Numerical Simulation of High Lift Trap Wing using
STAR-CCM+

Prashanth Shankara¹ and Deryl Snyder²
CD-adapco, Melville, NY, 11747

Results from numerical simulations of the NASA trapezoidal wing geometry using the unstructured finite-volume-based solver STAR-CCM+ are presented. General polyhedral meshes are generated using automated techniques that are amenable to rapid grid generation for industrial use. A steady-state RANS approach is employed using the SST k-ω turbulence model coupled with a predictive laminar-to-turbulent transition model. Results are compared to experimental measurements provided as part of the 1st AIAA High-Lift Prediction Workshop (HiLiftPW-1), both with and without slat/flap support brackets.

Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>AR</td>
<td>aspect ratio</td>
</tr>
<tr>
<td>b/2</td>
<td>semi-span</td>
</tr>
<tr>
<td>Cp</td>
<td>pressure coefficient</td>
</tr>
<tr>
<td>Cref</td>
<td>reference chord length (mean aerodynamic chord)</td>
</tr>
<tr>
<td>TR</td>
<td>taper ratio</td>
</tr>
<tr>
<td>S</td>
<td>reference area</td>
</tr>
<tr>
<td>Reθ</td>
<td>momentum thickness Reynolds number</td>
</tr>
<tr>
<td>γ</td>
<td>turbulence intermittency</td>
</tr>
<tr>
<td>CL</td>
<td>lift coefficient</td>
</tr>
<tr>
<td>CD</td>
<td>drag coefficient</td>
</tr>
<tr>
<td>CM</td>
<td>moment coefficient</td>
</tr>
<tr>
<td>Cf</td>
<td>skin friction coefficient</td>
</tr>
<tr>
<td>AoA</td>
<td>angle of attack</td>
</tr>
</tbody>
</table>

I. Introduction

Predicting high-lift flows using computational fluid dynamics (CFD) is difficult due to complexities both in geometry and physics. High-lift configurations contain multiple elements all in close proximity, and may be further complicated by brackets and/or other mechanical or aerodynamic devices. Generating suitable meshes for such complex geometries while maintaining efficient use of cells is a challenge. Point-to-point conformal block structured meshes can be tedious to build for such geometries, and may not be feasible in realistic configurations, such as where slats and flaps exist on separate span-wise sections of the wing. In these instances, overset mesh techniques or unstructured meshes become necessary. The physics to be modeled are also challenging, with flow phenomena such as turbulent transition, separation, confluent boundary layers, and highly vortical flow all playing an important role.

Due to the importance of aerodynamic performance during takeoff and landing to fuel consumption, payload, safety, and noise, the ability to accurately model this class of problem – particularly in a manner well-suited for industrial design use – is desirable. To this end, the first AIAA High-Lift Prediction Workshop (HiLiftPW-1) was organized and conducted to assess the capabilities of current-generation CFD codes in terms of meshing, numerics, turbulence modeling, and high-performance computing.

This paper presents the calculations, procedures, and results of applying the STAR-CCM+ CFD package (CD-adapco, Melville, NY / London, England) to the HiLiftPW-1 case. Of particular interest is the ability to model this class of problem in an industrial setting rather than a purely research setting. As such, performance utilizing an

¹ Technical Marketing Engineer; Email: prashanth.shankara@cd-adapco.com
² Director, Aerospace and Defense, AIAA Senior Member; Email: deryl.snyder@cd-adapco.com
unstructured polyhedral mesh is of interest due to the greatly reduced meshing effort compared to structured (both point-to-point conformal and overset) meshing approaches. In addition to the standard configuration, performance prediction including slat/flap brackets is investigated.

Due to the low Reynolds numbers experienced during takeoff and landing, laminar-to-turbulent transition is a consideration in analyzing these flows. Therefore, a two-equation predictive transition model is employed in addition to the steady-state RANS equations.

II. NASA Trap Wing Geometry & Experiments

The geometry studied in this effort is “Configuration 1” for the NASA Trapezoidal Wing (Trap Wing) model. Several wind tunnel tests have been conducted using this model, beginning in 1998. Data used in the HiLiftPW-1 were collected between 1998 and 2003, including aerodynamic forces and moments, pressure distributions, transition locations, and aeroacoustics. The overall objective of the tests was to obtain detailed data sets specifically to allow assessment and validation of CFD methodologies. As such, the trap wing is a representative high-lift configuration with a simple three-element design attached to a body pod. The wing has a relatively large chord and low aspect ratio. It has straight taper and sweep, but no twist or dihedral. An overall view of the geometry as well as a summary of key geometric parameters is shown in Figure 1. Configuration 1 is the fully deployed configuration, with slats at 30 degrees and flaps at 25 degrees. The experimental Reynolds number was approximately 4.3 million, with no transition tripping mechanisms used.

