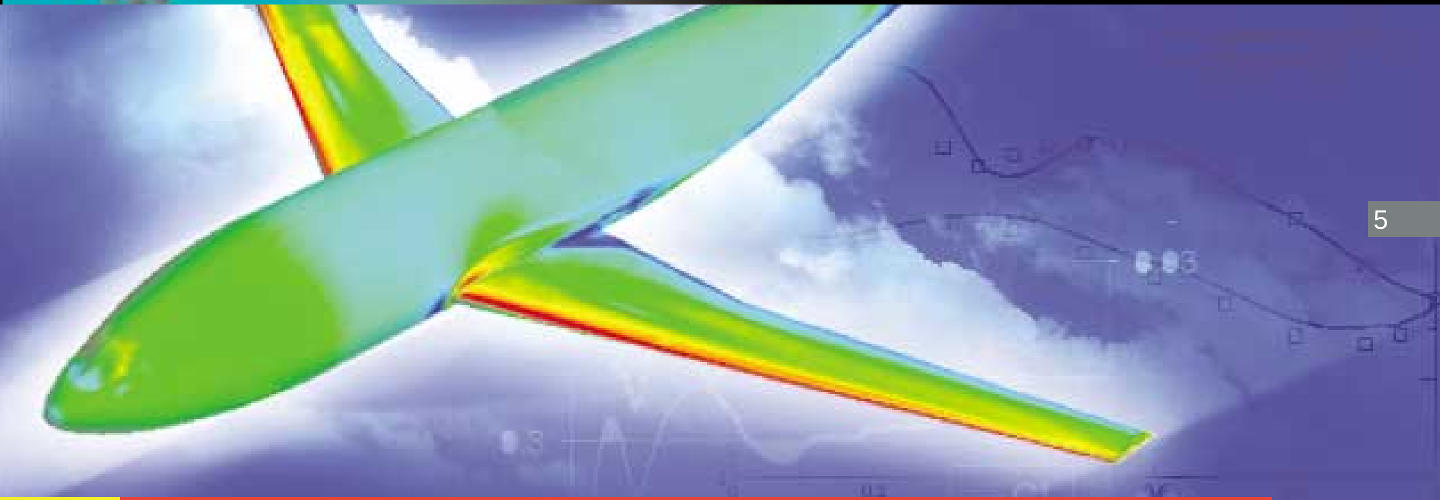


CFX Simulation for Predicting Aircraft Drag

Accurate results from a fully coupled formulation and algebraic multi-grid solver

By Florian Menter and Robin Lantry, ANSYS CFX

AIAA (American Institute of Aeronautics and Astronautics) has held two workshops in the last three years to evaluate the ability of modern CFD methods to accurately predict lift and drag of complete aircraft configurations. The first workshop was held in 2001 and had an active participation of 18 CFD groups. A DLR-F4 wing-body configuration test case was used with structured and unstructured grids provided by the organization and by ANSYS ICEM CFD. The results of this first comparison showed a wide spread between the results produced with different codes and turbulence models. Even different codes with the same turbulence model produced significantly different results. As a consequence of this outcome, it was decided to hold a second workshop on the same topic, but with different test cases.

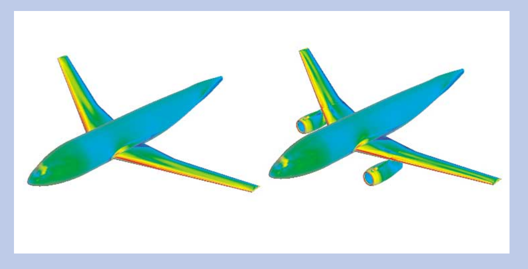
The second workshop was held in Orlando on June 21 and 22, 2003. Due to experience gained by participation in the EU aeronautics project FLOMANIA, CFX from ANSYS was able to compute the matrix of specified test cases. The geometries selected were the DLR-F6 wing-body (WB) and DLR-F6 wing-body-nacelle-pylon (WBNP) configuration. The goal of the comparison was to compute lift, drag and pitching moments of both configurations for a series of seven angles of attack.

To ensure the quality of the numerical results, participants also were required to perform a grid refinement study for each configuration. The structured grids provided by ANSYS ICEM CFD were:

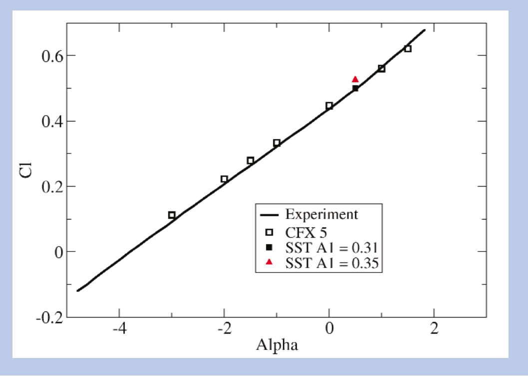
Grids	Coarse	Medium	Fine
WB	3,450,000	5,820,000	10,130,000
WBNP	4,890,000	8,430,000	13,690,000

Table 1: Number of nodes for different grids for WB and WBNP configuration

All of the grids were of the low-Re type with y^+ values of the order of one. Transition was specified in the numerical tests according to the transitions strips on the wind-tunnel models. The grid refinement study showed that the main solution parameters did not change between the medium and the fine grid. All simulations shown are computed on the medium grid.



