Published on *CD-adapco* (http://www.cd-adapco.com)

Modeling the Performance of the Standard Cirrus Glider

[1]

Thomas Hansen
CMR Prototech

Geometry and CFD methodology

The Standard Cirrus was designed by Dipl. Ing. Klaus Holighaus at the Schempp-Hirth factory and is a 15 m glider without flaps, originally designed
to compete in the standard class. The glider uses an all moving tail plane, is equipped with air brakes on the upper side of the wings, and can carry up to 80 kg of water ballast to increase the flight performance. The best glide ratio for the glider is about 1:37 and the maximum speed is 220 km/h.

By using a digitizing arm and a hand held laser, the ?as built? geometry of a Standard Cirrus - including the airfoils, elevator and rudder - were measured and imported as the underlying geometry for the CFD model. The fuselage of the glider was designed by modifying an existing CAD model of the Standard Cirrus, used to perform a similar study at the German Aerospace Center (DLR).

The Navier-Stokes solver, STAR-CCM+, was used to both create the numerical mesh and to perform the simulation of the glider. To accurately model the boundary layer effects and to capture the important laminar to turbulent transition process, the correlation based ?-Re? transition model in STAR-CCM+ was applied.

The simulations were conducted in two steps:

? 2D simulations of the airfoil at the outer part of the wing to validate the transition model,
? 3D simulations of the Cirrus glider for velocities ranging from 90 km/h to 160 km/h.

The simulations were validated by comparing the results to wind tunnel data and flight tests respectively. To model the turbulent flow, the k-? SST turbulence model was used in combination with the transition model. The Reynolds Averaged Navier-Stokes (RANS) equations were solved in STAR-CCM+ using the segregated solver and the air-flow was modeled as steady and incompressible using a constant density model. All simulations were performed on a Dell power blade cluster running 36 CPU?s in parallel.

The laminar to turbulent transition process is important when predicting the performance of gliders. For a glider under normal flight conditions, the Reynolds number does not exceed 3 million. Below this Reynolds number, the transition process often takes the form of a laminar separation bubble. When this occurs, the separating laminar layer is followed by turbulent reattachment, just behind a recirculation region. In the figure, an illustration of the transition process on the upper side of an airfoil is shown.

For the simulations of the Standard Cirrus, this transition process is modeled using the ?-Re? transition model. The ?-Re? transition model in STAR-CCM+ is correlation based, and solves two extra transport equations, one for intermittency and one for local transition onset momentum thickness Reynolds number. The only user input required to operate the transition model is the definition of the free-stream edge. In contrast, by solving the RANS equations as fully turbulent, the transition process is ignored and only turbulent air-?ow is present in the boundary layer.

Accuracy of the ?-Re? transition model

To investigate the accuracy of the ?-Re? transition model, the performance of an airfoil section located on the outer part of the Standard Cirrus wing was investigated in two dimensions. The simulations were validated by comparing the results to experimental data from the low-turbulence pressure wind tunnel at NASA Langley. By performing mesh dependency studies, the required quality of an O-mesh having the smallest number of cells when using the ?-Re? transition model was investigated in detail. The grids for the dependency study were constructed using a hyperbolic extrusion method in Pointwise and all grids have y+ values below 1. The two dimensional simulations are also compared to calculations performed using the panel codes Xfoil and Rfoil.

As seen in the graphs, the CFD simulations using the transition model compare well to the experimental data. The transition model predicts the lift coefficient equally well as the panel codes, Xfoil and Rfoil, for the angles of attack between -5 and 5 degrees. For higher angles of attack, the transition model compares better to the experimental data than the panel codes. However, the transition model is found unable to simulate the occurrence of the stall, and a too high lift coefficient is predicted in this region. The fully turbulent CFD model can be seen to underestimate the lift coefficient for all positive angles of attack. This is because the pressure around the airfoil is wrongly calculated as no laminar air-?ow is present in the model.

The drag predictions for the CFD simulations with the transition model compare better to the experimental data than Xfoil and Rfoil. In contrast, by not accounting for the the laminar air-flow in the boundary layer, the fully turbulent CFD model significantly over predicts the drag coefficients for all angles of attack. The O-mesh with the smallest number of cells that enables the ?-Re? model to converge for the interesting angles of attack have 600 cells wrapped around the airfoil, a growth rate of 1.05 and y+ values between 0.1 and 1. Further cell count reduction on the airfoil also reduces the range of solvable angles of attack. Also shown is a comparison of the position of the laminar separation bubble.

Standard Cirrus calculations

To simulate the performance of the Standard Cirrus, two CFD models were constructed and calculated:
? One model was created to calculate the lift and drag coefficients for the wing and fuselage of the glider.
? Another model was created to calculate the drag coefficient of the tail section.

