Turbulent Flow About M6 Wing

SmartNumerics Simulation Solutions Inc.

September 18, 2021

CFD codes are validated in part by comparing computed airfoil forces, pressure profiles, and friction profiles to data collected using airfoil or wing models inserted in wind tunnels. The angle of attack (AOA) used in the simulation must be made smaller than the experimental angle of attack to adjust for the effect of the wind tunnel walls. The need to adjust AOA is mostly due to the effect of the top and bottom walls of the tunnel. The tunnel side walls also effect the measurement due to the presence the side-wall boundary layers. These boundary layers reduce flow near the walls and increase it outside of the boundary layers.

The degree of agreement between the simulation with adjusted AOA and Mach number and the experiment, is used as a measure of the suitability of the turbulence model being used. Different turbulence models require slightly different values of adjusted AOA to obtain the best match to experiment. The most suitable turbulence model will produce the best match to lift, drag, Cp curves, C_f curves, and shock location.

The M6 validation test is based on the experimental measurements of Schmitt and Charpin [1]. The wind tunnel model had a half span of 1.1963 m and a mean aerodynamic chord of 0.64607. The root chord was 0.8059 m and the wing area was 0.7532 m^2. The turbulent intensity in the wind tunnel was 0.2%. The angle of attack was 3.06 degrees. The Reynolds number based on a mean aerodynamic chord of 0.64607 meters was $11.72x10^6$.

This validation test case uses the PLOT3D grid file m6wing.x.fmt from the NASA web page https://www.grc.nasa.gov/WWW/wind/valid/m6wing/m6wing01/m6wing01.html. The grid and test case are described on the website maintained by John W. Slater. The grid consists of 4 blocks and contains 294,912 cells.

The downloaded grid had a half span of about 1.0165 m and a root chord of 0.6737 m. Both values were somewhat smaller than the values given for the wind tunnel model. To get the correct root chord, the coordinates were multiplied by 1.1962. Boundary conditions and block connections were imposed on the grid and it was exported in CGNS format for subsequent use.

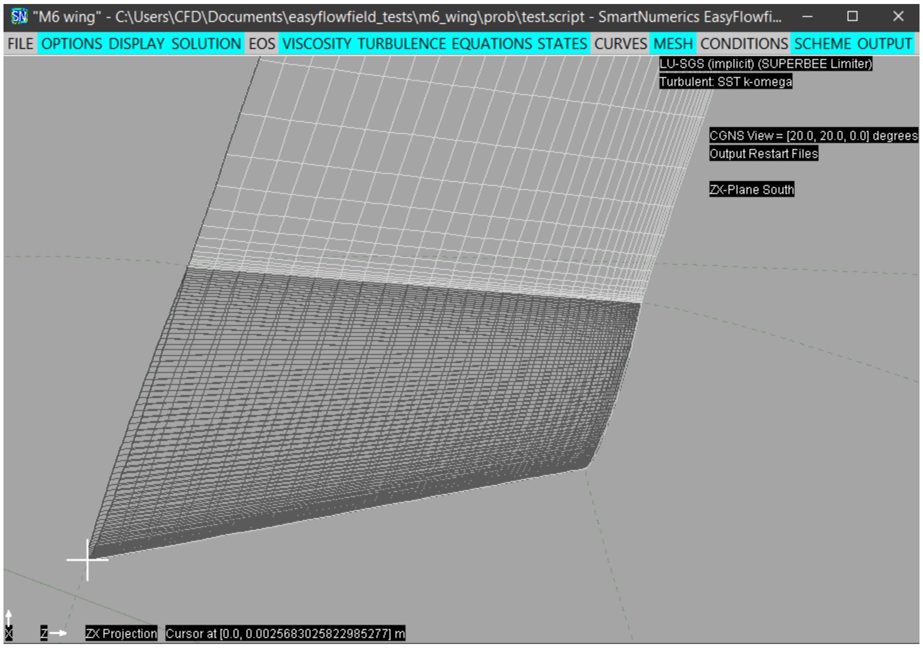

Fig. 1: Gridlines on wing and in wake region for M6 wing.

Figure 1 displays gridlines on the airfoil and in the wake region, using a viewing angle to display both sides of the wing surface. Without this viewing angle, the x-coordinate is seen to be chordwise and the z-coordinate is spanwise.

