SHAPING UP

Mesh morphing reduces the time required to optimize an aircraft wing.

By **Marco Evangelos Biancolini,** Researcher, and **Ubaldo Cella,** Research Partner, University of Rome Tor Vergata, Rome, Italy
Giorgio Travostino, CFD Manager, and **Michele Mancini,** Aerodynamics Engineer, Piaggio Aero Industries, Genoa, Italy

Most business aircraft manufacturers use computational fluid dynamics (CFD) and wind tunnel testing to evaluate wing designs one iteration at a time. However, with this method, optimizing a design typically runs into many months — time that leading manufacturers try to reduce. To speed up the optimization process, Piaggio Aero Industries, S.p.A., teamed with researchers at the University of Rome Tor Vergata to validate a new design optimization method that generates a single mesh as a starting point and morphs it to any new geometry studied. The morpher tool RBF Morph allows engineers to change the locations of nodes in the computational mesh to alter wing shape, then it matches the surrounding surface and volume mesh to the new shape. Researchers applied ANSYS DesignXplorer to drive the morpher to map out the complete design space, identify the optimal design and validate its robustness in just a few weeks.

Piaggio Aero produces business, transport and military aircraft such as the P180 Avanti business plane, which is the fastest turboprop available for commercial use. In the past, the company's engineers had optimized wing efficiency for a new business aircraft using traditional design methods. The goal of the efficiency study was to maximize the lift-to-drag (L/D) ratio — the amount of lift generated by the wing divided by the drag it creates when moving through the air. A higher L/D leads directly to better fuel economy, climb performance and glide ratio. The wing's leading edge is divided into two parts, each with a different sweep angle (defined as the angle between the leading edge of the wing and a line perpendicular to the axis of the aircraft fuselage). A kink wing section is located between the two leading-edge parts to smooth out the discontinuity between them. The key design variables are the sweep angles of each leading-edge part. As these sweep angles are changed, the kink wing section must also be adjusted

MORPHER CHANGES MESH SHAPE

To validate the new method, University of Rome researchers began by using ANSYS ICEM CFD to produce a hex mesh of the wing with 14 million cells. The traditional approach involves solving the CFD model, evaluating the results, then modifying the geometry, recreating the mesh and running the modified wing geometry. The new method uses RBF Morph software — an add-on to the ANSYS Fluent CFD solver — to change the wing shape and surrounding computational mesh to solve a series of design points without having to manually create new geometry and mesh. Researchers defined the sweep angles of the two leading-edge sections as shape parameters that form the basis of the design space.

Engineers employed an advanced approach to carefully control wing shapes while maximizing the quality of the volume mesh after morphing. Surface morphing was used to control sweep angles by imposing a translation in the flight direction to the external wing tip while constraining the kink area. The region altered by the morpher was defined by a bounding box with constrained boundary. The morpher changed the locations of nodes in the computational mesh originally created using ICEM CFD to adjust the wing's shape and to match the surrounding surface and volume mesh to the new shape.

The morpher applies a series of radial basis functions (RBFs) to produce a solution for the mesh movement using source-point inputs and their displacements (two shape parameters in this case). Computationally, RBFs are very lightweight compared to storing all the meshes that are created. The new shape is applied within the solver without needing to reload the mesh, and the Fluent solver continues from a previously converged solution. The winning set of shape parameters can be selected and applied to the geometry source to morph the original model. The morpher incorporates a volume mesh smoother that preserves volume mesh quality during morphing. The morphing operation can be executed in a matter of seconds, even on very large meshes, by using the parallel-processing capabilities of high-performance computing (HPC) clusters.

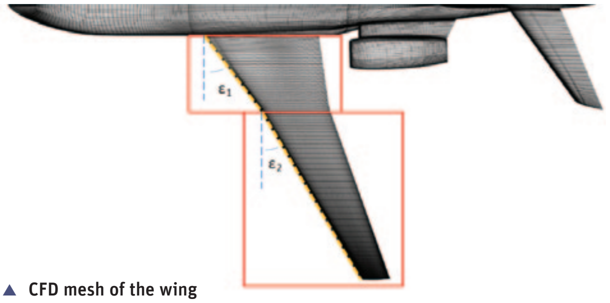
AUTOMATING THE CFD MODEL

The Fluent model was automated using the ANSYS DesignXplorer design optimizer to update its shape at initialization and its converged flow solution at baseline. The final solution data set from the baseline case was used to initialize simulation of each subsequent design point. Hence, the method significantly reduced the number of solver iterations needed for all design points past the first. The baseline model was iterated for about 1,500 iterations, while each subsequent run was resumed from previously converged solutions without memory loss, so they required only about 500 iterations for full convergence. Drag and lift were exported as output parameters. Design

