Ready for Liftoff

Recent technology developments from ANSYS help aerospace engineers address pressing engineering challenges.

The global aerospace industry faces many critical challenges centered around engineering and technology. R&D teams are often tasked with balancing multiple, sometimes conflicting, priorities and developing new strategies to address these challenges: decreasing aircraft weight, reducing noise and emissions, and maintaining passenger comfort and security while keeping budgets in check. Whether they're working on innovative new engine designs, reshaping wings for better aerodynamics or exploring the use of composites materials in the fuselage, engineers are positioning this industry for a future in which all of these goals can be achieved. Just as the aerospace industry has advanced over the years, ANSYS software has evolved to anticipate new engineering problems. In this article, ANSYS experts share some recent technology innovations that benefit the global aerospace industry.

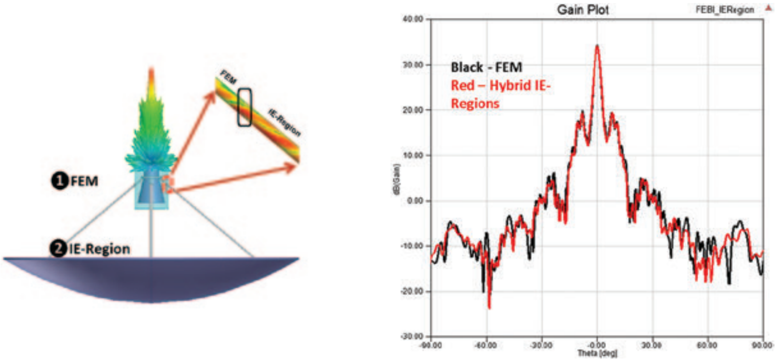
ANTENNA SYSTEMS: SIGNALING A NEW ERA

By Lawrence Williams, Director of Product Management, Electronics

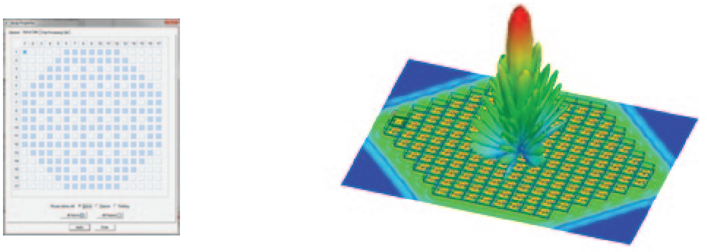
Antenna performance has always been a sophisticated engineering topic, but recent advances — including specialized active antennas, microwave circuits and devices, agile beam steering and shape, and digital space–time signal processing — have added even more complexity. In addition, advancements across the aerospace industry, including the growing use of composites materials in radomes and airframes, affect antenna performance in complicated ways.

For decades, ANSYS software has helped engineers to assemble 3-D antenna systems in a low-cost, low-risk virtual world and to predict system performance and electromagnetic effects with a high degree of confidence.

Recent technology advancements in ANSYS HFSS take advantage of high-performance computing resources and novel numerical methods, making it faster and easier to solve large problems. The software applies a domain decomposition method (DDM) to distribute



▲ New current-coupling capabilities enable modeling of an FEM region containing the feed antenna along with an IE region on the reflector's surface and feed-supporting struts that carry current from one region to another. Modeled radiation patterns are nearly identical to a traditional full-system FEM simulation.



▲ Complex radiation patterns of novel array shapes — including any missing elements — can be quickly identified using DDM computing techniques and a new array mask feature in ANSYS HFSS software.

large electromagnetic problems across a network of computers to solve in 3-D with higher fidelity. Material and geometry parametric sweeps, as well as solutions across frequencies, can be significantly accelerated via the multiple-design-point license.

ANSYS is a leader in developing hybrid solution techniques that accelerate solutions. As aerospace and defense engineers know well, some portions of an antenna problem are best solved by the finite element method (FEM), while other portions are best addressed using integral equation (IE) or physical optics (PO) methods.

For instance, once antenna performance has been optimized as a standalone system, the next step is to assess its performance when placed within a radome or on a vehicle. Hybrid solution techniques — combining FEM, IE and finite element boundary integral (FEBI) methods — enable antenna engineers to quickly and intuitively simulate the electric fields of antenna, radome and vehicle upon which it is mounted. Solving such large problems previously required very long and time-consuming engineering simulations.

New features in ANSYS 14.5 advance antenna modeling capabilities even further. An important solver enhancement for HFSS is the ability to current-couple FEM and IE regions. For a reflector antenna system measuring 50 wavelengths in diameter, the current-coupling method can speed solution time and reduce memory requirements by 81 percent — while modeling the system's radiation pattern with the same degree of accuracy as a traditional FEM simulation of the complete antenna system.

