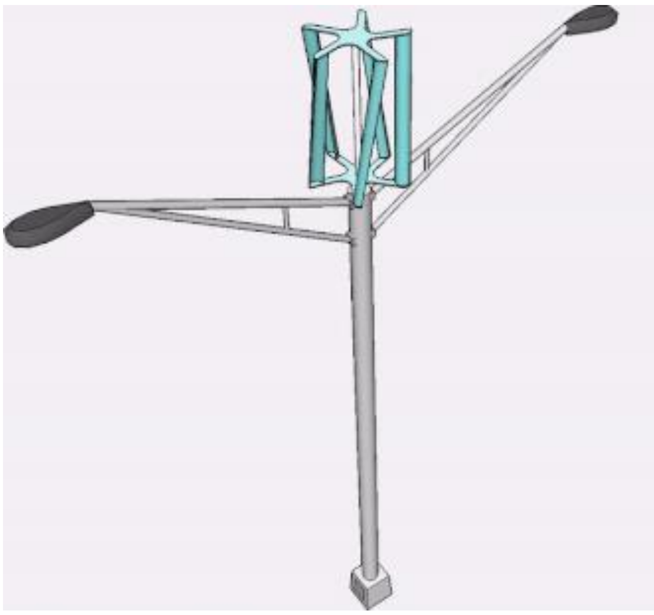
## *Analysis of a Vertical Axis Wind Turbine.*

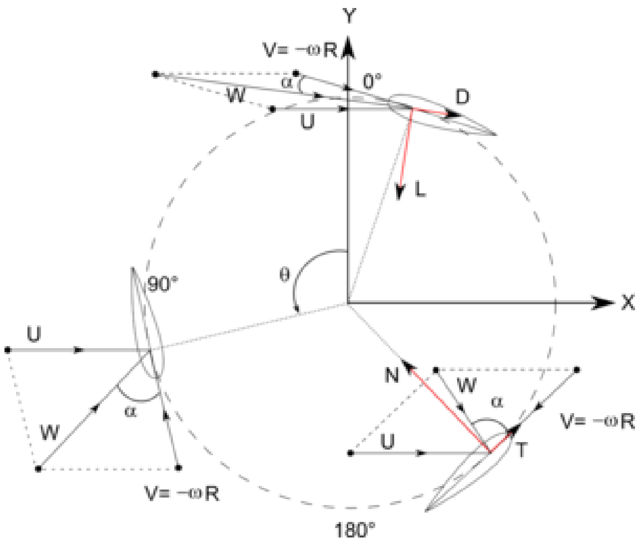
A **vertical-axis wind turbine** (**VAWT**) is a type of wind turbine where the main rotor shaft is set transverse t*o the wind while the main components are located at the base of the turbine. This arrangement allows the generator and gearbox to be located close to the ground, facilitating service and repair. VAWTs do not need to be pointed into the wind, which removes the need for wind-sensing and orientation mechanisms. Major drawbacks for the early designs (Savonius, Darrieus and giromill) included the significant torque ripple during each revolution, and the large bending moments on the blades. Later designs addressed the torque ripple by sweeping the blades helically (Gorlov type). Savonius vertical-axis wind turbines (VAWT) are not widespread, but their simplicity and better performance in disturbed flow-fields, compared to small horizontal-axis wind turbines (HAWT) make them a good alternative for distributed generation devices in an urban environment.

A vertical axis wind turbine has its axis perpendicular to the wind streamlines and vertical to the ground. A more general term that includes this option is "transverse axis wind turbine" or "cross-flow wind turbine.