An (implicit) LU-SGS solver was used with an initial CFL number of 1 which was gradually increased to 20 after a delay of 100 cycles. HLLC fluxes were used with the SUPERBEE limiter and a compression parameter of 1.49. The SST turbulence model of Menter [2] was used with an automatic (adaptive) wall function. The far-field turbulence intensity was set to 0.2% and the far-field eddy viscosity was set to one tenth of the laminar viscosity. The far-field Mach number was 0.8359, the pressure was 80506 Pascals, the temperature was 255.56°K), and the angle of attack was 3.06 degrees. The laminar viscosity was computed from temperature using the Sutherland formula.

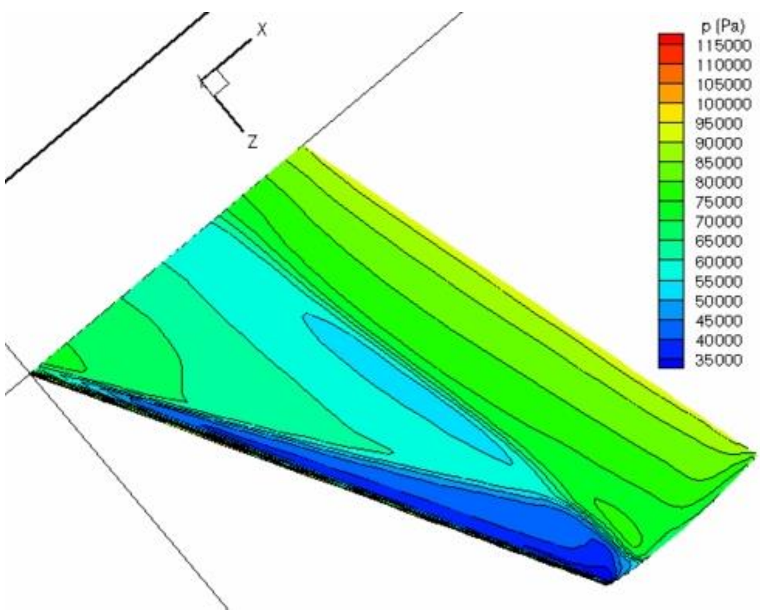

Fig. 2a: Pressure contours on M6 using solution of full grid.

Figure 2a displays pressure contours on the top of the M6 wing at 4000 cycles on the full grid. The inboard portion of the wing exhibits two shocks which merge near 80% of half span.

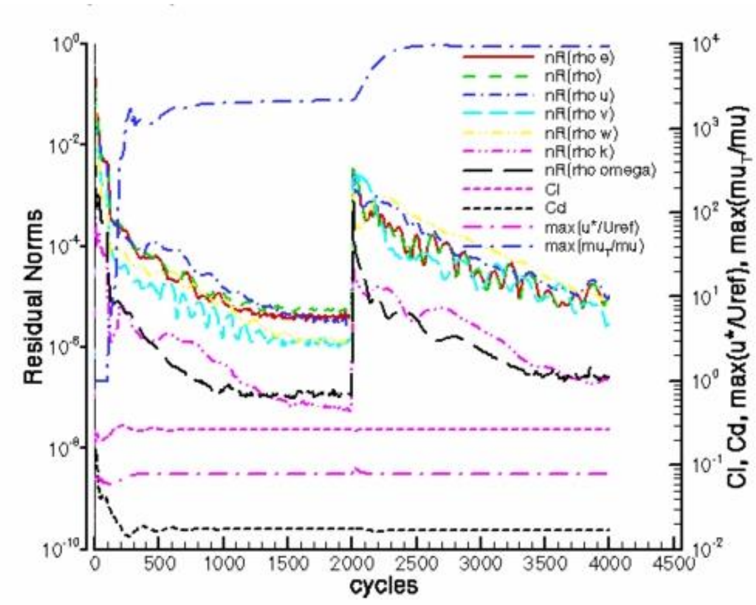

Fig. 2c: Convergence history for turbulent flow about m6 wing

An initial simulation was performed for 2000 cycles on a grid with one eighth the number cells. The solution was transferred to the full grid and extended for an additional 2000 cycles. Figure 2c displays the convergence history on the two grids. The maximum value of eddy viscosity divided by the far-field molecular viscosity is approximately constant after 1000 cycles on either grid as is the maximum friction velocity (u*) divided by the far-field velocity. The variation in lift (Cl) or drag (Cd) is negligible after 1000 cycles on either grid. The values of y^+ based on cell thickness next to the wall varied from 35 to 3.5 on the main grid and from 75 to 16 on the coarse grid. The coefficients of lift and drag on the full grid are respectively 0.266 and 0.0170 on the main grid and 0.263 and 0.0173 on the coarse grid. These values are in good agreement with the values of 0.268 and 0.0171 given by Osusky [3] who used a 128-block grid with 15.1 million nodes (average y^+ of 0.4 next to the wall).