Another significant HFSS enhancement is finite-sized phased array analysis, using DDM and an array mask. With this capability, engineers can easily assemble an array by drawing a single antenna element, then applying an array mask to represent the array shape and the possibility of missing elements. The full array radiation pattern and near-fields can be calculated to examine edge effects. A novel composite excitation feature allows highly efficient solutions for user-defined array-weighting functions.

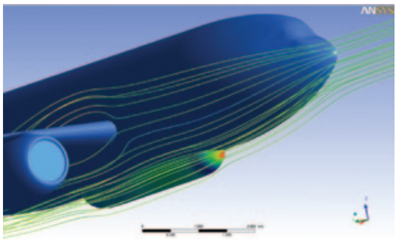
Thermal bidirectional links in ANSYS software make multiphysics studies faster and more intuitive than ever. This capability is especially important for applications such as dielectric resonator filters, which need to meet stringent design specifications, including those related to the operating environment and the device's power-handling capabilities. By linking HFSS and ANSYS Mechanical, engineers can confidently predict electrical performance under varying thermal and structural loads — a solution that seamlessly brings together electromechanical, thermal and structural analysis for the first time. A filter's bandpass frequency response can be quickly visualized as it shifts from low-power, ambient-temperature conditions to a high-power, thermal-deformation state.

By linking ANSYS HFSS and ANSYS Mechanical, engineers can confidently predict electrical performance under varying thermal and structural loads — a solution that seamlessly brings together electromechanical, thermal and structural analysis for the first time.

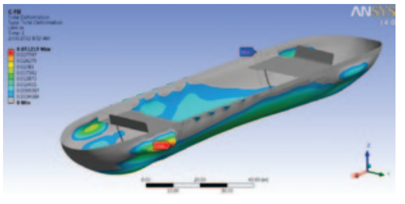
COMPOSITES MODELING FOR RADOMES: SHAPING A SOLUTION

By Sean M. Harvey, Senior Technical Services Engineer

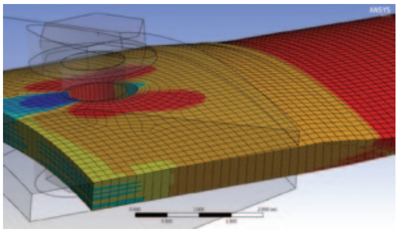
As concerns over fuel efficiency increase, lightweighting planes is an ongoing concern — and composites materials are an obvious solution. With their light weight, relatively low cost, electrical transparency, strength and structural stability, today's innovative composites are revolutionizing the aerospace and defense industry. For example, the latest generation of commercial aircraft from Boeing and Airbus are made up of over 50 percent composites materials.



▲ CFD fluid–structure interaction analysis reveals flow streamlines and pressure contours on a radome surface.



▲ Radome deformations resulting from airflow over the radome surface can be evaluated once whole-aircraft analysis has been conducted.



▲ Detailed solid composites mesh can be incorporated directly into mechanical assemblies and post-processed. Laminate details such as ply drop-offs and tapers are easily integrated into the model for analysis.

While composites offer many benefits, they present significant engineering challenges. Material layers must be stacked in different orientations, at varying thicknesses, to ensure structural stability while creating the complex, curving shapes that characterize aircraft. Perhaps nowhere is this challenge more apparent than in designing radomes, the curved weatherproof structures that house antennas.

With more than two decades of experience in modeling composites, ANSYS helps leading aircraft engineering teams to overcome the challenge of designing radomes and other composites structures. ANSYS Composite PrepPost (ACP), a module in ANSYS Workbench, enables engineers to import a radome model and perform ply stacking, draping and fiber orientation in an intuitive virtual design space. They can determine where composite layers should start and stop as well as design appropriate transitions between thick and thin material sections.

ACP also allows engineers to evaluate performance of a composites design, assess its structural strength, and identify potential regions of failure. By iterating this process, the team can easily optimize a design that thrives in real-world conditions. ACP is completely integrated into the Workbench platform, enabling aircraft engineers to change the radome's geometry and then automatically pass the new shape into the solver, eliminating intervention or rework.

New in ANSYS 14.5, solid composites geometries can be evaluated as solid 3-D mesh and integrated into ANSYS Mechanical solid assemblies within Workbench, enabling more accurate prediction of material stiffness and strength. This feature complements the existing shell representation capability, and it was designed with the very specific needs of aerospace and defense engineers in mind. New workflows in ANSYS 14.5 make composites design faster and more intuitive.