![NASA Trap Wing](image)

**Figure 1. NASA Trap Wing geometry and reference parameters.**

### III. Computational Approach

Steady-state RANS simulations were performed with STAR-CCM+ V6.04, which employs a cell-centered finite-volume method allowing the use of arbitrary polyhedral cell topologies. The formulation is fully-coupled and density-based, with second-order-upwind spatial discretization for the convective flux terms, and second-order central discretization for the diffusion terms. An implicit pseudo-time-marching scheme is used to arrive at a steady-state solution. Preconditioning is used to make this approach suitable for low-speed, isothermal flows. A point implicit (Gauss-Seidel) linear system solver is used with algebraic multigrid (AMG) acceleration to solve the resulting discrete linear system at each iteration. Turbulence closure was achieved using the SST k-ω model, which is well-known and therefore not discussed in detail here.

A grid-sequencing initialization technique was used to improve robustness and reduce solution time. This initialization approach computes a low-cost approximate solution to the flow problem. Specifically, an implicit incomplete-Newton algorithm is used to compute a first-order inviscid flowfield solution using a series of coarsened meshes.

IV. Transition Modeling

Due to the relatively low Reynolds number (4.3 million) and the absence of turbulent trip mechanisms, transition from laminar to turbulent flow is important to this case. To model transition, the correlation-based \( \gamma-\text{Re}_\theta \) transition model, formulated for unstructured meshes, was used. To avoid direct evaluation of non-local quantities such as momentum thickness Reynolds number, \( \text{Re}_{\theta_m} \), which cannot be computed on unstructured meshes, this quantity is related to a vorticity-based Reynolds number. A correlation for transition-onset momentum thickness Reynolds number, \( \text{Re}_{\theta_m} \), defined in the free stream, is propagated into the boundary layer via a transport equation. An
intermittency transport equation is further used in such a way that the source terms attempt to mimic the behavior of algebraic engineering correlations. This model, coupled to the SST k-ω turbulence model, solves two additional transport equations: one for $\gamma$ and one for $Re_{\theta_c}$. The model relates the local $Re_{\theta_c}$ to the transported transition-onset value, $Re_{\theta_c}$, and adjusts intermittency production appropriately. The $\gamma$-$Re_{\theta}$ transition model requires a high-quality, refined low-Reynolds number mesh with near-wall spacing such that $y^+$ is between 0.1 and 1.0. If transition is induced by laminar separation, the mesh must be sufficiently refined in the stream wise direction to resolve the curvature of the boundary where separation exists and the separation bubble itself. The interested reader can find details of the $\gamma$-$Re_{\theta}$ model implementation within STAR-CCM+ in Ref. 14.

V. CFD Mesh and Computational Domain

Four general polyhedral meshes were generated using STAR-CCM+: three meshes of varying resolution for the no-bracket case, and a single “Medium” density mesh including brackets. Meshing guidelines provided by the HiLiftPW-1 committee were generally followed, although adapted to an automated unstructured meshing procedure. A summary of the key mesh parameters are presented in Table 1. Figure 2 shows the surface mesh density for the Coarse mesh near the root and tip. Surface faces were refined along the leading and trailing edges of each element, near the tip, and at the wing/body junction and other sharp corners.

Via the use of arbitrarily-shaped volumetric control regions, the volume mesh was refined in several ways, as illustrated in Figure 3, showing several section views. First, an overall refinement was made near the wing, in the near wake, and near the wing tip. In addition, mesh refinement was defined in the regions between the slat and main-wing, and the flap and main-wing in order to capture separated flow regions and better resolves the wake/boundary layer interaction. A robust automated prism layer meshing algorithm was used to capture the boundary layer, with wall $y^+$ values kept in the 0.1 to 1.0 range. The Medium and Fine grids were generated by reducing the global cell size of the mesh by 18% and 30%, respectively, and adding additional cells in key areas such as the leading and trailing edges and wing tip. Meshes were not customized for any particular angle of attack, rather angle of attack sweeps were accomplished by changing the free stream flowfield direction while leaving the mesh unchanged.

