

Homework No.4 - Model Wing Simulation Example from courses.ansys.com

Osamu Katagiri-Tanaka : A01212611

September 7, 2020

1 Problem Statement

Ansys' simulation example is to analyse low-speed air flow corresponding to take-off and landing conditions over a mock-up aircraft wing in a wind tunnel test set-up. Along with analyses of lift and drag generated by the wing. The required Mesh file and associated case & data files were downloaded from [Ansys' website](#).

The model consists of a mock-up wing mounted at 5° angle of attack inside a wind tunnel for testing with the following characteristics:

- Test section dimensions: $16.96\text{m} \times 6.8\text{m} \times 6.11\text{m}$ ($L \times W \times H$)
- Wing span: 1.8m
- Wing reference chord: 0.665m
- Wing platform area: 1.0942m^2

The domain consists of two fluid region filled with air, named as “Wind tunnel test section” and “Wing nearfield”. See Figure 1. The inlet velocity is $68 \frac{\text{m}}{\text{s}}$, which corresponds to Mach 0.2. Effects of air compressibility are negligible when the mach number is less than 0.3. Therefore, the air is at standard sea level conditions and assumed to be incompressible $\left(\rho_{air} = 1.225 \frac{\text{Kg}}{\text{m}^3} \text{ and } \mu_{air} = 1.7894 \times 10^{-5} \frac{\text{Kg}}{\text{m s}} \right)$.

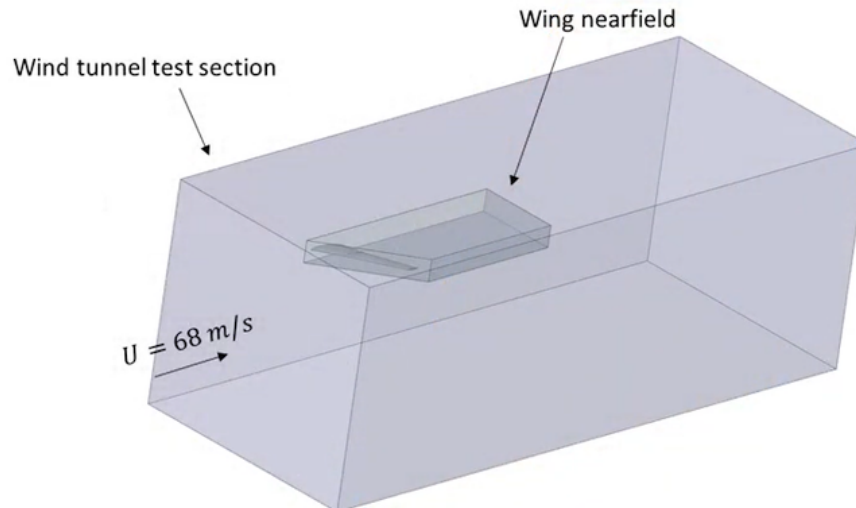


Figure 1: Problem Statement - Fluid Regions : adapted from [1]

The Reynolds number based on the reference chord is $Re = 3.0 \times 10^6$, implying that the flow over the wing is fully turbulent. The air entering the tunnel is smooth with very little turbulence. As shown in Figure 2, the boundary conditions are as follows:

- inlet: velocity inlet
- outlet: pressure outlet
- side and mounting walls: slip walls
- wing: no-slip wall

A steady state simulation is to be done using the pressure-based coupled solver in Ansys Fluent.