The ANSYS Mechanical suite enables parametric analysis for composites designs, delivering increased speed and insight for mechanical engineers. Teams can perform what-if analysis to quickly gauge the effects of design alterations — for example, changing the fiber orientation, thickness or ply drop-off locations, or even suppressing or including ply stacks parametrically.

As aircraft engineers use these features to make refinements, they can look at multiple design points, applying aerodynamic or inertial loads to assess material strength and displacement. They can replicate mechanical impacts created by real-world forces, such as bird strikes or hail, to ensure radome integrity. Engineers can also incorporate the effects of thermal changes on the design.

The integration and flexibility of the entire ANSYS suite allow radome engineers to apply tools such as ANSYS HFSS, industry-standard simulation software for 3-D full-wave electromagnetic field simulation. Used together, the suite helps to ensure the structural strength of a radome design as well as to verify that it delivers high signal-transmission performance — obviously a critical concern in the radome application.

ANSYS Composite PrepPost enables engineers to import a radome model and perform ply stacking, draping and fiber orientation in an intuitive virtual design space.

AIRCRAFT ENGINES: RETHINKING INDUSTRY STANDARDS

By Brad Hutchinson, Vice President Industry Marketing, Turbomachinery

Few technology areas receive as much attention and critical review as aerospace engines. As concerns over fuel burn, emissions and noise increase, aircraft engineers are rethinking every aspect of the traditional engine. ANSYS software is a key enabler of their efforts to develop cleaner, quieter, durable and more environmentally friendly designs that also fulfill critical safety and reliability promises.

Several recent developments in the ANSYS suite reflect emerging trends in aircraft engine design. For example, engineers are increasingly concerned with raising turbine entry temperatures to improve fuel efficiency. However, these rising temperatures push the limits of traditional materials and engine technologies — and necessitate innovative hot-section cooling strategies.

ANSYS Mechanical is invaluable for studying the structural strength and durability of promising new materials that may withstand higher combustion temperatures. Coupled with ANSYS Fluent or ANSYS CFX computational fluid dynamics (CFD) software, Mechanical helps engineering teams analyze the effectiveness of engine cooling systems via a multiphysics approach. New communication and file handling improvements in ANSYS 14.5 make it easier than ever to link fluid flows, heat transfer and other multiphysics phenomena via ANSYS Workbench. HPC-compatible features enable users to solve complex, numerically large cooling problems quickly, as well as to simulate hundreds of cases for studying all possible operating conditions.

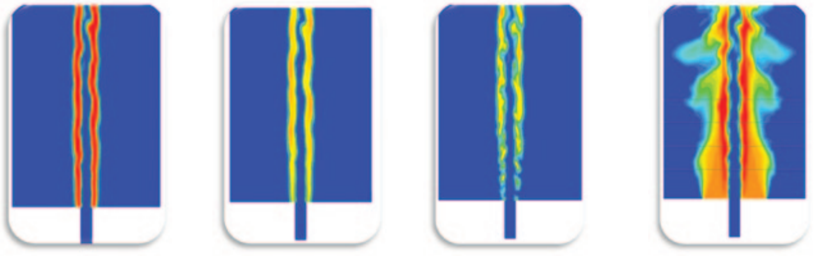
Today, aerospace engineers are rethinking even the most basic processes that power engines, including combustion. To support their efforts, ANSYS offers advanced combustion models, including thickened flame, improved spray and fuel evaporation models. These simulation capabilities help engine designers capture complex phenomena, such as fuel–air mixing, heat release and emissions, that have traditionally been difficult to replicate in the virtual world.

ANSYS software has the speed and power to more realistically simulate turbulence via scale-resolving simulation (SRS) methods, such as large- and detached-eddy simulation (LES and DES), which are better suited to model the complex behavior of combustors than traditional Reynolds–stress (RANS or URANS) models. The recent development of a novel scale-adaptive simulation (SAS) model provides a RANS solution in stable flow regions while resolving large-scale turbulence structures in regions where such phenomena are significant, such as bluff body wakes or free shear layers.

