

# ANSYS AIM - Compressible Flow over a Wing-Body Junction

Author(s): Sebastian Vecchi, ANSYS Inc.

## [Problem Specification](#)

- [1. Start-Up](#)
- [2. Geometry](#)
- [3. Mesh](#)
- [4. Physics Setup](#)
- [5. Solution/Results](#)

## Problem Specification

The airplane body below is subjected to 600 miles per hour winds at 0 degrees to simulate it traveling mid flight. In an effort to study the flow over the wing-body junction, the velocity vectors and streamlines will be calculated.

[Go to Step 1: Start-Up](#)

[Go to all ANSYS AIM Learning Modules](#)