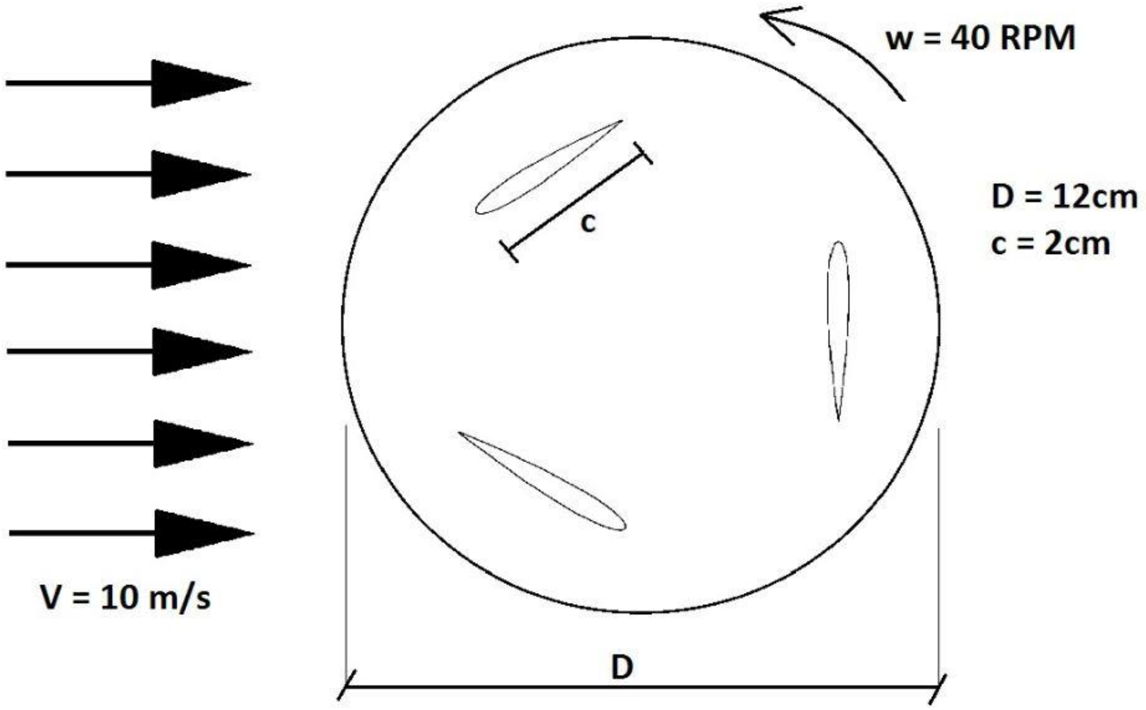
Drag-type VAWTs such as the Savonius rotor typically operate at lower tip speed ratios than lift-based VAWTs such as Darrieus rotors and cycloturbines.

Computer modelling suggests that wind farms constructed using vertical-axis wind turbines are 15% more efficient than conventional horizontal axis wind turbines as they generate less turbulence.





**Objective:**




Consider a uniform flow of V = 10m/s passing through a Vertical Axis Wind Turbine (VAWT) as sketched above. The VAWT has a diameter of 12cm and 3 equally spaced blades, each one with a chord length of 2cm. For simplification, consider that it spins with a constant angular velocity of 40 RPM*. The center of each blade is located 0.04m from the center of the hub.

Note that this is a *Darrieus* VAWT, which is Lift based; in contrast to the *Savonius* VAWT, which is Drag based. This is an intensive field of research, and at Cornell we have the Fluid Dynamics Research Laboratory, directed by Prof. Charles Williamson. In the last section, we will compare results with the experimental data obtain by the lab.

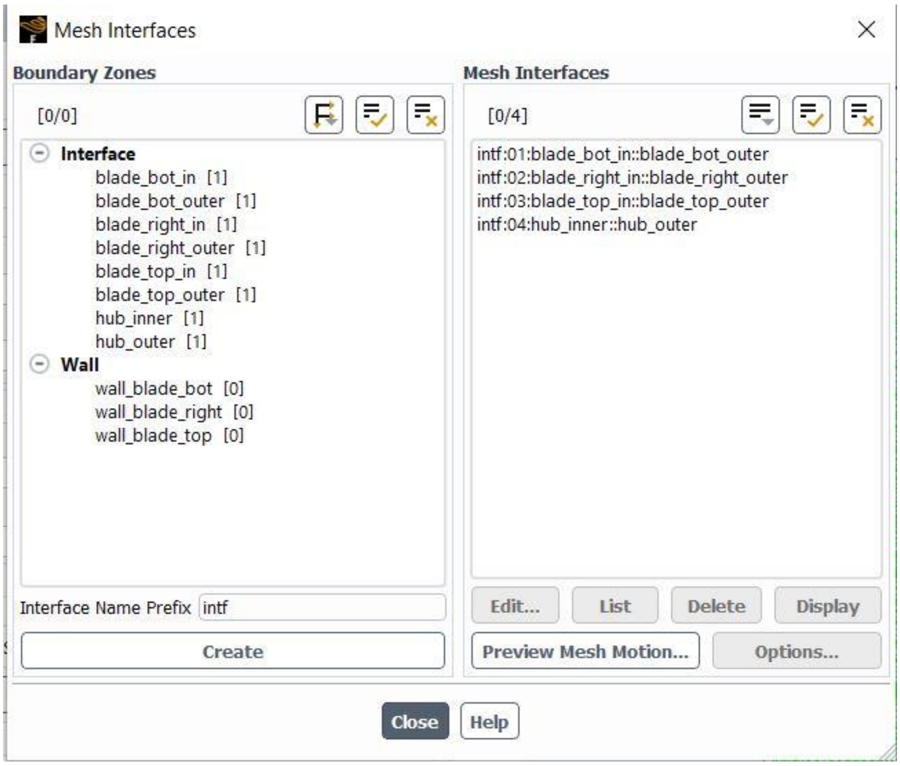
**Frame motion:** In a steady flow simulation where time-averaged quantities are requires to understand steady state for the study frame motion is used. In this a reference frame is used to simulate the motion. Here, the rotation axis is provide with the angular velocity for the frame.