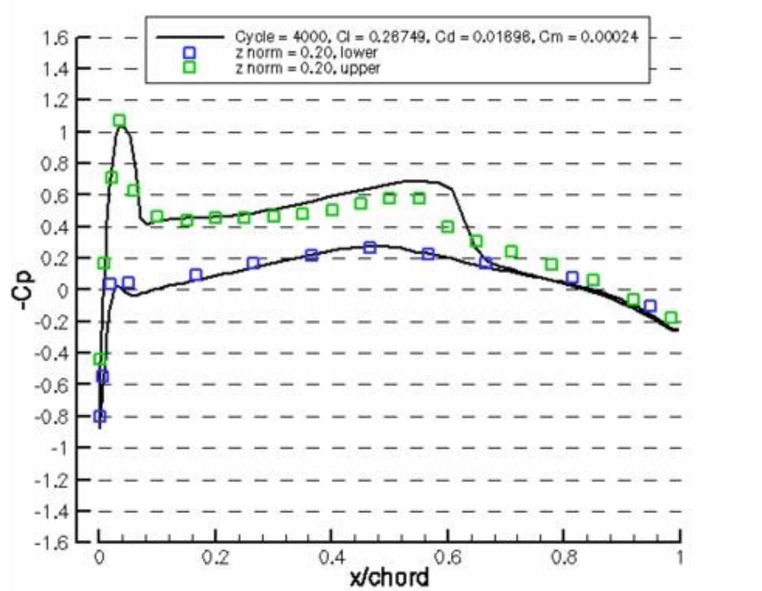

Fig. 2d: Cp from simulation at 20% of chord compared to experimental values.

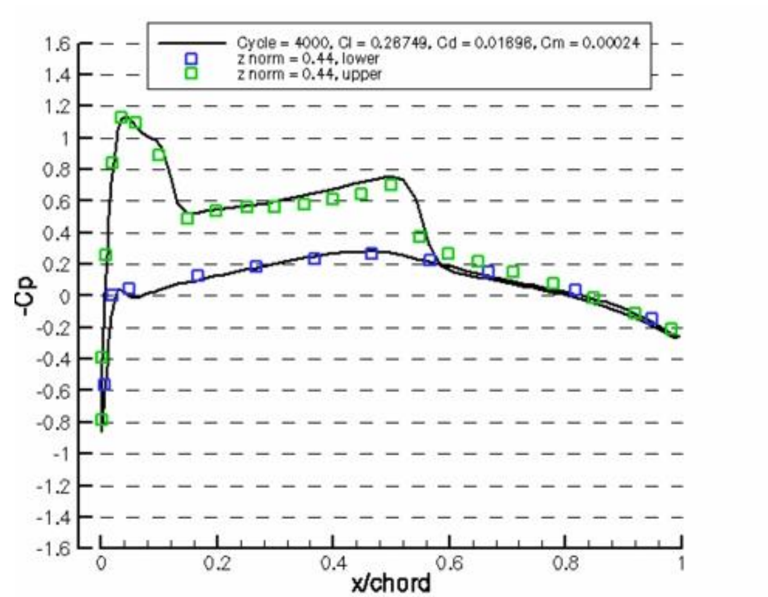

Fig. 2e Cp from simulation at 44% of chord compared to experimental values.

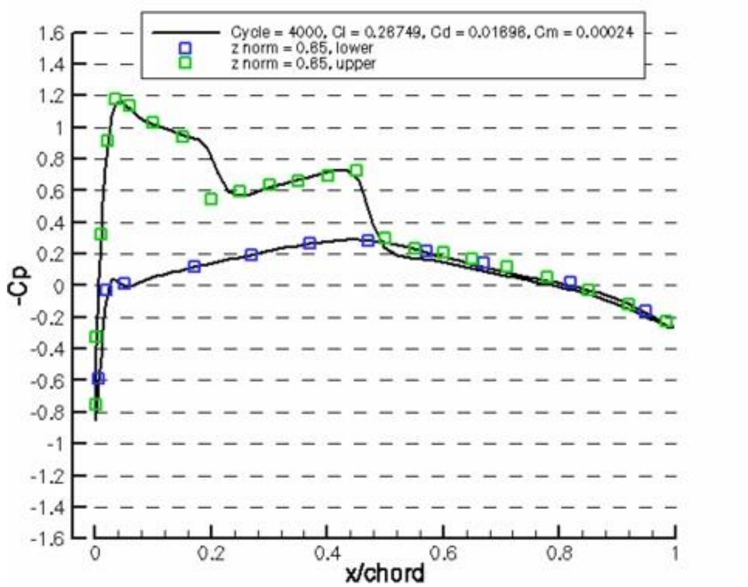

Fig. 2f: Cp from simulation at 65% of chord compared to experimental values.