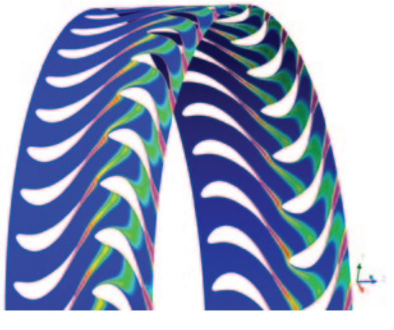
Turbine and compressor aerodynamics are at the heart of engine design. After many years of development, these components are highly evolved, so realizing additional performance gains is a challenging task. However, much of the analysis has been steady-state, for practical reasons. Developing additional insight requires applying unsteady solution methods, because that is the real nature of the flow in the many successive rows of blades that comprise a compressor or turbine. An impediment has been a lack of availability of efficient, HPC-enabled unsteady blade row methods. Fortunately, ANSYS has introduced and evolved powerful transient blade row (TBR) simulation methods, known as the transformation family of TBR methods. These methods address the issue of unequal pitch between adjacent blade rows, providing full-wheel solutions while simulating only one or a few blades per row. The approaches deliver tremendous savings in computational time and required disk storage space, yielding results files of a much more manageable size for post-processing.

One valuable enhancement in ANSYS 14.5 couples ANSYS Mechanical with the ANSYS CFX TBR transformation methods, streamlining aeromechanical analyses such as blade flutter and damping. Now the highest-fidelity aero and mechanical methods are linked and are more efficient to use than ever. This enables rapid, accurate investigation of aeromechanical phenomena critical to developing safe and durable engines.

TBR methods deliver tremendous savings in computational time and required disk storage space, yielding results files of a much more manageable size for post-processing.



▲ With ANSYS thickened flame models, the flame is dynamically thickened to limit thickening to the flame zone only. An efficiency function takes into account unresolved chemistry/turbulence interactions.



◄ TBR transformation methods in ANSYS CFX dramatically reduce computational time and resources, providing unsteady full-wheel blade row solutions — yet they require simulating only one or a few blades per row.

ADJOINT SOLVER: STREAMLINING CFD STUDIES

By Chris Hill, Principal Software Developer

Computational fluid dynamics analysis is foundational to the aerospace industry. From external component aerodynamics to the complex ventilation systems inside commercial jets, CFD studies are critical in many engineering tasks.

However, with so many complicated physical processes to consider, CFD simulations can be complex and numerically intensive. If many design iterations are needed to identify a design that meets requirements, then the overall computational cost can be very high. Ongoing improvements to the discrete adjoint solver in ANSYS Fluent continue to make these sophisticated problems easier and faster than ever to model and solve.

The adjoint CFD solver from ANSYS enables engineers to focus on one specific aspect of performance — for example, pressure drop across a system of ducts — and optimize design inputs based on that single performance measure. The adjoint solver accomplishes the remarkable feat of tracking the effect of changing hundreds of thousands of design variables simultaneously, via a single computation.

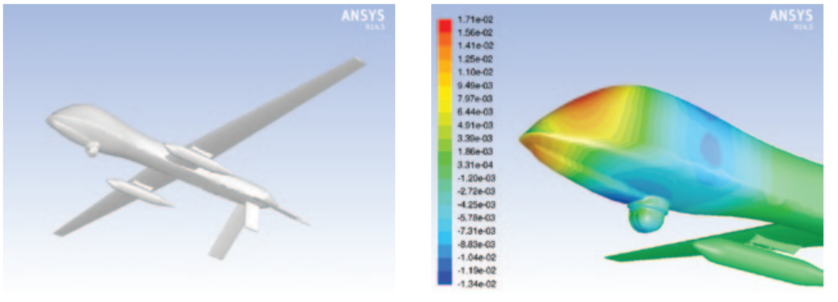
Fluent's adjoint solver significantly speeds overall simulation time by targeting those design areas that are most important in influencing drag, lift, pressure drop or other critical performance measures. Engineers can optimize designs quickly and systematically by homing in on the most influential parts of the system, instead of running a series of simulations and optimizing the geometry via trial and error.

Customers can use the adjoint solver in Fluent to redesign components such as heating, ventilation and air conditioning (HVAC) systems inside aircraft. The solver has the scale and fidelity to model the complexities of these piping and ductwork systems, which function as a life-support system for human occupants.

Engineering teams also can apply the adjoint solver to external aerodynamics problems, such as the drag effects caused by antennas, sensors and cameras mounted on the fuselage. An R&D team recently modeled an unmanned aerial vehicle to assess design changes that would increase its lift-to-drag ratio.

ANSYS continues to refine this revolutionary simulation capability with each new software release. New in Fluent 14.5 are improved workflows for setting up and post-processing simulations. Engineers can use this tool to manage many aspects of design performance, including flow uniformity, flow splits and variances for internal flows. Furthermore, they can optimize standard aerodynamic forces and moments, including lift-to-drag ratios, for external flows.