**Mesh motion:** When a time-accurate solution (rather than a time-averaged solution) is desired, you must use the mesh motion to compute the unsteady flow field. The sliding mesh model is the most accurate method for simulating flows in multiple moving reference frames and is the most computationally demanding. As was mentioned, the sliding mesh model is theoretically the most accurate method for simulating rotating flows, and it can correctly describe the whole transient startup. However, it is also the most computationally demanding. This technique, applied to the specific case, results in two cell zones created separately. Each cell zone is bounded by an interface that meets the opposing cell zone. The two cell zones will slide relative to one another along the mesh interface in discrete steps. The AMI (arbitrary mesh interface) operates by projecting one patch's geometry onto the other. In other words, the two sub-domains are geometrically separated but numerically connected by the AMI, ensuring that the values of a generic field are the same on both sides of the interface. The external cell is a simple steady partition in which the mesh does not move during the calculations. Instead, the internal partition, the cylindrical one, is a cell zone that rotates during the simulations. After each step, the internal mesh is rotated at a prescribed angle. The advantage of this technique is that it ensures the best accuracy avoiding, at the same time, mesh deformation.

**Setup:**

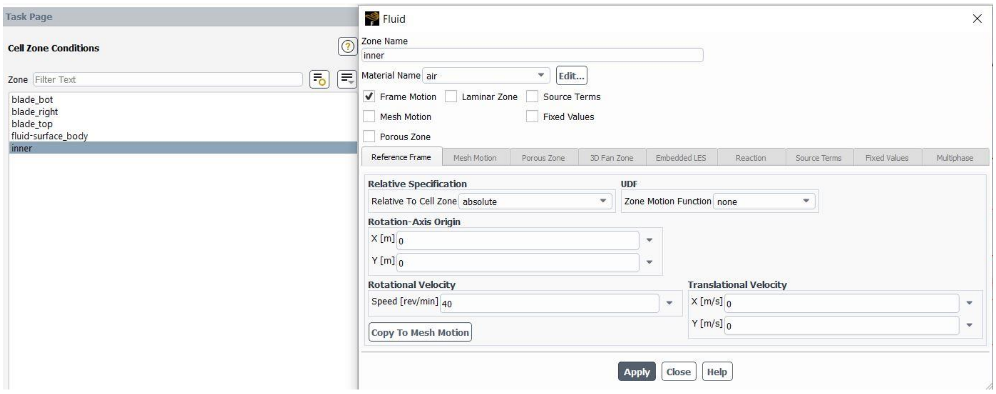
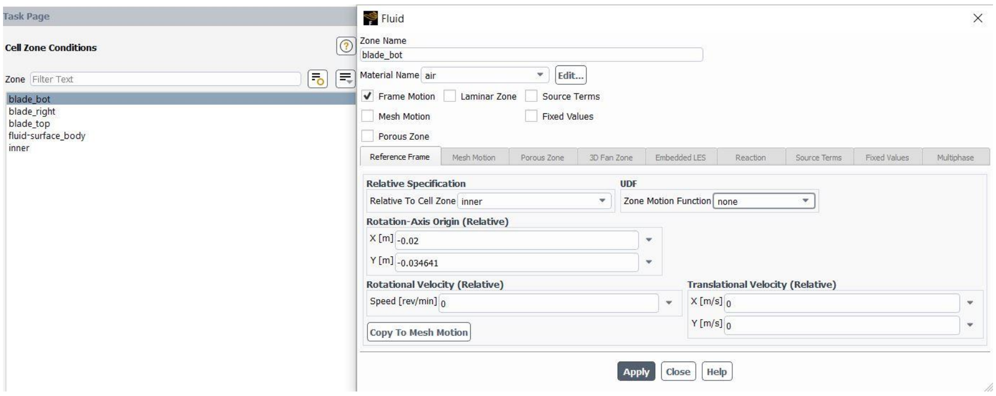
A 2-D analysis where the fluid is air. We use a ***k-epsilon Realizable*** as the viscous model. The mesh consists of different regions for which interfaces need to be created. These interfaces will lead in to creation of cell zones and the boundary conditions will be given accordingly. The snippets of the process are shown below.