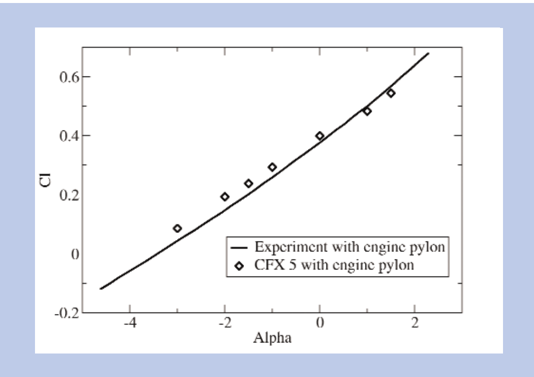
DLR-F6 configuration used as test case at the 2003 AIAA Drag Prediction Workshop. Left: wing body (WB); Right: wing body-nacelle-pylon (WBNP)



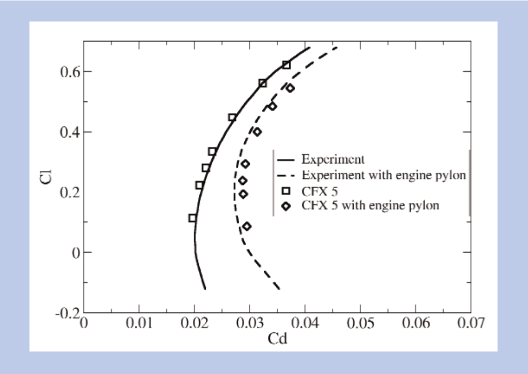
Lift vs. angle of attack for WB test case. Angles of attack were -3°, -2°, -1.5°, -1.0°, -0.0°, 1.0°, and 1.5°. Modification to the SST model, which mimics the adverse pressure gradient performance of the S-A model, produced an appropriate shift as shown by red triangle.

Numerical Method

All simulations were computed with CFX-5.6. CFX-5 is a finite, volume-based CFD method for unstructured hybrid meshes. It features some of the most advanced CFD techniques available today, namely a fully coupled formulation in combination with an algebraic multi-grid solver. For a pressure-based method, this combination has proven essential to obtain high-quality converged results for the current transonic test cases. CFX-5 is highly optimized in its physical modeling. The current simulations have been carried out using the SST turbulence model in combination with the CFX-automatic wall treatment.



Lift vs. angle of attack α for WBNP test case



Drag polar for WB and WBNP test case

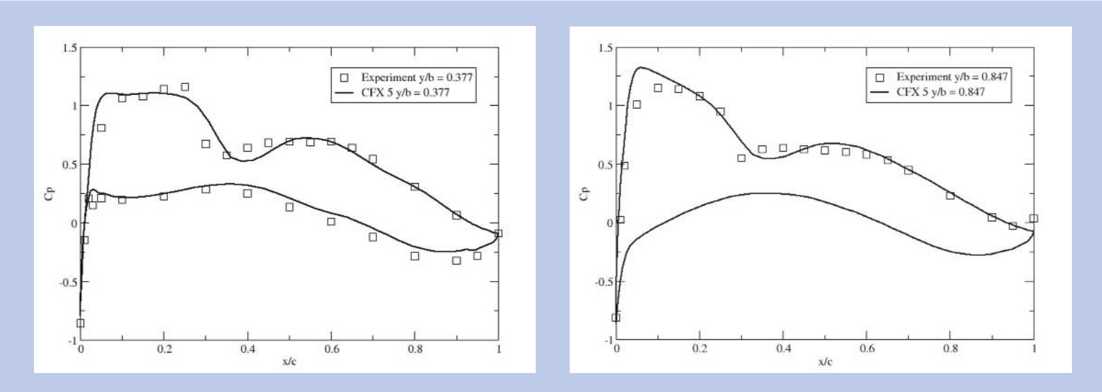
Results

The computed lift, C_l, vs. the angle of attack, a, for the wing-body test case (WB) is in very good agreement with the experimental data over the entire a-range. Surprisingly, it was found that the simulation of the lift curve proved difficult for most other codes in the comparison. In many cases, the lift curves were shifted by $\Delta\alpha$~0.25-0.5°, which is a significant deviation in aeronautical simulations. As the majority of these simulations have been computed with the Spalart-Allmaras (S-A) one-equation turbulence model, the sensitivity of the simulations to the turbulence model formulation has been investigated after the workshop using CFX. It was found that a modification to the SST model, which mimics the adverse pressure gradient performance of the S-A model, did indeed produce a shift of the expected magnitude.

