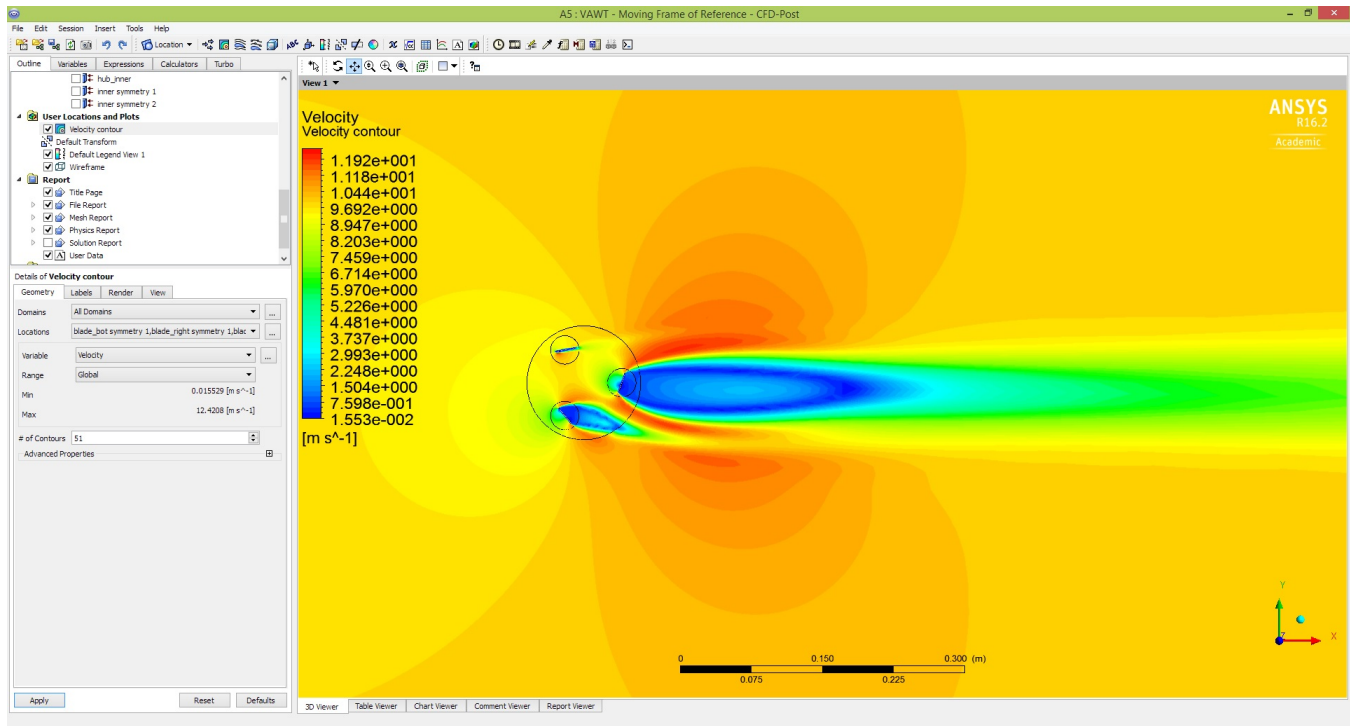

Variable: change is to "*Velocity*".

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.