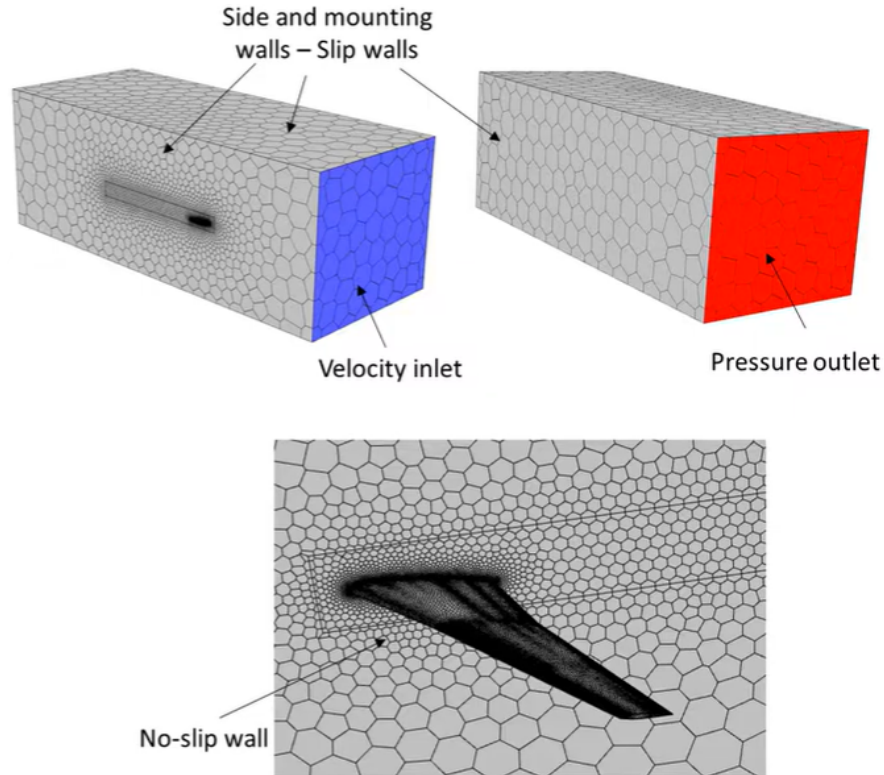
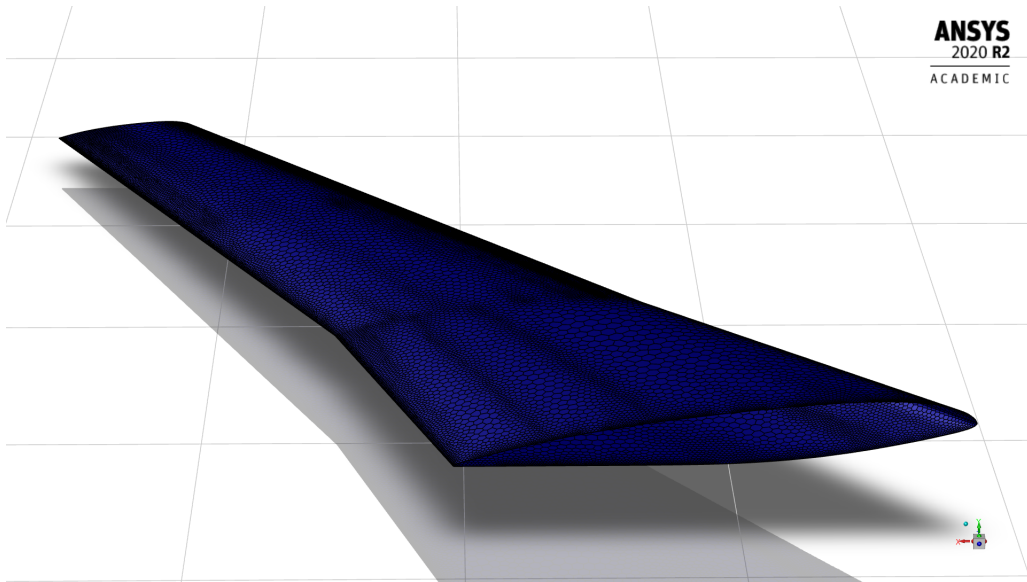


Figure 2: Problem Statement - Boundary Conditions : adapted from [1]

2 Simulation & Results

First step is to read the mesh file, also known as the “wing model” by doing: FILE → READ → MESH



Step two is to set-up the physics by opening the PHYSICS tab. Under the REFERENCE VALUES... menu, the wing properties (listed in the Problem Statement section) are specified.

Task Page

Reference Values

Compute from

Reference Values

Area (m2) 1.0942

Density (kg/m3) 1.225

Enthalpy (j/kg) 0

Length (m) 0.665

Pressure (pascal) 101325

Temperature (k) 288.15

Velocity (m/s) 68

Viscosity (kg/m-s) 1.7894e-05

Ratio of Specific Heats 1.4

Yplus for Heat Tran. Coef. 300

Reference Zone

fluid-near-wing

Under the ZONES menu, the boundary conditions are specified as depicted in the Problem Statement section.