1- Create interfaces

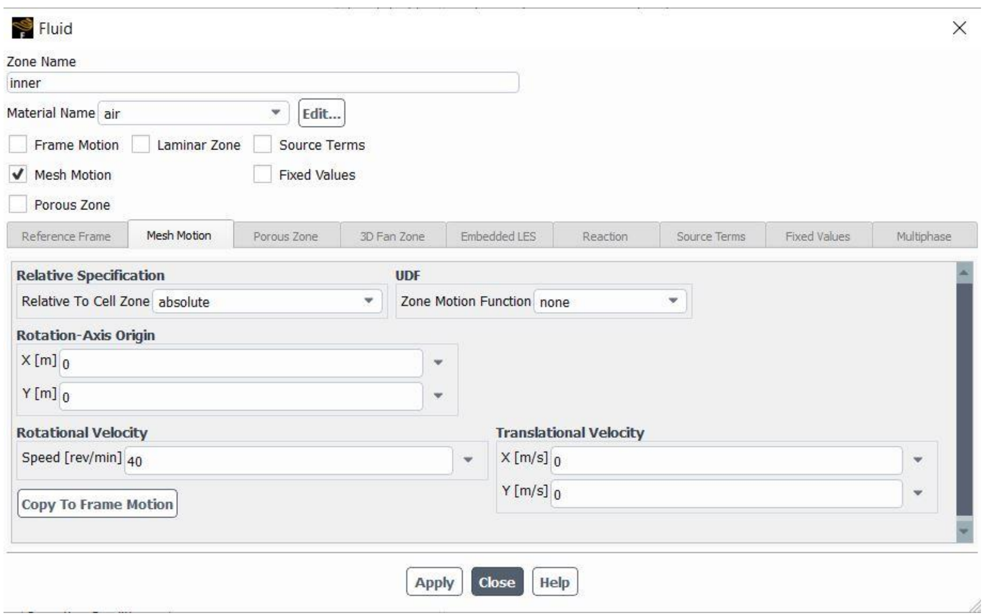


2- Cell zones (setting the frame motion)

a. Frame motion

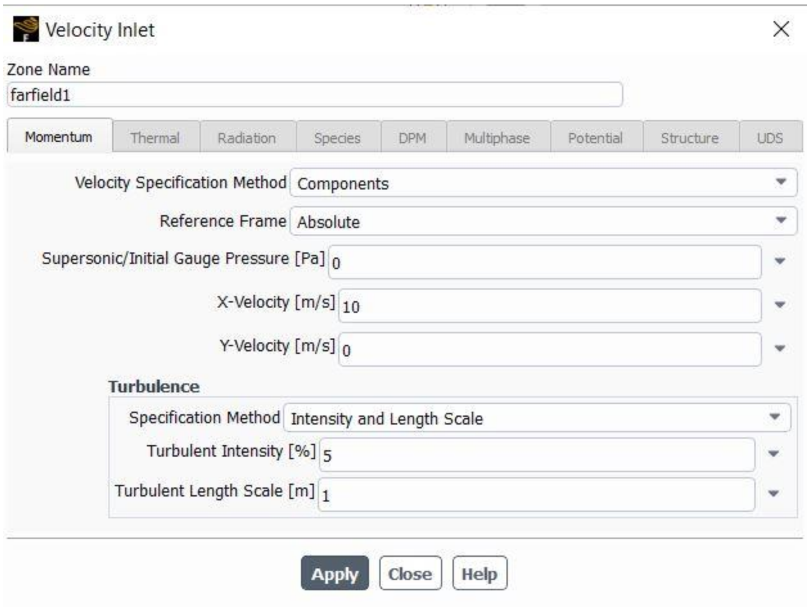
<table>
<tr>
<td>Frame motion for the inner zone and providing it with the angular velocity</td>
<td>
</td>
</tr>
<tr>
<td>Frame motion for the blades keeping the relative specification to Inner zone and providing the axis of rotation.</td>
<td>
</td>
</tr>
</table>

b. Sliding mesh

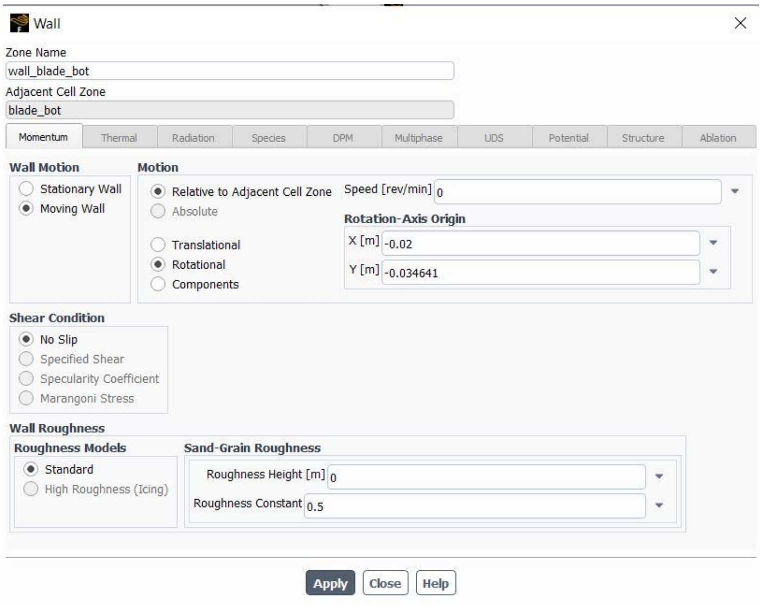
<table>
<tr>
<td>Change Frame Motion to Sliding mesh and copy other values as they are.</td>
<td>
</td>
</tr>
</table>

3- Specifying the Boundary conditions

a. Inlet (*Outlet is a pressure outlet*)