Figure 4 shows the surface mesh on the flap and slat supports. For the initial study, no special refinement zones were created to address the flow around the brackets and the surface mesh is allowed to grow slowly into the volume mesh.

<table>
<thead>
<tr>
<th>Table 1. Mesh Parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
</tr>
<tr>
<td>Number of Cells</td>
</tr>
<tr>
<td>Number of Surface Faces</td>
</tr>
<tr>
<td>Target Prism Layer Height</td>
</tr>
<tr>
<td>Number of Prism Layers</td>
</tr>
<tr>
<td>Cells Across Trailing Edge</td>
</tr>
</tbody>
</table>

Figure 2. Surface views of the Coarse mesh showing general face sizes and refinement. (a) Mesh near the root. (b) Mesh near the tip.
The computational domain used in this study models a free stream condition rather than attempting to model the wind tunnel test section. A symmetry condition is used at the body pod mounting plane, and the domain extends approximately 100C_ref from the wing in the lateral and span wise directions. Free stream boundary conditions specifying flow direction, Mach number, temperature, static pressure, and turbulence quantities are used at the far field boundaries. Table 2 summarizes the free stream conditions applied.

<table>
<thead>
<tr>
<th>Table 2. Free stream Boundary Conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mach</td>
</tr>
<tr>
<td>Temperature</td>
</tr>
<tr>
<td>Static Pressure</td>
</tr>
<tr>
<td>Angle of Attack</td>
</tr>
</tbody>
</table>

(a) Mesh refinement in the general region of the wing and wake. (b) Mesh refinement in the separated region between the main-wing and flap. (c) Prism layers and refinement region between the slat and main-wing. (d) Close-up view of the slat trailing edge and slat/main-wing gap.

Figure 3. Section views of the Coarse mesh showing polyhedral mesh structure and resolution.

Figure 4. Surface views of the mesh on the slat and flap brackets. (a) Flap brackets. (b) Slat brackets.
VI. Results

A. Configuration 1, No Brackets

Solutions at angles of attack from 6 to 37 degrees are obtained on each of the three meshes in order to assess the effect of mesh resolution on the results. These results include the $\gamma$-Re$\theta$ transition model. At angles of attack below 28 degrees, the solutions were obtained by initializing using the grid-sequencing initialization procedure described previously in Section III. For angles of attack 28 degrees and higher, the solution procedure was to start from the converged solution at the previous angle of attack. In some instances (28, 32, 34 degrees) the transition model was turned off as the solution was changed from one angle of attack to the next in order to maintain solution stability, then turned back on to obtain the final solution. Above 34 degrees the incremental angle of attack change was sufficiently small (1 degree) such that the transition model could remain active throughout the solution procedure. Due to the inherent unsteadiness at the higher angles of attack, the final force coefficients are the averaged values over a number of iterations after the coefficients settle in to an oscillation about a mean.

1. Force and Moment Coefficients

Figure 5 shows the lift coefficient as a function of angle of attack. Overall agreement with experimental measurements is quite good, with a $C_{L\text{Max}}$ deviation of 1.6% for the Medium mesh. Similarly, the experimental measurements show stall occurring at 36.1 degrees, and the CFD solution indicates stall occurring between 36 and 37 degrees (angle-of-attack increments of 1 degree are used near stall). Post-stall $C_L$ is under-predicted, although the result is generally satisfactory due to the inherent unsteadiness of the massively-separated flowfield. Agreement between the mesh resolutions is also good, with little variation at angles of attack below 28 degrees, although the $C_{L\text{Max}}$ is higher for the Medium mesh compared to the Coarse and Fine meshes. Generally, the prediction of lift coefficient improves with mesh refinement. The $C_{L\text{Max}}$ deviation for coarse mesh is 3.3%, medium mesh is 1.6% and fine mesh is 0.4%.

The drag coefficient as a function of angle of attack is presented in Figure 6. Again we see very good agreement with the experiment overall, and little deviation with grid refinement at angles of attack below 28 degrees. Above 28 degrees, the fine mesh yields answers more consistent with the experiment, although the results from all three meshes stay within the experimental error bounds. Similarly, the predicted drag polar is plotted in Figure 7.