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.
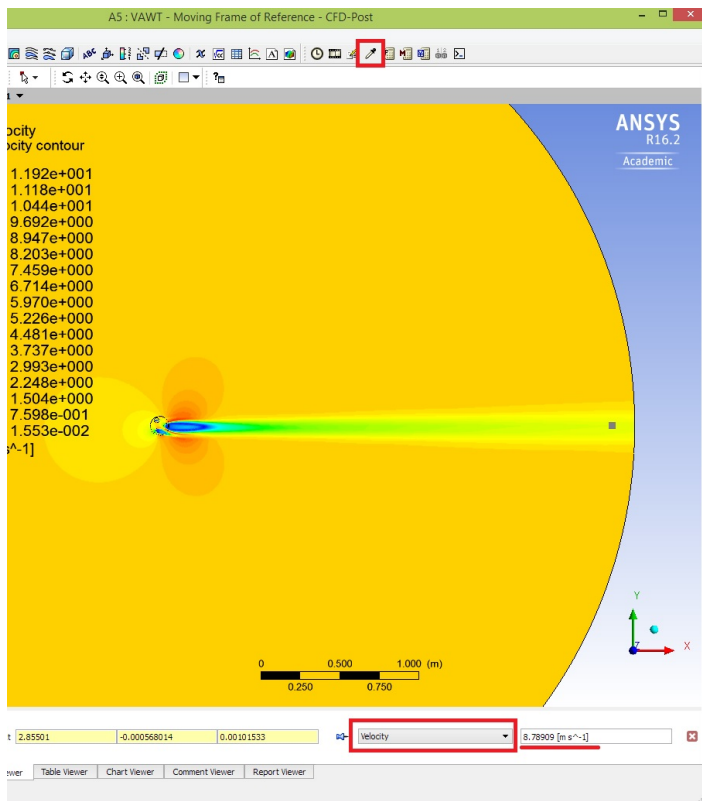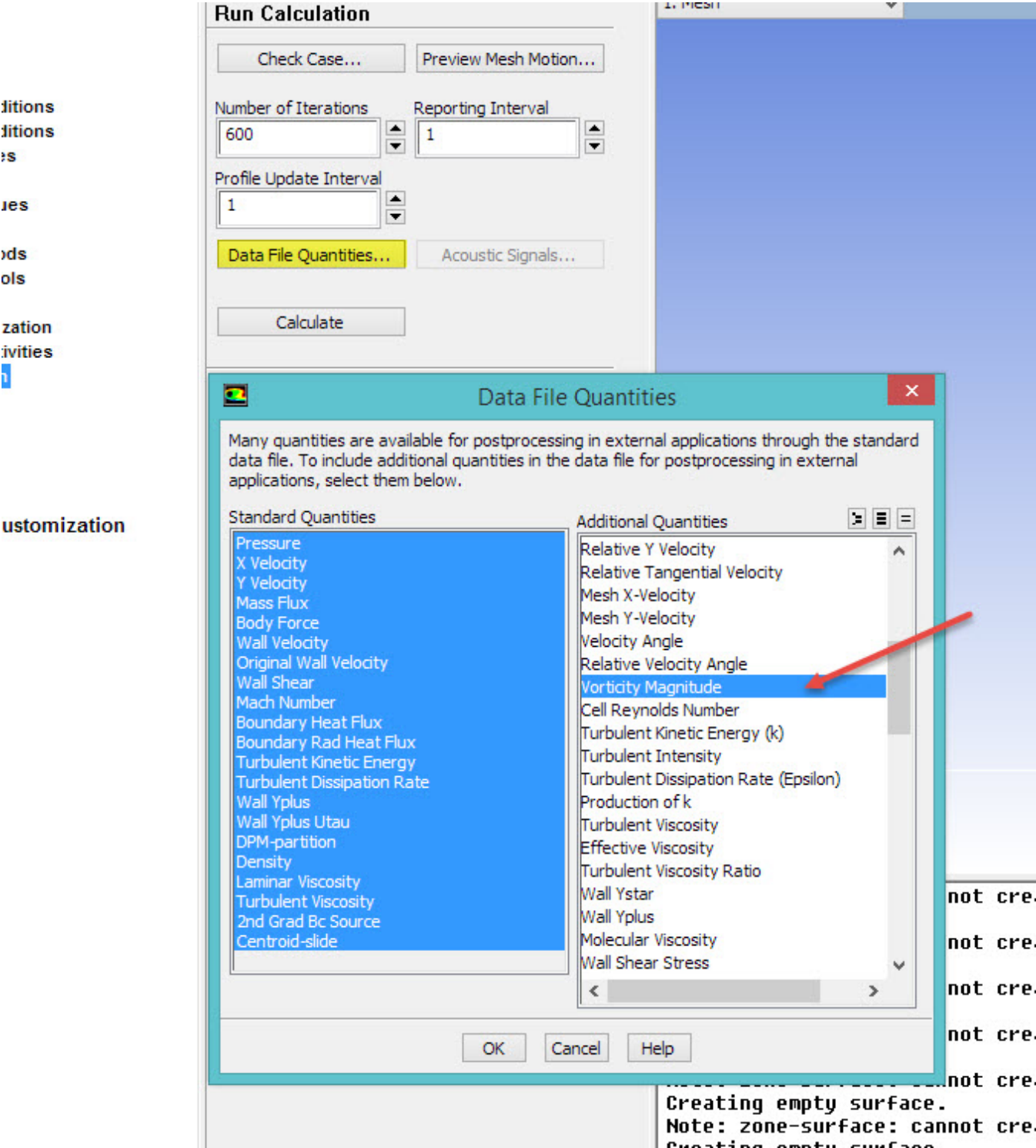
Vorticity Contours