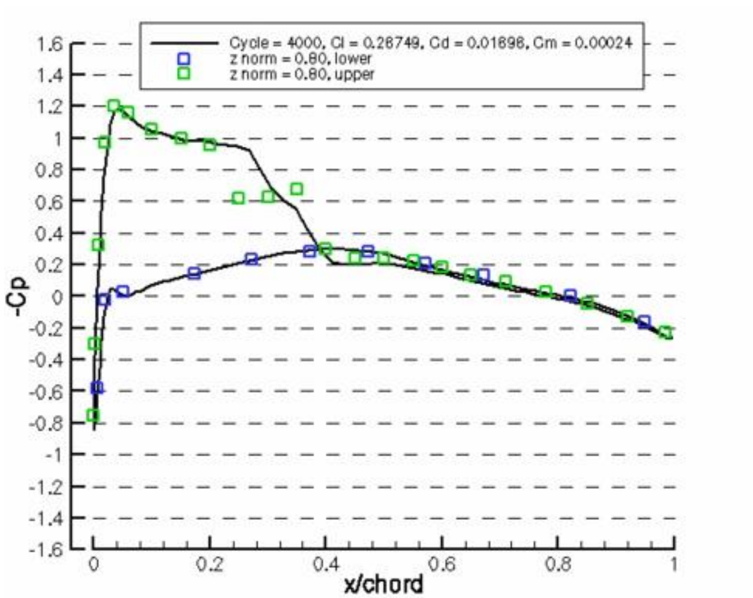

Fig. 2g: Cp from simulation at 80% of chord compared to experimental values.

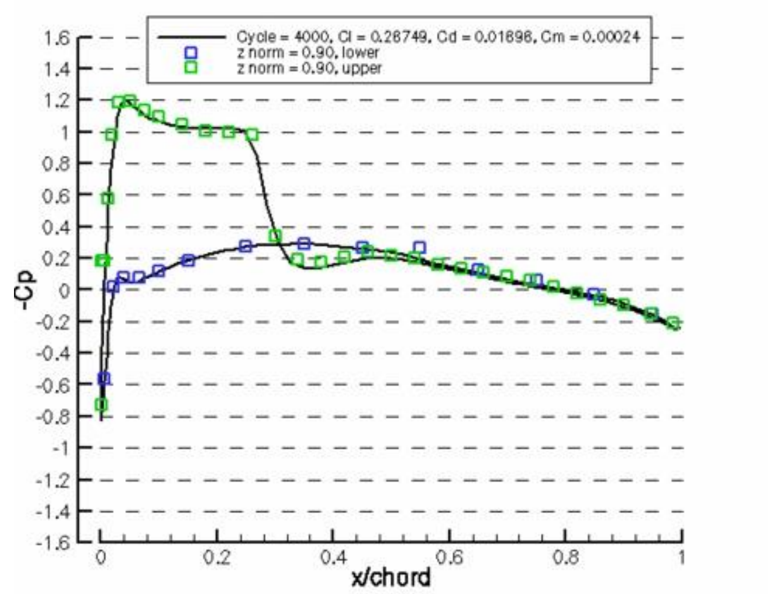

Fig. 2h: Cp from simulation at 90% of chord compared to experimental values.