Task Page

Boundary Conditions

Zone Filter Text

inlet

interior-fluid-near-wing

interior-fluid-tunnel

interior-fluid-zones-boundaries

mounting_wall-near-wing

mounting_wall-tunnel

outlet

tunnel-walls

wing

Phase Type ID

mixture velocity-inlet 50

Edit... Copy... Profiles...

Parameters... Operating Conditions...

Display Mesh... Periodic Conditions...

☐ Highlight Zone

Velocity Inlet

Zone Name

inlet

Momentum Thermal Radiation Species DPM Multiphase Potential UDS

Velocity Specification Method Magnitude, Normal to Boundary

Reference Frame Absolute

Velocity Magnitude (m/s) 68

Supersonic/Initial Gauge Pressure (pascal) 0

Turbulence

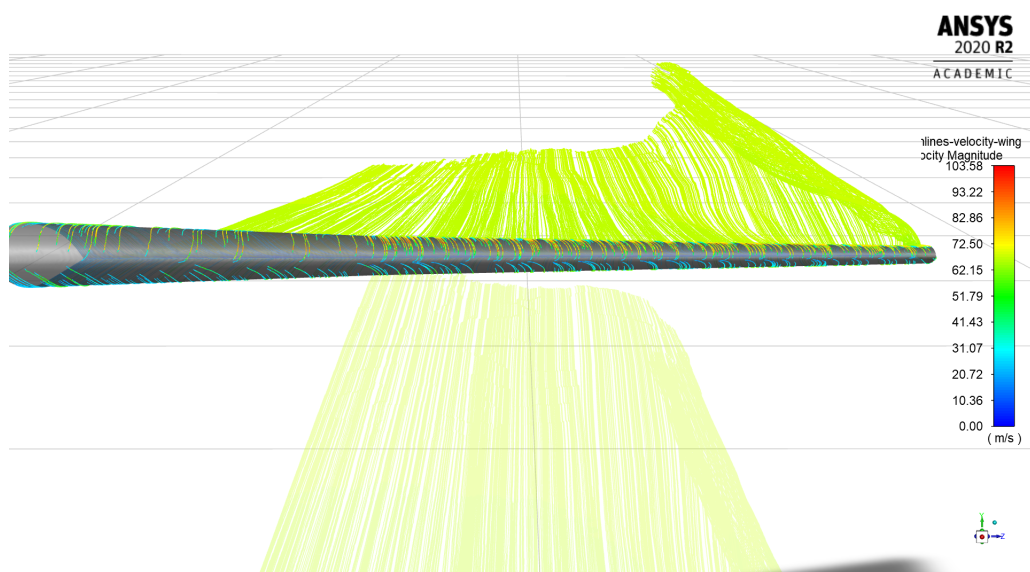
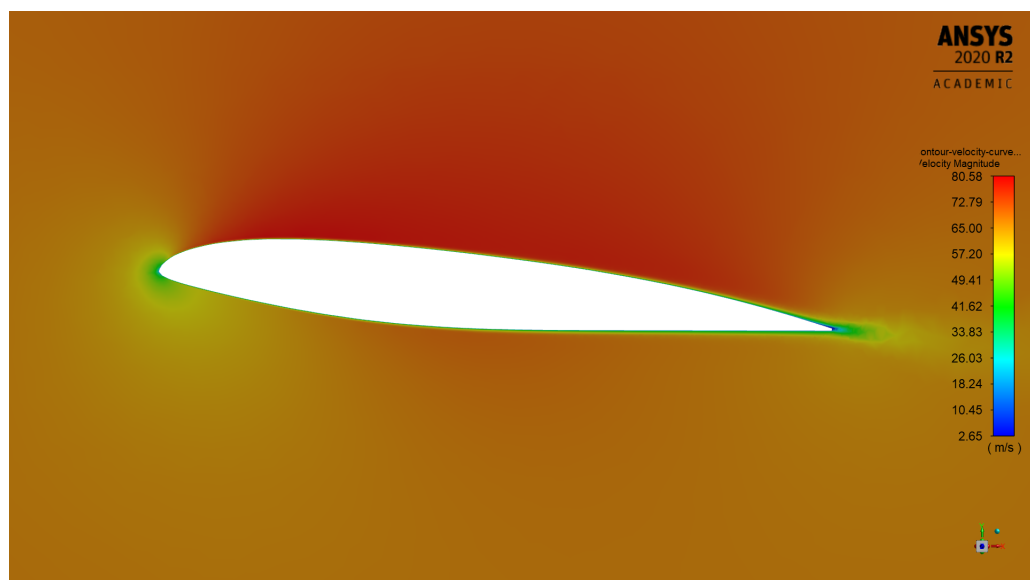
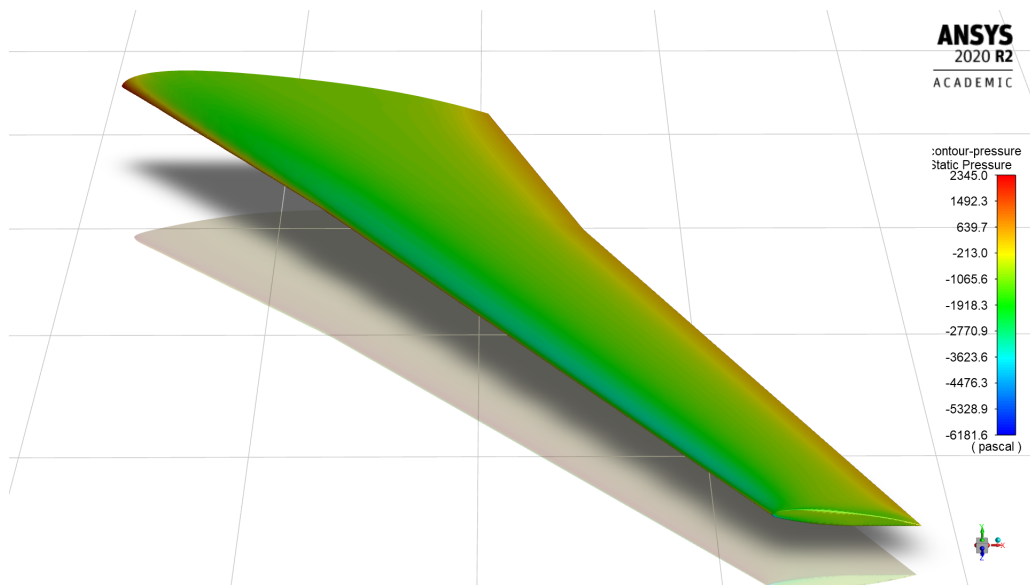
Specification Method Intensity and Viscosity Ratio