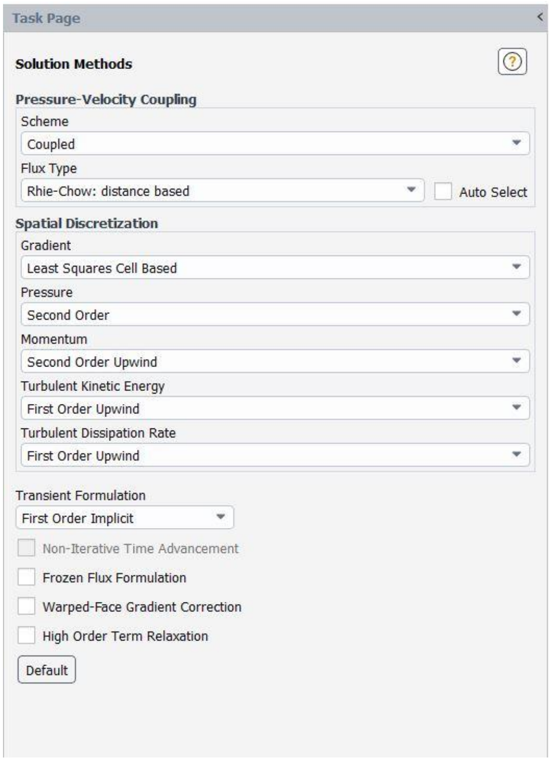

b. Walls: Moving wall that is rotating and the rotation-axis origin are provided to position the centre in the geometry.




For the other walls the axis of rotation is as follows:

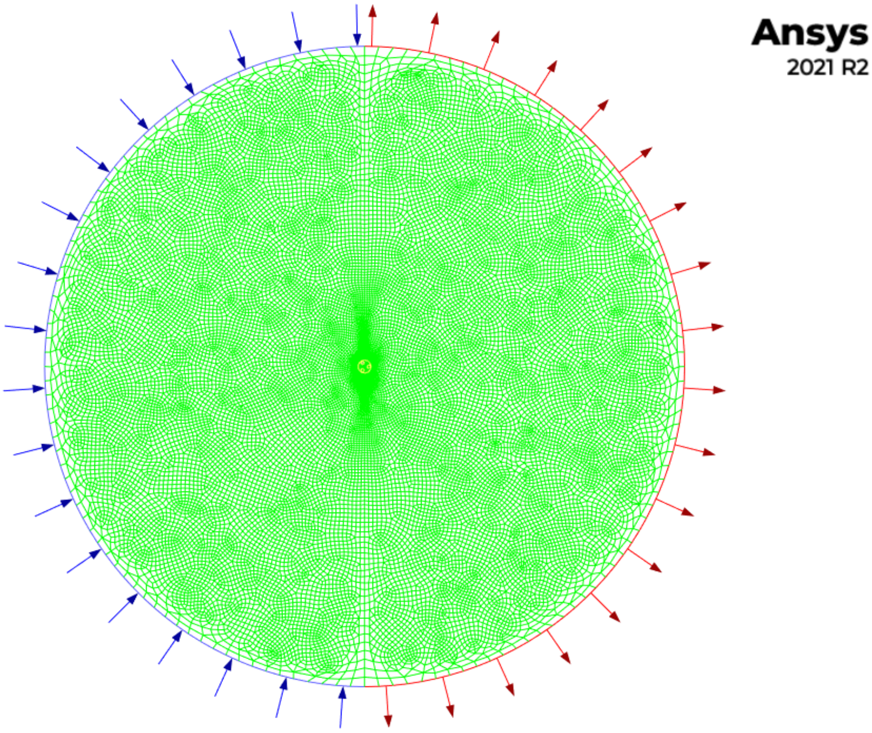
| | |
|---|---|
| Wall_blade_top | -0.02, 0.034641 |
| Wall_blade_right | 0.04, 0 |