A trimmed hexahedral mesh was used to discretize the two models in STAR-CCM+. To reduce the number of cells in the mesh, symmetry conditions were applied and only half the glider was simulated. The required quality for the 3D mesh when using the ?-Re? transition model was investigated in a mesh dependency study. The outer boundary of the flow domain was created as a half sphere, positioned 50 m from the glider surface and it is constructed using a velocity inlet upstream of the glider and a pressure outlet downstream. To accurately capture the boundary layer flow, 20 layers of hyperbolic extruded prismatic cells were constructed from the glider geometry. Since the y+ value for the mesh scales with the velocity, the grids at high velocities required a smaller distance to the ?rst cell centroid. At high angles of attack, more cells on the wing of the glider are needed to obtain a converged solution. The number of cells in the mesh was thus increased from 28 million to 41.8 million cells for the 95 km/h to the 160 km/h simulation, respectively. The free-stream edge definition for the transition model was specified as 5 mm away from the glider surface. For comparison, both fully turbulent and transitional calculations were performed using the same mesh.

In the picture above, the production of turbulent kinetic energy on the top surface of the glider is depicted. The figure shows the results from the 95 km/h and 160 km/h simulation on the right and left wing respectively. The production of turbulent kinetic energy is a measure of the turbulence in the air-flow and the regions with highest production indicates the location of the laminar separation bubble. On the top surface, the 160 km/h simulation shows that the separation bubble has moved slightly backwards compared to the 95 km/h simulation, also indicating a larger separation bubble.

The bottom surface results show that for the 95 km/h simulation, the transition occurs slightly behind mid-chord along the span of the wing. For the 160 km/h simulation, the transition has moved slightly forward. Again, the size of the separation bubble is significantly larger compared to the 95 km/h simulation due to the higher production of turbulent kinetic energy. At the tip of the wings and at the wing to fuselage fairing, the ?-Re? transition model predicts regions of laminar ?ow. This scenario is not believed to model the real ?ow conditions correctly and will be the subject of future investigations using a more refined mesh in these regions.

Also shown is the transition on the tail section and lower side of the elevator with 95 km/h to the left and 160 km/h to the right at zero angle of attack. The fuselage can be seen to play a significant role in causing transition at the leading edge of the lower part of the fin. There is less turbulence as we go up the fin and transition occurs later. The drag coefficient of the tail section is found to be Reynolds number

dependent and is 10% lower at 160 km/h compared to 90 km/h.
The right plot on the left page shows the comparison of the calculated speed polar for the Standard Cirrus compared to ?ight measurements from Ida?ieg. The simulations performed using the ?-Re? transition model compare well to the real ?ight data. For velocities below 100 km/h, the simulations closely match the ?ight measurements. At higher velocities, the sinking speeds are slightly under predicted. The measured best glide ratio for the Standard Cirrus from the Ida?ieg ?ight tests was found to be 36.51 at 94.47 km/h. From the simulations performed using the ?-Re? model, the best glide ratio is found to be 38.51 at 95 km/h. The fully turbulent simulations predict the best glide ratio to be 28.96 at 90 km/h. The large deviations in the prediction of the glide ratio when using fully turbulent simulations are caused by the absence of laminar air-?ow in the boundary layer of the glider.

Conclusion

It was found that STAR-CCM+ is able to predict the performance of the Standard Cirrus glider well. The simulations performed using the ?-Re? transition model
improved the results for the lift and drag predictions compared to the fully turbulent calculations for low angles of attack. For high angles of attack, i.e. stalled condition, the ?-Re? transition model was found unable to converge. By accounting for the extra drag due to the elevator de?ection needed to sustain steady level ?ight and including the interference drag from the cockpit edges, as well as the drag from the tail and wing tip skids, it is expected that the results will be further improved. Future studies should investigate the drag production from the glider in more detail and focus on applying the ?-Re? transition model for high angles of attack.

Industries:
Aerospace & Defense [2]
Aerospace - Technology [3]
Turbulence Modeling [4]
Products:
STAR-CCM+® [5]

Source URL: http://www.cd-adapco.com/case_study/modeling-performance-standard-cirrus-glider?page=0%2C1

Links:
[1] http://www.cd-adapco.com/case_study/modeling-performance-standard-cirrus-glider
[2] http://www.cd-adapco.com/industries/aerospace-defense
[3] http://www.cd-adapco.com/industries/aerospace-technology
[4] http://www.cd-adapco.com/industries/aerospace-defense/%E6%B9%8D%E6%B5%81%E5%BB%BA%E6%A8%A1
[5] http://www.cd-adapco.com/products/star-ccm%C2%AE