Viscous Flow Analysis of a Twin-engine Commercial Transport Aircraft in High Lift Landing Configuration

Rajesh Ranjan¹, Abhishek Khare², Stimit Shah³, Kishor Nikam⁴
Computational Research Laboratories Ltd., Pune, India, 411016

and

Anutosh Moitra⁵
The Boeing Company, Seattle, USA

Extensive unstructured-grid CFD analysis has been performed for predicting performance of a twin-engine commercial transport airplane in landing configuration. The objective of the work was to identify and resolve complex gridding and solver related issues relevant to accurate prediction of complex flow physics associated with airplane high-lift systems. A variety of grid generation and CFD solution techniques were investigated for their efficacy in predicting performance of a complete airplane including high-lift devices e.g., multiple flaps and slats, and nacelle chines. Both steady and unsteady Reynolds Averaged Navier-Stokes (RANS) solution techniques were utilized. The role of relevant flow-physics phenomena in attainment of maximum lift as well as computational requirements for adequately modeling these phenomena were investigated. Computed aerodynamic forces are compared with available wind-tunnel test data.

I. Introduction

Prediction of flow over aircraft in high-lift configuration is of critical and vital importance in aircraft design. Small increases in lift-coefficient can result in relatively large performance benefits in terms of payload and range. This importance has prompted a great deal of effort undertaken by the fluid dynamics research community towards understanding the complex flow phenomena¹ associated with airplanes in high-lift flight conditions. Achieving that objective still presents a major challenge due to large complexities in the geometry of airplanes with deployed high-lift devices and the resulting flow-field characterized by flow separation, reattachment, wake/boundary-layer interactions, boundary-layer transition and other phenomena as yet not routinely tractable by CFD methods even for two-dimensional computations²³, or for simplified geometries⁴. Validation of CFD methods for high-lift prediction is further complicated by scarcity of publicly available wind-tunnel test data and by issues related to accurate modeling of the effects of tunnel-wall interferences⁵. A validated CFD method for accurately predicting airplane performance in high-lift mode will pay large dividends in the airplane design process by reducing design cycle time. While CFD is not expected to replace wind-tunnel testing, it can help reduce design costs by reducing the number of needed tests. This principally was the motivation for the present work.

The work described here was part of a collaborative project undertaken by Computational Research Laboratories (CRL), Pune, India, and The Boeing Company, Seattle, USA with the objective of identifying and addressing current limitations in CFD technologies for analysis of high-lift aerodynamics of airplanes and prediction of maximum lift. A previous phase of the collaborative project had addressed basic CFD issues using a simplified high-lift configuration, the NASA Trapezoidal Wing – a generic high-lift wing with leading and trailing-edge devices⁶. Results from this previous phase were documented by Khare, et al.⁷. The current phase builds on process

¹ Member of Technical Staff
² Member of Technical Staff
³ Member of Technical Staff
⁴ Senior Engineer and Head, CFD
⁵ Associate Technical Fellow-Boeing, Associate Fellow-AIAA
American Institute of Aeronautics and Astronautics
improvements developed in the previous phase and extends them to a real-world airplane configuration based on Boeing’s 777 airplane model. Recent advances in CFD solver technologies as well as high-performance computing platforms have provided feasible means for addressing the challenges of computing high-lift flow-fields. The present work utilized Metacomp Technologies’ CFD++ suite of flow solvers executed on CRL’s massively parallel computing system eka – currently the largest commercially available supercomputer in the world.

CFD++ solutions were obtained in both steady and unsteady modes. While steady Reynolds Averaged Navier-Stokes (RANS) solutions were found adequate at low to moderate angles of attack, unsteady RANS was utilized in attempts to resolve the unsteady flow-fields characteristic of high angles of attack. Grid refinement studies were performed to establish grid densities required for resolving complexities in the geometry and associated flow phenomena. Massively parallel computing technology was exploited wherever possible in pre-processing, solver, and post-processing phases to reduce analysis cycle time.

Subsequent sections of this paper will describe details of geometry and grid preparation, flow simulation, CFD analysis results, and comparisons with available test data followed by conclusions and an indication of future work.