**Numerical solution**: *Coupled*



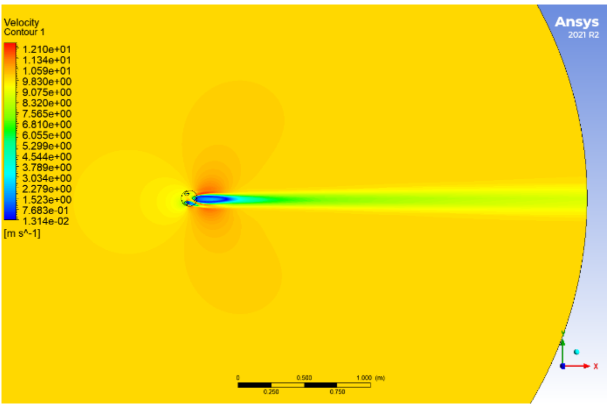
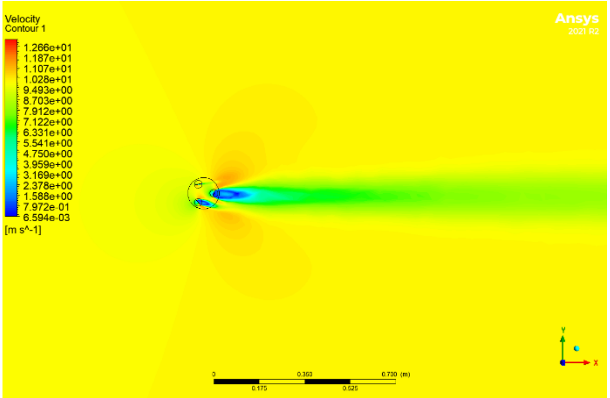
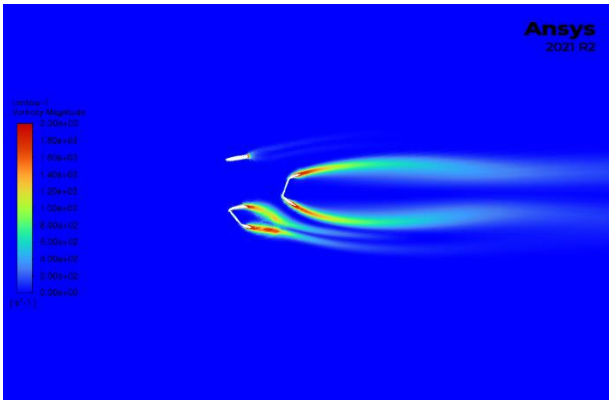
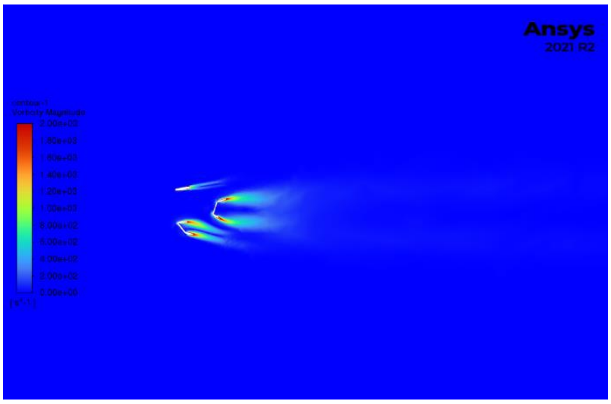

Furthermore, a report definition for Moment is created on the right wing.

After setting up the case the new updated mesh will look as shown:

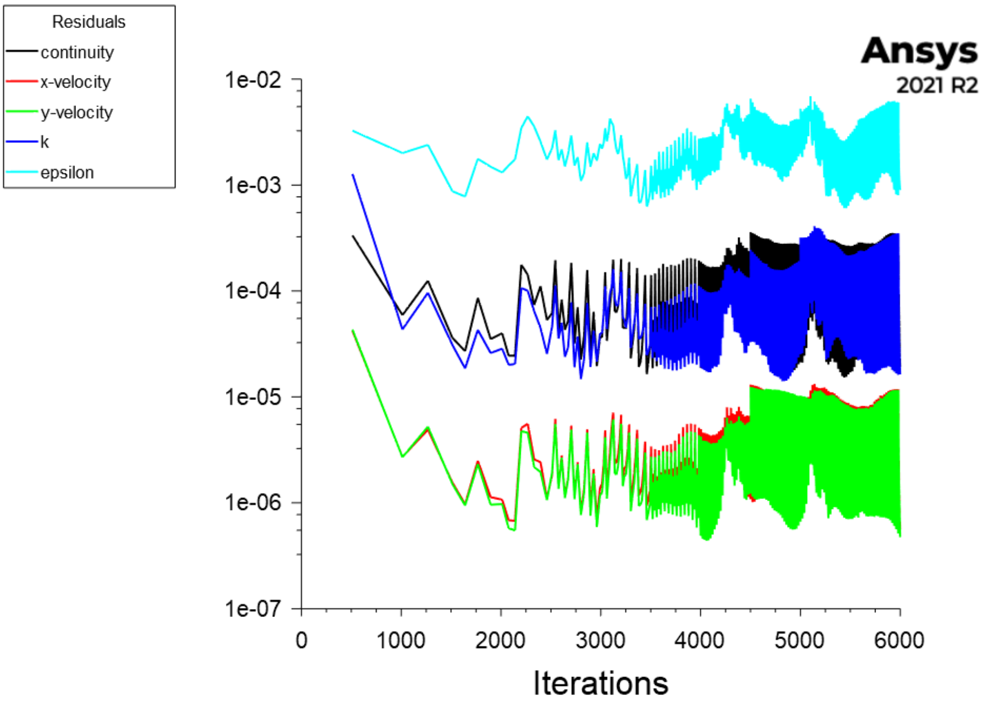
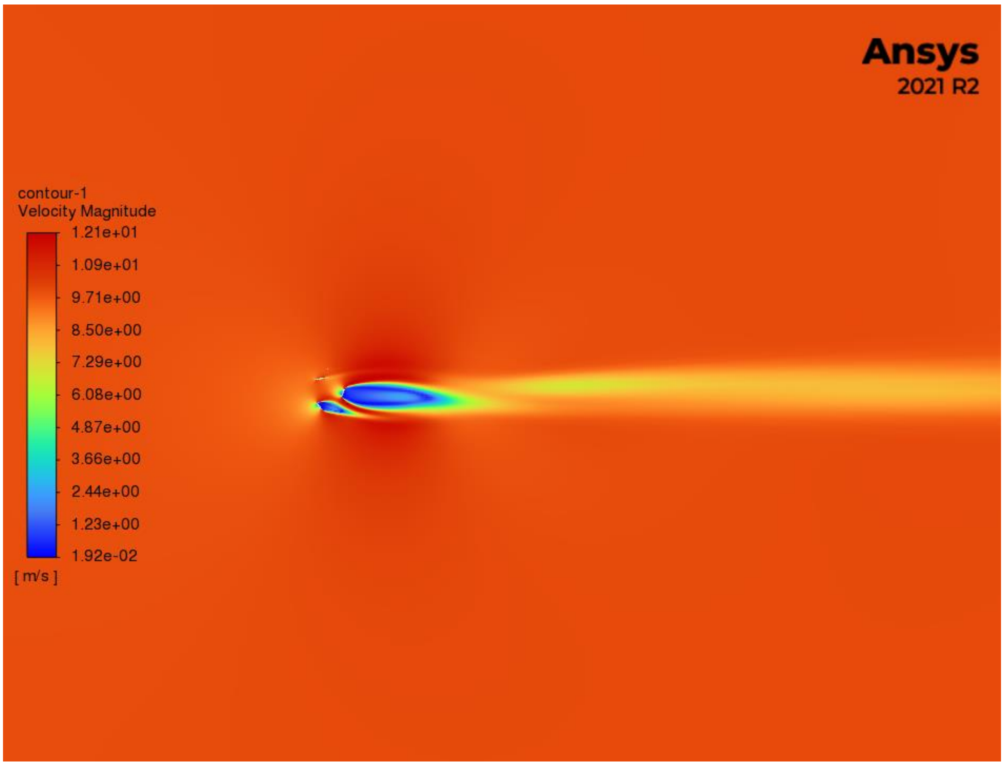