Pitching moment behavior is plotted in Figure 8. Here we see a distinct improvement in the predicted values using the Medium mesh. The discrepancy in moment coefficient prediction is mostly due to the differences in resolution of flow separation on the flap between the grids. More investigation on the flap flow separation by refining on different levels along the flap surface is needed to understand this behavior. Figure 9 shows the effect of mesh refinement on prediction of the flap body separation with the transition model. The finest mesh predicts very little separation on the flap body and also gives the best results. For the coarse and medium mesh, the effects of under-predicting the flap separation are offset by the surface pressure predictions, resulting in good prediction of the force coefficients.
Figure 5. Predicted lift coefficient for Configuration 1 without brackets.

Figure 6. Predicted drag coefficient for Configuration 1 without brackets.

Figure 7. Predicted drag polar for Configuration 1 without brackets.
2. **Surface Pressure Coefficients**

A comparison of the surface pressure distribution with experimental data is also presented. This analysis indicates the ability of the CFD simulation to predict the different surface flow phenomena accurately. Experimental measurements are provided at specified span wise locations, indicated by \( \eta \) as a percentage of the total semi-span of the wing. Figure 10 shows the \( C_p \) results at 13 degrees AoA at four different sections along the span. The numerical results accurately predict the pressure distribution at the 17%, 50% and 85% sections on all three elements. At the 98% section, flow separation on the flap is predicted prematurely. This is a direct result of the under prediction of the reattachment of the flow on the main element. Further mesh refinement at the wing-tip is necessary to better resolve the multiple vortex structure that exists on wings with flat end-caps\(^5\). Resolution of these structures is expected to improve the pressure predictions in this region.

In Figure 11, the \( C_p \) prediction at 28 degrees AoA is again seen to be excellent at all span wise locations on the slat. On the main element, computational results are in good agreement at the 17%, 50%, and 85% sections. At the 98% section, as seen in the 13 degrees case, the vortical flow from the wingtips is harder to predict. Also, at this higher angle of attack, there is significant flow separation near the wing tip as the flow approaches full stall. The flow separation is over-predicted on both the main element and the flap near the wing-tip. Also, there is slight under-prediction of the pressure on the suction side of the flap at 50% span, likely due to insufficient resolution of the confluent boundary layer.

Figure 12 shows similar comparisons at 34 degrees angle of attack, where the flow is beyond \( C_{l_{\text{max}}} \) and is nearing full stall. Even at this high angle of attack, the computational results agree favorably with the experimental data and show similar behavior at 17% and 50% span. At 98% span, there is even greater deviation from experimental data on the main element and the flap but the overall prediction of the lift coefficient is still good compared to test values at this angle of attack.
Figure 10. Cp distributions for 13 degrees AoA at four span wise locations. (a) $\eta = 17\%$. (b) $\eta = 50\%$. (c) $\eta = 85\%$, (d) $\eta = 98\%$

Figure 11. Cp distributions for 28 degrees AoA at four span wise locations. (a) $\eta = 17\%$. (b) $\eta = 50\%$. (c) $\eta = 85\%$, (d) $\eta = 98\%$
Figure 12. $C_p$ distributions for 34 degrees AoA at four span wise locations. (a) $\eta = 17\%$. (b) $\eta = 50\%$. (c) $\eta = 85\%$, (d) $\eta = 98\%$

Figure 13 shows “oil-flow” visualizations on the surface via the calculation of surface-constrained streamlines. Also visualized is the skin friction coefficient. At AoA = 13 degrees, the flap shows a significant flow separation just aft of mid-way along the chord for nearly the entire span. Also, evidence of the strong tip vortex is noted by the higher values of skin friction near the wing tip. At AoA = 28°, the flow separation on the flap is reduced compared to 13° angle of attack, and limited to the outboard sections being largely influenced by the tip vortex. Flow separation at the flap-body intersection is also indicated by these surface streamlines. Significant flow separation is apparent near the wing tip of the main element at AoA = 34 degrees, while the flow separation on the flap is yet smaller than the lower angles of attack. Further visualization of flow separation near the wing-tips and their influence by the tip vortex is given in Figure 14, which shows the flow streamlines at the same selection of angles of attack.

Figure 13. Computed surface streamlines. (a) AoA = 13 degrees. (b) AoA = 28 degrees. (c) AoA = 34 degrees.
Figure 14. Computed streamlines. (a) AoA = 13 degrees. (b) AoA = 28 degrees. (c) AoA = 34 degrees.