II. Geometry and Grids
The geometry model for the airplane configuration studied in this work was provided by Boeing Commercial Airplane (BCA) pre-configured for high-lift analysis. No changes or modifications were made to this geometry to facilitate CFD analysis. The full-scale model had all high-lift devices pre-positioned in a landing configuration. The model included inboard and outboard flap, slats and a krueger leading-edge device. It also included the engine-nacelle with a chine vortex generator. Horizontal tail surfaces and landing gear were not included in the model.

Figure 2. Surface grid distribution in wing leading edge
The grid systems used in the present work were generated using Boeing’s Modular Aerodynamic Design Computational Analysis Process (MADCAP) and Advancing Front Local Reconnection (AFLR3) grid generator. MADCAP is a surface grid generator which takes surfaces in various formats such as STL, IGES etc. Geometry pre-processing and surface parameterization prior to input to MADCAP were accomplished using Boeing’s System for Low-Speed Unstructured Grid Generation (SLUGG) software system. AFLR3 is a volume grid generator which takes a triangulated surface grid in UGRID format and generates volume grid on that. AFLR3 generates the volume grid in two steps. In the first step it generates the viscous grid with prisms elements. Size and number of layers of prisms can be controlled by input parameters given to AFLR3. In the second step of volume grid generation AFLR3 uses advancing front algorithm to fill the remaining domain with tetrahedral elements.

Figure 5. Surface grid on aircraft

An initial grid study was performed in order to identify areas of the grid in need of further enrichment. These areas included the wing-body junction, gaps between elements – both chord-wise and span-wise, coves and wing-tips. The initial normal spacing at the solid surface was chosen to correspond to a \( y^+ \) value of 1.0. Different values of the grid stretching-ratio were investigated for improvements in the resolution of the boundary layers and wakes. Apart from this no special wake-resolution techniques were applied. Figure 2 to 5 display surface grids in critical regions. The sizes of the resulting volume-grids ranged from 60 million to 200 million cells.
III. Flow Simulation and Analysis of Results

A. Flow Solver

A compressible Reynolds Averaged Navier Stokes solver, CFD++ has been used to perform simulations. CFD++ uses cell centered, finite volume and implicit/explicit algorithms to solve the Navier Stokes equations on unstructured/structured grids. In CFD++ various topography parameter free models are used to capture turbulent flow features. The nonlinear subset of these models accounts for Reynolds stress anisotropy and streamline curvature. All turbulence models can be integrated directly to the wall or with a sophisticated wall function which accounts for compressibility as well. In the present study all results were computed by integrating to the wall surface. A minmod flux limiter is enabled to limit the interpolation slope in the second order solution scheme. Convergence of the simulation results varied with the turbulence model used and the angle of attack.

B. Simulation Conditions

Simulation conditions for the current analysis are tabulated in table 1. These conditions correspond to 4.2% scaled down model as tested in wind tunnel. For computational analysis a full-scale airplane model was used in free-air mode, i.e., no wind-tunnel walls were modeled in the computational scheme. CFD results were obtained at the Reynolds number corresponding to wind-tunnel tests. Results were compared with test data corrected for wind-tunnel wall and blockage interference effects.

A rectangular domain of size 100 times the body length in all three directions was used for simulations. Free stream pressure and velocity are imposed at the boundaries. Solve to wall approach was used and the grid was resolved to a small $Y^+$ (less than 1) value. The flow was assumed to be fully turbulent and different turbulence models were used to capture turbulent flow-fields and wake features. Both steady and unsteady simulations were carried out.

The simulations have been performed by CRL researchers on CRL’s massively parallel supercomputer eka, situated at Computational Research Laboratories, Ltd., Pune, India. “eka” is a cluster of high-end compute nodes connected with high speed communications networks. With 1800 nodes, the system has a peak compute capacity of 172 teraflops and has achieved sustained compute capacity of 132.8 teraflops for the LINPACK benchmark.

The principal objective of this collaborative effort between Boeing and CRL was to determine and address issues related to current shortcomings of CFD processes for predicting maximum lift produced by aircraft high-lift systems. Therefore the present analyses investigated the lift-coefficient $C_L$ as the principal metric. The drag coefficient $CD$ has been presented as an auxiliary metric but no special attention was directed towards its accurate prediction.