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 the simulation again so Fluent can retrieve the desired value. It will converge in 2 iterations. Close Fluent and Refresh the project.
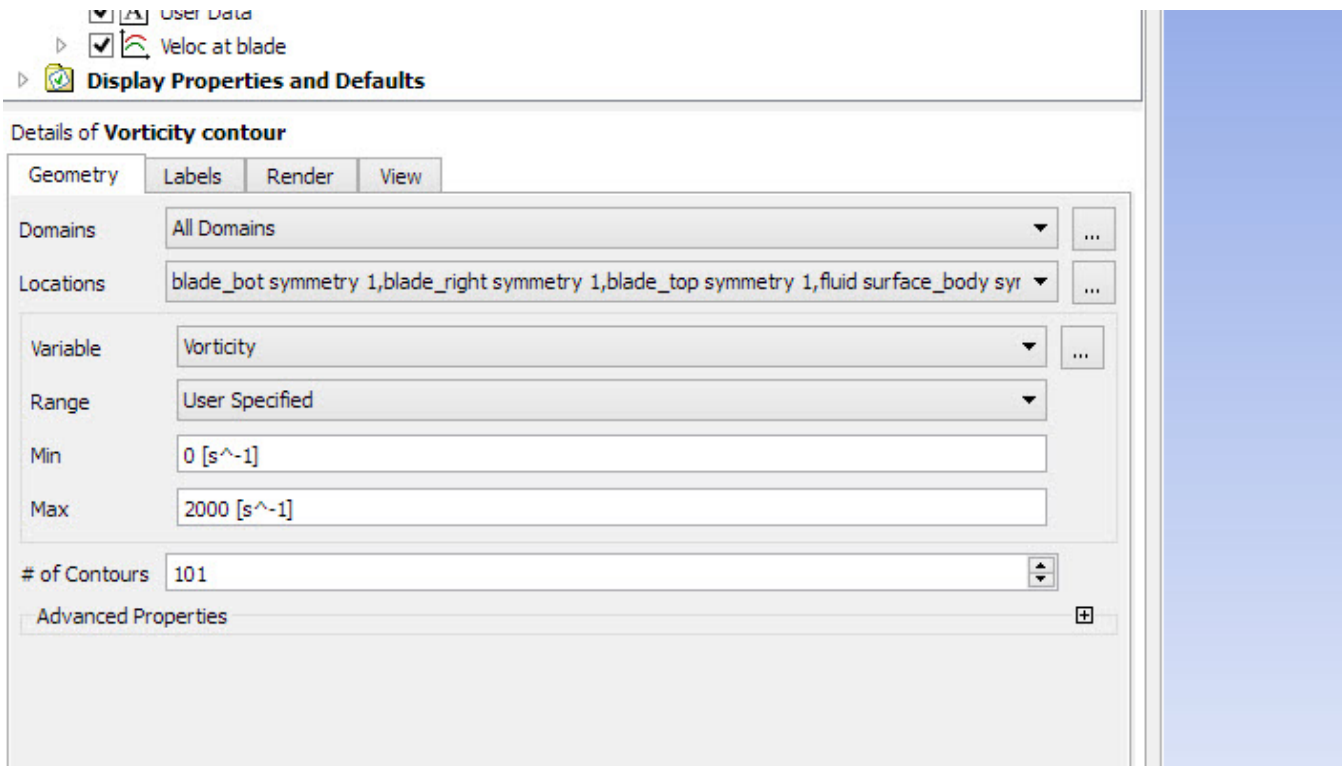
Open CFD-Post and create a new contour. Name it "Vorticity Contour".