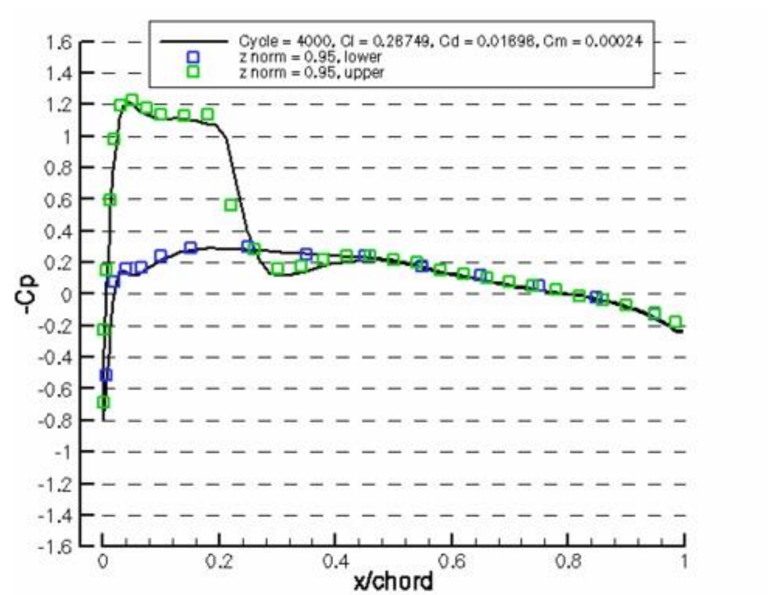

Fig. 2i: Cp from simulation at 95% of chord compared to experimental values.

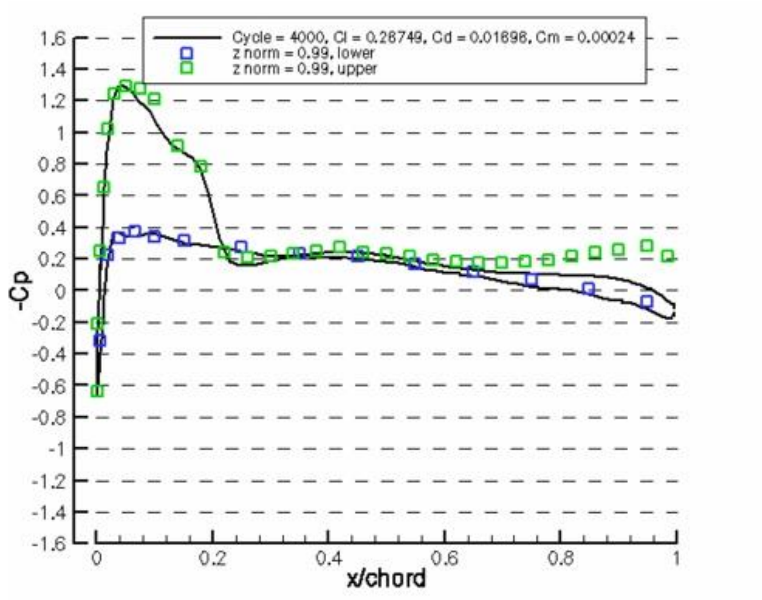

Fig. 2j: Cp from simulation at 99% of chord compared to experimental values.

Figures 2d through 2j display C_p with reversed sign from the main-grid simulation and from experimental measurements at various spanwise locations. The pressure coefficient from the simulation is computed using

$$c_p = \frac{p_w - p_\infty}{\frac{1}{2}\rho_\infty U_\infty^2}$$

where p_w is the local pressure at the wall, p_∞ is the far-field pressure, ρ_∞ is the far-field density, and U_∞ is the far-field velocity. This is plotted versus x divided by the local chord. To match the offset used for the experimental values, the value of x from the simulation is adjusted so that the leading edge is at zero.

The spanwise plots of Cp are a good match to experiment except near the wing root, near the point of meeting of the two shocks at about 80% of half span, and at 99% of half span. The results of Osusky [3] and many other authors using a variety of turbulence models show similar departures from the experimental points. Osusky [3] obtained similar results on a on a 35-million node grid which indicates that these differences are not caused by lack of resolution but are produced by other effects such as the influence of the wind tunnel walls. In addition, the symmetry condition used at the wing root by the simulation does not quite match the experimental use of a boundary layer diverter in the wind tunnel.

References

[1] Schmitt, V., Charpin, P. C., "Pressure distributions on the ONERA-M6-Wing at transonic Mach numbers," in Experimental Data Base for Computer Program Assessment, AGARD Report AR 138, 1979.

[2] Menter, F. R., Improved two-equation k-ω Turbulence Models for Aerodynamic Flows, NASA TM-103975, 1992.

[3] Osusky, M., A Parallel Newton-Krylov-Schur Algorithm for the Reynolds-Averaged Navier-Stokes Equations, PhD Thesis, Department of Aerospace Science and Engineering, University of Toronto, 2013.

[4] Wilcox, D. C., Turbulence Modeling for CFD, 3rd ed., DCW Industries Inc., 2006.