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Alphas</td>
<td>9.0 – 22.0</td>
</tr>
<tr>
<td>Beta</td>
<td>0.0</td>
</tr>
<tr>
<td>Mach</td>
<td>21.0</td>
</tr>
<tr>
<td>Reynolds</td>
<td>6000000.0</td>
</tr>
<tr>
<td>Turbulence</td>
<td>Fully Turbulence</td>
</tr>
<tr>
<td>Ambient Temp</td>
<td>518.67 R</td>
</tr>
</tbody>
</table>

Table 1. Simulation Conditions

Figure 6. Simulation Convergence

a. Residual Plot

b. Convergence of force quantities
Each steady state simulation on EKA takes around 24 hours of wall time to complete 1000 implicit iterations on a grid size of around 90 million grid cells with 192 processors. Convergence histories of both residuals and forces were monitored as shown in Figure 6. The simulation is considered to be converged when the energy or density residual reaches five orders reduction in value and the change in lift coefficient is no more than 1 count for the last 100 iterations.

C. Steady RANS Results

The first phase of the project used steady-state RANS computations for predicting $C_L$ values at various angles of attack. Although a highly accurate prediction of the absolute value of maximum $C_L$ is not strictly required for airplane design, a CFD process must demonstrate its ability to accurately model all flow phenomena associated with high-lift configurations at and beyond stall in order to establish its validity as a design tool.

Initial computations were performed for angles of attack ranging from -5 to 25 degrees and results were compared with experimental data in order to establish limitations of the CFD process in accurately predicting test data. Representative convergence plots for residuals and lift coefficient are shown in Figure 6. It was found that while reasonable agreement between computed and test data was obtained in the linear range of the $C_L$-alpha curve, severe discrepancies were noted as the angle of attack approached the value corresponding to maximum lift. Later phase of this study therefore focused on lift prediction at high angles of attack.

A baseline grid system was generated using Boeing provided grid parameter values in SLUGG defining surface and volume grid densities. These parameter values have been deemed to satisfy minimum grid requirements as a result of previous work at Boeing. This grid system had a normal grid stretching-ratio of 1.23. The baseline grid system had approximately 70 million cells. Initial results from this grid using the SA, SST, and KERT turbulence models available in CFD++ are shown in Figure 7. Computed lift values for all three turbulence models show early stall beyond an angle of attack of 14 degrees. Test results show stall at around 19 degrees. The extent of abrupt stall and the associated flow separation seems to be the largest for the SA model. At lower angles of attack $C_L$ values for all turbulence models show reasonable match with test data with SA and KERT models slightly over-predicting $C_L$ values. Subsequent phases of this study focused on angles of attack of 14, 16, and 18 degrees. Surface streamlines computed at an angle of attack of 18 degrees shown in Figure 8 clearly show large separation areas on the upper surface of the wing.

Figure 7. Effect of Turbulence Models – Baseline Runs

a. Lift co-efficient Vs Alpha

b. Drag co-efficient Vs Alpha
The non-physical separation on the inboard portion of the wing was seen to be largely responsible for the early stall. Using the surface flow-field as guide the surface grid was refined on the inboard wing and near the wing-tip region. The volume grid was also refined by reducing the normal grid stretching-ratio from 1.23 to 1.2. Previous studies of the Trapezoidal wing\textsuperscript{7} had shown that a stretching ratio of 1.2 was required for resolving the boundary layer in high-lift flows. The grid resulting from these modifications contained 100 million cells. Lift increments for the KERT turbulence model resulting from the reduction in stretching-ratio are presented in Figure 9.
The plot shown in Figure 10 presents comparisons of computed lift and drag coefficients between the baseline grid and the new refined grid. Although the new grid is seen to cause substantial increases in the computed lift values, it is clearly seen that the relevant physics is still not captured accurately and early stall is still indicated in the computed data. The KERT turbulence model shows the largest improvement in lift coefficient but stall behavior is not predicted correctly.

Figure 10. Effect of stretching ratio
Upper surface streamlines from SA results at 14 and 18 degrees angle of attack are presented in Figure 11 and 12 respectively. The difference in surface flow patterns at the two angles of attack illustrate the reasons for the early stall noted in lift coefficients. At 18 degrees the flow on the inboard wing is seen to be very unstable and a large region of flow separation is seen to exist on the outboard part of the wing. This extent of separation was not indicated in test data.

