

Summary: CFD Calculation Domain for a Scaled Model of Boeing 737

This study focuses on the computational fluid dynamics (CFD) analysis of airflow around a model of the Boeing 737. The Boeing 737 is a legendary aircraft known for its efficiency and reliability in commercial aviation. The specific model examined features a wingspan of 35.8 meters and a taper ratio of 0.218, which contributes to its aerodynamic performance. The aircraft typically operates at a cruising altitude of 11,000 meters, flying at a speed of 842 kilometers per hour. During takeoff, it requires a minimum speed of 250 kilometers per hour and a runway length of 2300 meters to achieve lift-off successfully.

In the context of CFD simulations, calculating the Reynolds number is essential to characterize the flow regime experienced during various flight phases. For the takeoff phase, the calculated Reynolds number is 18,782,789.2677, indicating turbulent flow conditions. In contrast, during cruise, the Reynolds number increases to 23,658,907.410, reinforcing the need to analyze flow behavior under both scenarios.

To perform these calculations, specific aerodynamic parameters are utilized. The root chord length of the wing is 5.71 meters, while the wingtip chord measures 1.25 meters. The mean aerodynamic chord, which plays a crucial role in determining the lift characteristics of the wing, is calculated to be 3.95 meters. At an altitude of 11,000 meters, the air density is approximately 0.3639 kg/m^3 , and the temperature is 216.65 Kelvin (equivalent to -56.5 degrees Celsius). In comparison, at sea level, the air density is 1.225 kg/m^3 , with a temperature of 293.15 Kelvin (20 degrees Celsius). The dynamic viscosity values are also critical; they are $1.421 \times 10^{-5} \text{ m}^2/\text{s}$ at 11 kilometers altitude and $1.789 \times 10^{-5} \text{ m}^2/\text{s}$ at runway level. These factors contribute to identifying the turbulent regime within the flow around the aircraft.

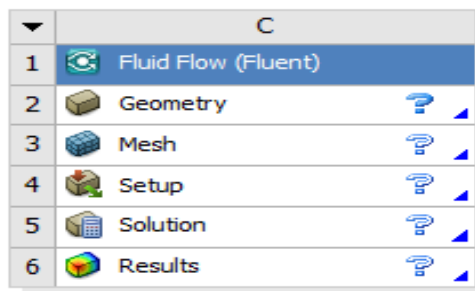
The methodology for this study includes importing the provided Boeing model into ANSYS Design Modeler, a software tool used for creating and analyzing CAD models. Once imported, fluid flow blocks are created to facilitate the simulation of airflow around the aircraft.

A key aspect of the analysis involves establishing a parameterized angle of incidence for the wing, which encompasses the wings, fuselage, and tail structures. This angle, referred to as alpha, is set to 3 degrees with the nose of the aircraft tilted upwards, simulating the conditions during takeoff and initial climb.

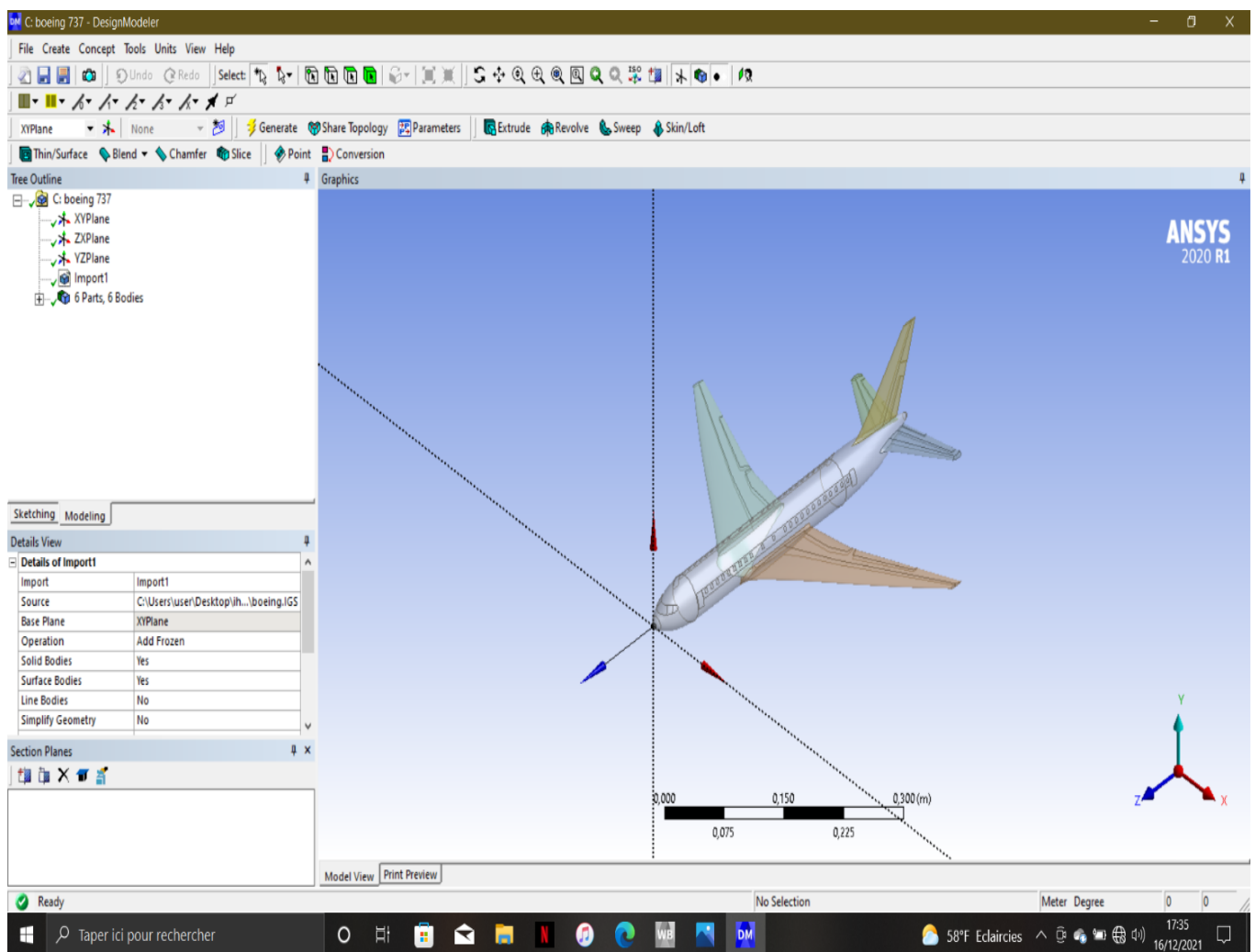
To accurately model the flow environment around the aircraft, a cubic computational domain is defined. This domain encompasses the entire model, ensuring that the simulation captures the airflow characteristics. The upstream distance from the aircraft is set to 6 meters, while the downstream distance is 12 meters. The vertical clearance above and below the aircraft is maintained at 3 meters, allowing for an unobstructed airflow profile around the fuselage and wings.

Within this setup, six named selections are created to facilitate the boundary conditions of the CFD simulation. These selections include: inlet (the area where airflow enters the computational domain), outlet (where the airflow exits), wing (the main lifting surface), fuselage (the body of the aircraft), horizontal tail (the stabilizing surface at the rear), and vertical tail (the rudder structure).

1)&2)



boeing 737



4) & 5)