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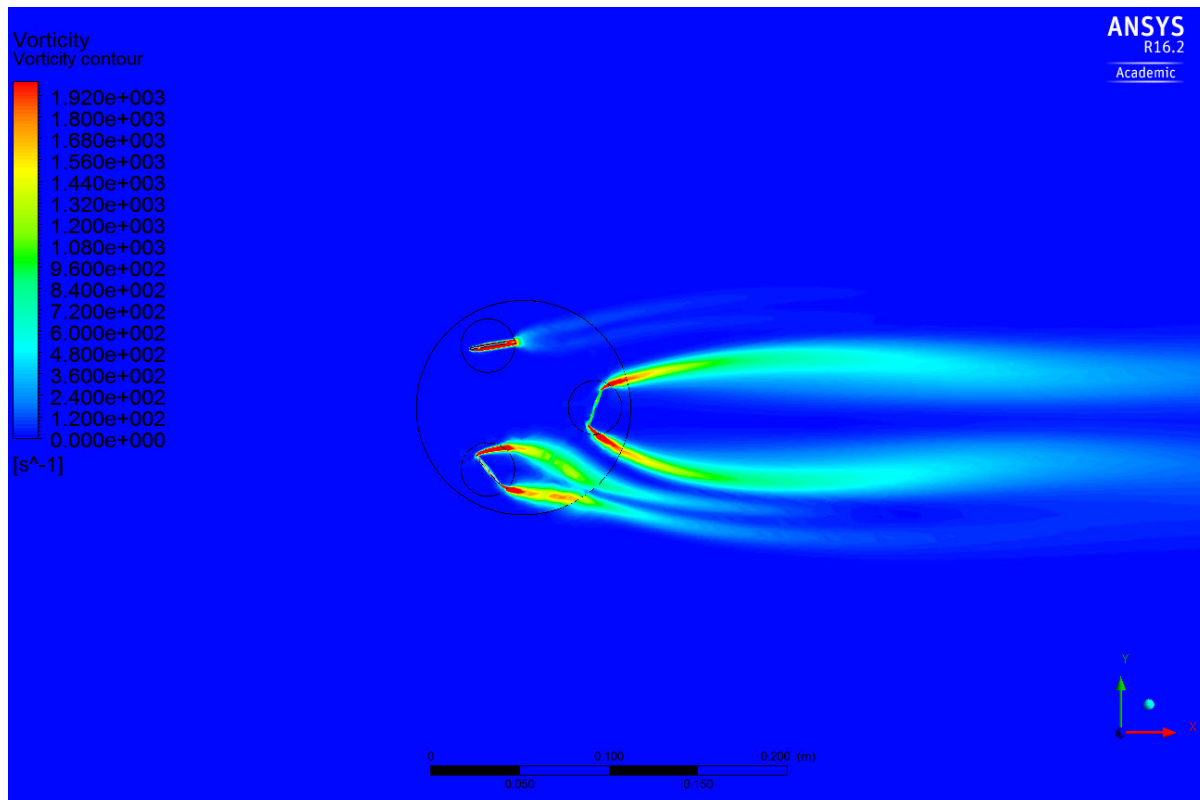
For "*Variable*", select "*Vorticity*".

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.

You should get something as the following figure.

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