Figure 11. Upper surface streamlines at alpha = 14 degrees

Figure 12. Upper surface streamlines at alpha = 18 degrees
Based on these computational experiments it was concluded that the reasons for the discrepancies between computational and test data could be related to the inherent unsteadiness of high-lift flow-fields at high angles of attack. The inboard wing flow-field is characterized by strong and unsteady interactions of the nacelle-chine vortex with the wing boundary layer. Previous attempts at resolving this issue have been reported by Rogers, et al., and Slotnick, et al. A steady-state computational model may not be adequate for accurately capturing this interaction. To explore this possibility, unsteady RANS computations were undertaken in the next phase of the project.

C. Unsteady RANS Simulations

Vortex systems produced by the chine on the engine-nacelle plays a crucial role in attainment of high lift at high angles of attack. The chine-vortex interacts with the boundary layer on the inboard portion of the upper wing surface and delays its separation. This interaction is inherently unsteady. Particle traces denoting the chine-vortex are shown in Figure 13. There are other unsteady interactions between the slat-edge vortices and the boundary layer as well. An attempt was made to capture these unsteady phenomena by means of unsteady RANS computations using the SA turbulence model. Streamlines on the upper surface of the wing presented in Figure 14 gives and indication of the evolution of these unsteady interactions.

Figure 13. Particle traces of the chine-vortex

Figure 14. Unsteady vortex/Boundary-layer interactions
With continued iteration the unsteady flow demonstrated a periodic behavior. The time-step size for time-accurate computations was adjusted to provide 100 time-steps within each periodic cycle. The extent of vortex interactions can be seen in the vorticity contours presented in Figure 15 and 16.

Figure 15. Vorticity Iso-surfaces

Figure 16. Vorticity contours in sectional plane
A typical plot of residual convergence at 18 degrees in unsteady RANS mode is shown in Figure 17. Wing upper surface streamlines for converged state of unsteady RANS simulation at 18 degrees are presented in Figure 18. The separation region at the wing-tip is seen to be much smaller in the unsteady solution as compared to steady results in Figure 12. The inboard wing region shows a large separation region in both steady and unsteady flow-fields.

Figure 17. Residual convergence for unsteady computation.

Figure 18. Unsteady RANS upper surface streamlines
Lift and drag coefficients computed by unsteady RANS are plotted against angle of attack in Figure 19 and 20 respectively. Compared to steady RANS results the stall behavior of the lift curve is much smoother, however stall angle is still under predicted. The reasons for the low lift level at high angles of attack can be attributed to inadequate grid resolution of the chine-vortex core and its interactions with the boundary layer on the inboard portion of the wing upper surface. An effort to address these issues by further refinement of the volume grid was undertaken but could not be completed in the duration of the project. This work is planned for a future project.

Figure 19. Unsteady RANS Lift coefficient Vs Angle of attack
IV. Conclusion

Steady and unsteady RANS simulations were performed for prediction of maximum lift of a twin-engine commercial transport airplane in high-lift landing configuration. Grid density on the surface of the airplane as well as in the volume discretization showed large effects on the accuracy of the computed lift levels. Unsteady computations showed substantial improvements in the predicted stall behavior by avoiding abrupt stall compared to test data. The final converged states of the flow-field were found to be significantly different for steady and unsteady computations. Both types of simulation resulted in quasi-steady flow-fields, however the end-state flow-physics had very different characteristics. It appeared that the states representing higher lift values could only be approached in a time-accurate computational mode. Computed maximum lift coefficient values were within 8% and 2% of test data for steady and unsteady simulations respectively. The angle of attack corresponding to maximum lift was under predicted by 5 degrees for steady analysis and 2 degrees by unsteady simulation. The chine-vortex and its interactions with the boundary layer were noted to have large effects on computed lift levels. Further grid studies are required for accurately simulating these interactions.

Acknowledgments

The authors are grateful to Metacomp Technologies, Inc., USA for their valuable support and insights regarding the CFD++ solver suite.
References