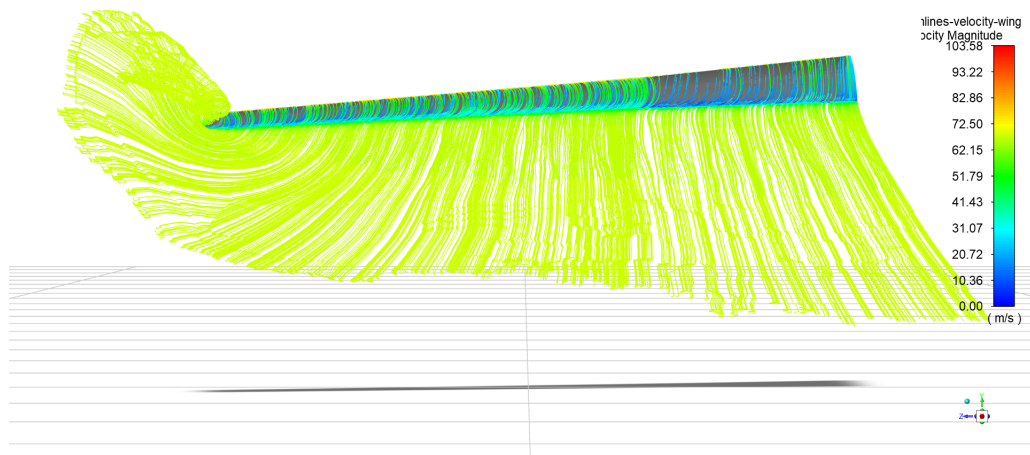
Turbulent Intensity (%) 1

Turbulent Viscosity Ratio 2

Apply Close Help

Once the model is initialized and solved, the results can be visualized as *contours* and *path-lines*





References

- [1] *Model Wing - Simulation Example - ANSYS Innovation Courses*. [Online]. Available: <https://courses.ansys.com/index.php/courses/dimensional-analysis-and-similarity/lessons/simulation-examples-homework-and-quizzes-5/topic/model-wing-simulation-example/> (visited on 09/07/2020).