B. Support Brackets

Analysis of the effect of flap and slat support brackets was conducted at different angles of attack from 6 to 28 degrees. An investigation into the numerical prediction of high-lift aerodynamics including the support effects will pave the way for understanding phenomena such as local flow separation, and the influence of the bracket wakes on stall. The 22M cell medium polyhedral mesh for Configuration 1 is used as the baseline to generate the mesh for the geometry including the supports, as discussed earlier in Section V. Figure 15 shows the comparison of the lift coefficient between the experiments and numerical predictions with and without the support brackets. The numerical simulations produce reasonable results at angles of attack of 21 degrees and below, with a slight under-prediction of $C_L$ compared to the simulations without brackets. After 21 degrees, the numerical simulations with brackets predict the flow is separating, resulting in loss of lift. Figures 16 and 17 show the comparison of drag coefficient and pitching moment, respectively, which again deviate from the experiment above 21 degrees angle of attack.

Figure 15. Predicted lift coefficient for Configuration 1 including brackets.
Figures 18 and 19 shed some light on the cause of the premature loss of lift. Visualization of the skin friction shows the wakes of the slat brackets on the upper surface induce local flow separation on the main wing at 28°. A possible remedy for this behavior is to increase the mesh resolution in these regions. The approach used in this study is to create the mesh for the brackets based on the Medium mesh without focused refinement around the brackets and their wakes. Future studies will include grid adaption to capture the wake flows from the brackets more accurately. Also, the influence of the transition model needs further investigation.
C. Transition Model Analysis

The inclusion of the $\gamma$-Re$_\theta$ transition model in the numerical simulations was found to improve the prediction of lift, especially at higher angles of attack, and to better estimate the onset of stall. The solution procedure used was to initially use the SST k-ω turbulence model without transition to arrive at an intermediate solution. Using this solution as the initial guess, the transition model was activated. Figure 20 shows the effect of the $\gamma$-Re$_\theta$ model for the 28 degree AoA case. The fully turbulent solution converged to a $C_L$ value of 2.8, while the transition model captured the locations of separation/transition more accurately, leading to an increase in the predicted $C_L$ up to 2.86 (experimental value of 2.9). The same methodology is followed for all angles of attack, with similar behavior.

Flow visualization techniques are used to identify the onset of transition for each case. One of the important parameters for visualizing transition is the turbulence intensity in the flow field, showing the transition of the flow from laminar to turbulent. Another parameter is the turbulence intermittency, $\gamma$, where a value of 0 suggests laminar flow and a value of 1 refers to fully turbulent flow. Figure 21 shows a comparison of the turbulence intensity at 70% span for the 13 degree AoA case with and without the transition model. Without the transition model, the flow is modeled as turbulent all along the suction surface of the flap, while the transition model predicts laminar-turbulent transition on the flap, visualized from the change in turbulent intensity as indicated in the figure. Figure 22 shows the prediction of transition by the $\gamma$-Re$_\theta$ model on the slat at mid-span section. Seen is the clear onset of transition on the upper surface at 28 degrees AoA, where the intermittency changes from 0 to 1. At 13 degrees AoA, the intermittency value close to the upper surface stays at 0 throughout, indicating a laminar boundary layer.
Figure 21. Comparison of turbulence intensity contours around flap for fully turbulent (left) and transitional (right) simulations at $\eta = 70\%$ and $\alpha = 13$

Figure 22. Predicted intermittency at $\eta = 50\%$ near the slat at (a) $\alpha = 13$ degrees and (b) $\alpha = 28$ degrees.

VII. Conclusion

Numerical results for high-lift prediction of the NASA Trap wing obtained using STAR-CCM+ agree very well with the experimental data. Attempting to capture the laminar-turbulent boundary layer transition phenomenon via the use of the predictive $\gamma$-Re$_θ$ transition model is seen to improve the prediction accuracy. Grids of all three sizes show good comparison with test data, with the results agreeing favorably even at the coarsest grid level. Surface pressures are well-predicted at all angles of attack prior to full stall and at all stations along the span, except very near the wing-tip, where more mesh resolution is needed to capture the development of the wing-tip vortices along the flat tips. Preliminary results for simulations including bracket supports show premature loss of lift compared to experimental data, as was the trend for other codes participating in the HiLiftPW-1$^{,4}$. Insufficient grid resolution of the bracket wakes is the likely cause, but further investigation is needed.

Acknowledgments

The authors would like to acknowledge the contribution of Penguin Computing for providing hardware resources for completing the computational runs. The authors would also like to thank James Clement and David Vaughn from CD-adapco for their intellectual contributions.

References

3http://hilifpw.larc.nasa.gov/

American Institute of Aeronautics and Astronautics