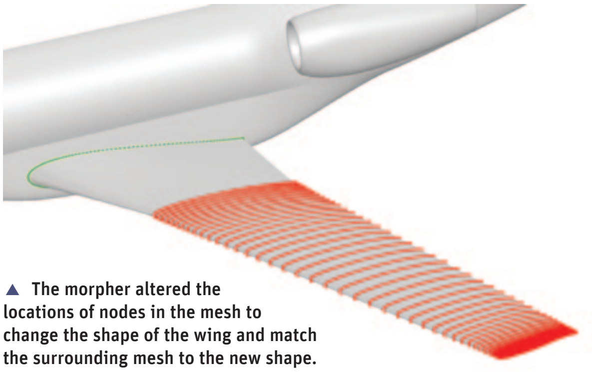
The morphing operation can be executed in a matter of seconds, even on very large meshes, by using the parallel-processing capabilities of high-performance computing clusters.

▲ Business aircraft wings were originally designed using a combination of CFD simulation and wind tunnel testing.

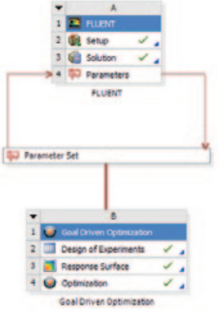
PHOTO COURTESY PIAGGIO AERO INDUSTRIES.



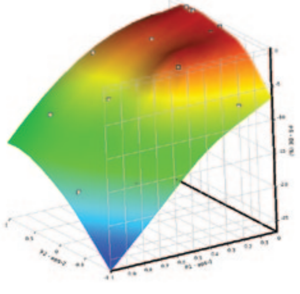
▲ CFD mesh of the wing shows two key design parameters: sweep angles of the two wing parts.



▲ The morpher altered the locations of nodes in the mesh to change the shape of the wing and match the surrounding mesh to the new shape.



▲ The automation platform triggered the morpher to morph the shape of the wing and to solve the new shape, then passed the results to ANSYS DesignXplorer.



▲ The response surface made it easy to visualize the wing's performance over the design space.

Table of Schematic B4: Optimization

	A	B	C	G
1		P1 - eps-1	P2 - eps-2	P6 - DE (%)
2	⊟ Optimization Domain			
3	Lower Bound	0	-1	
4	Upper Bound	1	1	
5	⊟ Optimization Objectives			
6	Objective	No Objective	No Objective	Maximize
7	Target Value			
8	Importance			Default
9	Constraint Handling			
10	⊟ Candidate Points			
11	Candidate A	0,0051281	0,52358	0,69639
12	Candidate B	0,0055937	0,57232	0,68803
13	Candidate C	0,0087079	0,54126	0,67251

▲ The MOGA algorithm within ANSYS DesignXplorer was used to simultaneously optimize lift and drag over the design space.

of experiments (DOE) identified a small sample of design points to represent the design space in such a way that when the aerodynamic performance was calculated at this small set of points, it could be interpolated with DesignXplorer to predict performance of any other design point within the design space with minimal error.

The ANSYS Workbench platform provided seamless interconnections for the interapplication data transfer, and DesignXplorer sequentially simulated all of the design points and collated outputs from the simulation. When the "update all design points" button was selected, the platform sent the first set of parameter values representing the first shape to the CFD solver. This triggered the morpher to morph the shape to match the new shape being explored. The CFD solver then solved this new shape and passed the output results, such as drag force, to DesignXplorer, where the results were stored. The automation platform then sent the next set of parameter values to the morpher. The process continued until CFD solutions for all the design points were completed.

GENERATION OF RESPONSE SURFACE

ANSYS DesignXplorer created a response surface using Kriging regression analysis from the collected data set of drag and other aerodynamic forces. The multi-objective genetic algorithm (MOGA) optimization method was used to define design goals and constraints, to create new design points, and to evaluate their performance based on the response surface. DesignXplorer offers direct and hybrid optimization methods. The response surface enabled the researchers to visualize the wing's aerodynamic performance over the entire design space and intuitively understand how output variables, such as drag, are dependent upon the chosen design parameters.

By evaluating the complete design space, the optimization showed that a number of points in the design space delivered a 1 percent improvement in L/D. The design optimization took a couple of weeks, less than a tenth of the time required to optimize the design using conventional methods. Solving the complete design space made it possible to evaluate robustness of the various design candidates. The designs that are most robust are those in regions of the response surface with the least slope. Researchers identified an optimal design that delivered consistent performance even as design parameters were varied over a 25 percent range. Piaggio plans to use these design optimization methods to reduce time to market and wind tunnel testing expenses for next-generation aircraft. ∧

The design optimization took a couple of weeks, less than a tenth of the time required to optimize the design using conventional methods.