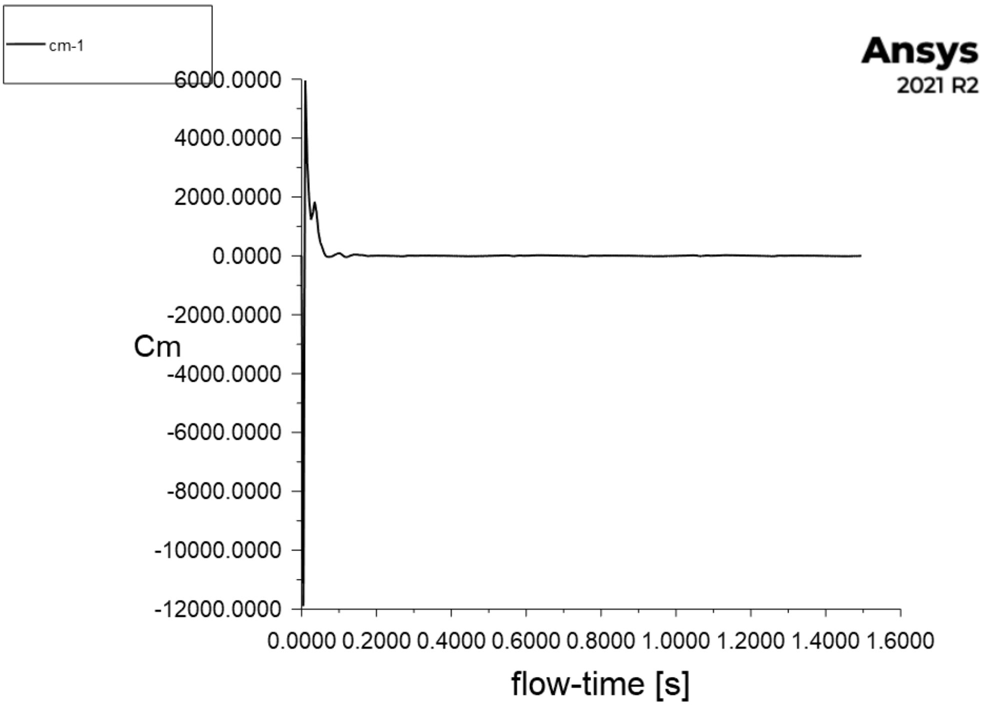
## Result

### *Frame motion*

<table>
<tr><td></td><td>Initial Mesh</td><td>Refined mesh</td></tr>
<tr><td>Velocity</td><td>
</td><td>
</td></tr>
<tr><td>Vorticity Magnitude</td><td>
</td><td>
</td></tr>
</table>

***Mesh Motion***

| | |
|---|---|
| Residuals |   |
| Velocity |   |

| | |
|---|---|
| Moment |  |
| Animation of motion | [Click here]() |

In the Sliding mesh (moving mesh) approach the moment came out to be ***-1.7155***. This means that there was a force acting on the wing. From this, we can conclude that the moment action on the Wind Turbine would spin.

