**MAE 5020 Wind Blade Fluent Steps**

Geometry:

1. Import STL of wind blade into Geometry
2. Open SpaceClaim
3. Check units – set to mm, etc.
4. Rotate wind blade such that fluid flow direction is along the positive Z-axis direction, blade axis along Y-direction.
5. Select tip face, hide all other faces
6. Click z-axis, create new plane (hit Escape so it stops creating more planes)
7. View from y-axis
8. Create points at leading and trailing edges (carefully) and draw the chord line, it becomes part of the
9. Go back to 3D mode – face is split but this is OK – hide points, highlight the plane, measure angle between xy-plane and chord line
10. Zoom extents 🡪 show all faces, select body, move, anchor origin, rotate about y-axis to get it to neutral.
11. Then rotate it to get the pitch setting right
12. Delete plane and line to keep a single face
13. Move the wind blade 40mm away from the origin in the y-direction to simulate the hub of the wind blade.

BREAK -

1. Use XY plane as a sketch plane, create a 1/3 circle with the reference frame as the center of the circle. The wind blade should be oriented along the x-axis and the edges of the cylinder are 60 degrees from the x-axis in both directions. The circle radius should be 2.5 wind blade lengths.
2. Using pull command, create a solid 1/3 cylinder by extruding the surface 2.5 wind blade lengths for the inlet and 5 wind blade lengths for the outlet. The blade used for the 4021/5020 project has a length of about 300 mm when including the hub.
   1. Select “no merge” to make sure not to merge the wind blade into the extrusion
3. Use the combine function, select the enclosure as the desired shape and the wind blade as the cutter body. Deselect “Keep Cutter”
4. 3 solids should be in the Structure tree; the wind blade, and the enclosure with the wind blade. Suppress wind blade and hide it.
   1. Verify results by hiding a face of the enclosure and confirming that the wind blade is still in the enclosure
5. Create the following named selections: inlet, inlet\_top, outlet, Period 1, Period 2
   1. Periods should be defined so that the blade rotating around positive z has P1 leading
   2. Rename the enclosure under structures as fluid\_domain
   3. Highlight the wind blade body and create a named selection called blade
6. SAVE
   1. Include external files if saving as wbpz

Fluent Mesher:

1. Drag the Fluent with Fluent Meshing tool from the ANSYS toolbox into the ANSYS workspace
2. Drag the geometry from the Geometry tool into the Mesh option Fluent Meshing
3. Launch the Fluent Mesher, enabling double precision and selecting the desired number of cores
4. Display 🡪 Mouse Buttons 🡪 workbench default.
5. Import geometry
6. To use clipping planes: Display all 🡪 insert clipping planes 🡪 limits in z, y, x as desired
7. Add Local Sizing to the blade named selection, use defaults, it shows little sample cells
8. Use defaults, create surface mesh
   1. Curvature & Proximity turned on
   2. CHECK TRAILING EDGE using clipping planes
9. Use fluid region with no voids under Describe Geometry, select no and no for next two boxes
10. Right click on Describe Geometry, select new task, then select set up periodic boundaries
    1. Use default settings (Rotational) and select Period 1 and 2
    2. Check the trailing edge – if it is no longer correct, change “Remesh Asymmetric Mesh Boundaries” from “auto” to “no”
11. Should by default pick the right boundaries
12. Click Update Regions
    1. Should be two regions, blade and fluid domain
    2. Set the blade region to dead (should be fluid by default) (fluent reads it just as a cavity)
    3. Leave fluid\_domain as fluid
13. Add a boundary layer using default values
14. Create Volume mesh using default values – check the units to match what I was using in Spaceclaim
15. Once mesh is complete, check the number of nodes/cells using report and check mesh under the mesh tab
    1. Check curvature and proximity on surface mesh if we are having overlapping errors
16. SAVE
17. Check mesh and click Switch to Solution if Mesh is correct

Model Setup

1. Under Setup, Models, select Viscous
   1. Pick k-omega for model and GEKO for k-omega model
   2. For this test, use default air values (Materials -> Fluid)
   3. Should be 1.225 for density and 1.7894e-5 for viscosity
2. Under Cell Zone Conditions > Fluid > fluid\_domain
   1. Select frame motion, set Rotation-axis origin to {0,0,0} and rotation-axis direction to {0,0,1}. Set rotation velocity to 98 rad/s (tip speed ratio of about 5)
3. Under boundary conditions
   1. Select inlet, change magnitudes to components and set the Z-velocity component to 6 m/s
   2. Repeat 26a for inlet\_top
   3. For outlet, ensure the gauge pressure is 0 and the operating conditions is 101325 Pa
   4. Check that the periodic boundary conditions are set.

Numerical Solution:

1. Under Solutions (top bar), select definitions, new, surface, integral
   1. Ensure field variables are set to pressure, static pressure
   2. Select surface to be blade
   3. Under Create, select Report File and Report Plot
2. Expand Monitors, double click on Residual
   1. Keep absolute criteria at 1e-3
3. Under Initialization, select Standard Initialization
   1. Compute From > inlet, double check values (Z-velocity should be 6 m/s)
   2. Click Initialize
4. Click on Run Calculation, set number of iterations, then run calculation

Postprocessing:

1. Pressure contour on the blade
   1. Countour 🡪 name 🡪 pressure 🡪 global 🡪 100 contours
2. Pressure contour & Velocity Vectors on section y = .1
   1. Location 🡪 create plane 🡪 xz plane 🡪 pick y = .1
   2. Contour 🡪 location plane 1 🡪 variable pressure 🡪 range local 🡪 100 contours
3. Power
   1. Calculators 🡪 location blade wall 🡪 axis global Z 🡪 fluid air 🡪 calculate
   2. Multiply by angular speed to get power output.