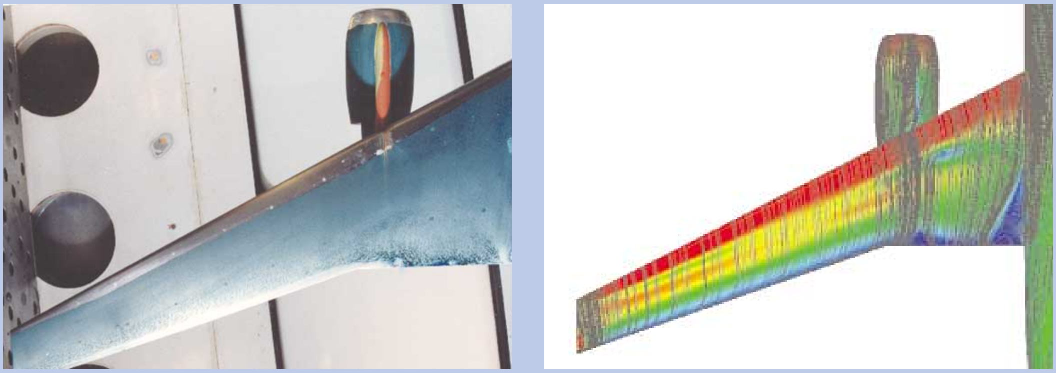
The results for the lift-curve for the WBNP case are again in good agreement with the experiments, except for negative angles of attack. Compared to most of the other codes, CFX was still quite close to the data. All simulations showed a tendency of over-predicting the lift at negative angles of attack. The main reason seems to be an over-prediction of a separation at the wing-pylon intersection, which affects the circulation around the wing. This issue is currently being investigated in more detail by several of the participating groups.

For an airplane, the drag is one order of magnitude smaller than the lift. It is therefore a notoriously difficult quantity to compute. The agreement of the CFX results with the experimental data for the drag-polar for both configurations is therefore surprisingly good for all angles of attack. It was possible to clearly identify the additional installation drag due to pylon and nacelle. The CFX results could therefore satisfy the main demand to accurately predict lift and drag of complete aircraft configurations using CFD posed by the workshop.

The shock on the upper surface, which is captured quite well by the simulations, can clearly be identified in the chart of surface pressure distributions at different span-wise locations on the wing of the WBNP case.



Wall pressure distributions at two span-wise locations for WBNP test case (yb/0.377 and yb/0.847) show the shock at the upper surface.



Separation details on top-side of wing. Left: experiment with oil flow. Right: CFX-5 simulation. Region of reverse flow can be identified.

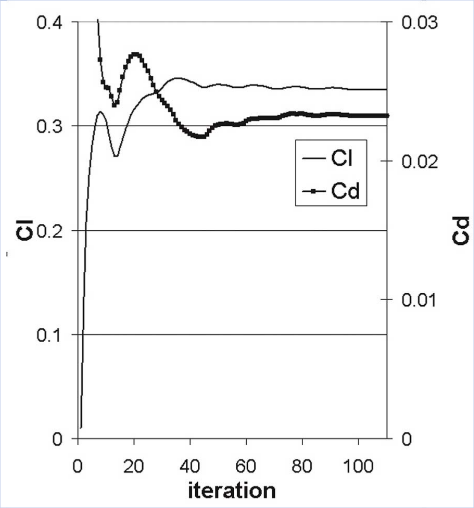
In the CFX analysis of flow details on the upper side of the wing for the WBNP configuration, small regions of flow reversal, which are in good agreement with experiment, can be identified.

Numerical Performance

One of the main limiting factors in CFD simulations for complete aircraft configurations is the relatively high computing requirements. It is therefore important that the nonlinear iterative CFD procedure converges with high efficiency. For the lift and the drag force for the WB case, CFX typically converged for all cases in about 100-150 iterative steps. This was one of the lowest iteration counts in the workshop comparison, although the comparison is not straightforward due to the many different numerical schemes involved. Due to the scalable parallelization of CFX-5, simulations for a grid of about 6 million nodes can be converged overnight on a modern 16-process PC cluster.

Conclusions

CFX from ANSYS has participated successfully in the validation study of AIAA. The studies showed that pressure-based methods are very well suited for the simulation of aeronautical flows at transonic conditions. Of the 25 participants, CFX software produced one of the most consistent sets of results in good general agreement with the experimental data.



Convergence of forces for WB test case

Web sites for More Information

Links

Results of First AIAA CFD Drag Prediction Workshop, Anaheim, CA June 9–10, 2001
aaac.larc.nasa.gov/tsab/cfdlarc/aiaadpw/Workshop1/Final_Schedule_and_Results.html

FLOMANIA
cfd.me.umist.ac.uk/flomania/

Results of Second AIAA CFD Drag Prediction Workshop, Orlando, CA June 21–12, 2003
aaac.larc.nasa.gov/tsab/cfdlarc/aiaadpw/Workshop2/Final_Schedule_and_Results.htm