**Discussion:**

***Analytical calculations***

In this study we are interested in evaluating the tip speed ratio (TSR). Which is given as

$$\lambda = \frac{Velocity\ at\ the\ blade\ tip}{Incoming\ wind\ velocity}$$

$$\lambda = \frac{r * \omega}{U}$$

r = distance from the canter to the mid-point of the blade = 0.04m

$\omega$ = Angular velocity 40 RPM 4.1888 rad/s

U = velocity of the flow.

Substituting the values we obtain,

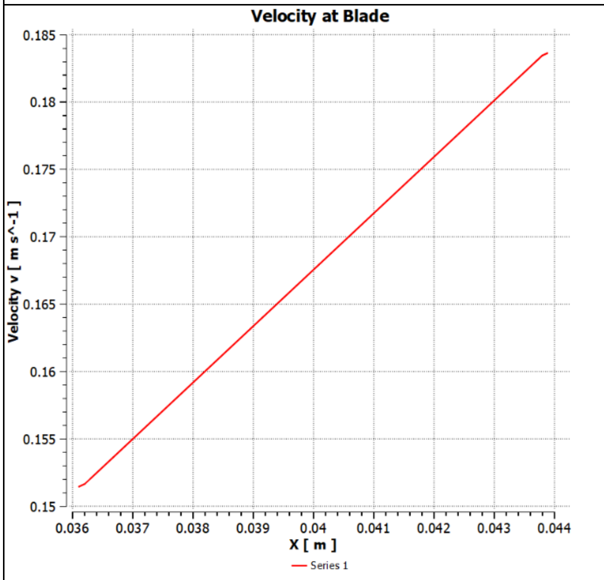
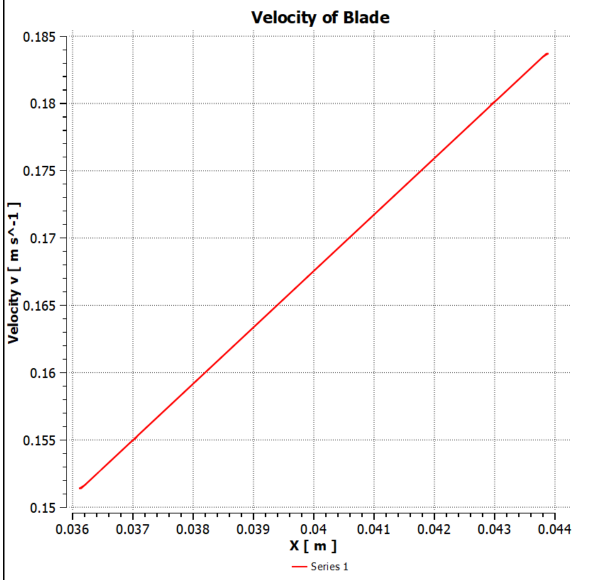
$$\boldsymbol{\lambda = 0.01675}$$

***Numerical results:***

As 2 cases were run for a steady-state (Frame motion) and transient State (Sliding Mesh)

Both of which are compare with the analytical result.

*Note: The transient simulation was run for 1.5 seconds.*

| Frame motion | Sliding Mesh |
| --- | --- |
|  |  |

As it is observed there is no difference between both the approaches the tip velocity is between 0.1676 to 0.17 at the centre i.e., x=0.04m. As the analytical value of TSR is 0.01675 here the TSR = tip speed / flow velocity will lead to

$$TSR = \frac{0.1676}{10}$$

This gives the **TSR = 0.01676** which is not far off from the analytical value of **0.01675**.

***Points to Remember:***

**Verification:** The numerical study is often verified by checking if the values conform to the laws. Eg. In Fluent we can check the mass flow rate through the domain to check if the mass is being conserved. Here that can be done by using the process as follows.

| Net Mass flow rate | Initial Mesh | Refined Mesh |
| --- | --- | --- |
| Inlet and Outlet | 0.00012 | 8.021e-8 |
| Hub | 1.53e-16 | 5.640e-17 |

This gives us a good idea about the mesh and the validity of our study as these values tend to zero which mean a good mass balance is achieved.

This verification can be done in fluent by going through the process:

**Report >> Flux >> *Select the relevant areas (inlet and outlet)***

Another point to remember is that of non-conformal mesh. Since, there are moving parts in the geometry we create separate Zones and they are connected to the whole domain via interfaces.

## Conclusion:

This study helps us understand the concept of Frame motion and Mesh Motion in Fluent. This study of Vertical Axis Wind Turbine gives a good idea about the effects of the incoming air in the turbine which are validated using the Tip speed ratio formulation. Seeing the accuracy of results with the set of meshes (initial and refined) we conclude that the setup is validated and accurate.