The adjoint CFD solver from ANSYS enables engineers to focus on one specific aspect of performance and optimize design inputs based on that single performance measure.



▲ Aircraft geometry (left): The contour plot shows displacement that should be applied to the aircraft surface to achieve a 10 percent improvement in lift/drag. This displacement is extracted directly from the adjoint solution data set generated in ANSYS Fluent (right).

TURBULENCE: STREAMLINED SOLUTIONS FOR A COMPLEX PROBLEM

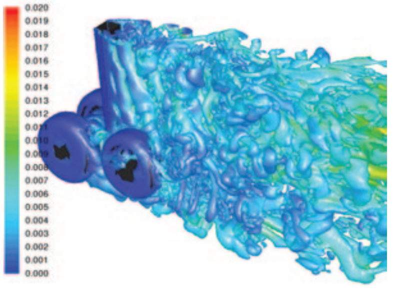
By Gilles Eggenspieler, Senior Fluid Product Line Manager

Turbulence modeling is critically important in many aerospace applications, as engineers seek to continually improve performance of their designs. Accurate prediction of a system's aerodynamics, heat transfer characteristics, mixing performance and other factors is key to determining performance with high precision — so accurate and robust CFD turbulent modeling capabilities are required for aerospace and defense applications, as well as in many other industries.

Today, engineering teams use CFD with advanced turbulence modeling to evaluate performance with maximum accuracy. Only highly accurate predictions deliver the performance improvements — which may be only a small fraction of a percentage — that differentiate good design from excellent design. Aerospace companies rely on accurate simulation modeling capabilities to gain a competitive edge. Steady-state or RANS models reduce the complexity of turbulent flow by averaging the velocity field, pressure, density and temperature over time. These models offer engineers a highly attractive solution to predict the effects of turbulence without having to explicitly capture all scales involved in turbulent flows — and they are very accurate for the vast majority of applications. This technology enables aircraft wing designers to predict factors that affect lift, drag and ultimate fuel efficiency — perhaps the industry's most pressing challenges.

Wing lift for aircraft is better predicted via capabilities that model separation and re-attachment of fluidic flows during flight. Understanding the upstream laminar boundary layers that transition into a turbulent flow is key to predicting wing lift or even compressor performance.

Some applications require more advanced unsteady models. Large-eddy simulation turbulence models resolve the large turbulent structure in both time and space and simulate only the influence of the smallest, nonresolved turbulence structures. These unsteady models help aircraft engineers to reduce the external noise generated by wheels and wings



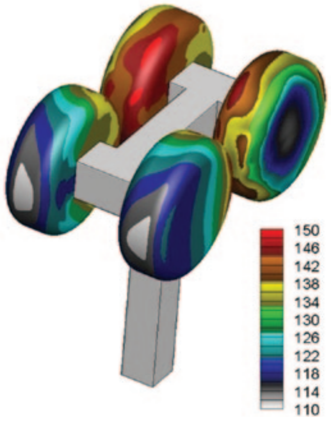
▲ Detached delayed-eddy simulation of flow over aircraft landing gear captures the turbulent flow structures created by the landing gear structure.

during takeoff and landing — a growing challenge as global noise regulations become more stringent.

ANSYS, with its team of leaders in innovative turbulence model development and application, continues to push the envelope in this highly complex but vital field. In parallel, ANSYS is making huge strides in high-performance computing technology that enables engineers to make appropriate trade-offs relevant to highly accurate results for complex simulations within an acceptable time frame.

Because high-definition LES simulation is often too time-consuming for fast-paced development cycles, ANSYS has combined the best of steady and LES approaches. These hybrid models deliver high-fidelity results for the right computational price. HPC acceleration ensures that results are delivered quickly, so engineers can evaluate many designs in a short time. Hybrid models, such as SAS and DES, use the power of RANS steady models to simulate flow in the vicinity of aircraft skin while unleashing the power of LES for the rest of the flow, in cases for which resolving large turbulent structures is critical to the study of aircraft noise or blade flutter.

As CFD applications become more complex, more sophisticated turbulence models are needed. Choosing the right turbulence model to match the application results in accuracy and optimized computational resources. ANSYS is a technology leader in this area, offering a wide range of the most advanced models.



▲ **Capturing the turbulent flow structures allows prediction of the sound pressure level created by airflow around the landing gear structure.**

Accurate and robust CFD turbulent modeling capabilities are required for aerospace and defense applications.

Reference

Menter, F. Turbulence Modeling for Engineering Flows, Technical Brief, ansys.com/Resource+Library/Technical+Briefs/Turbulence+Modeling+for+Engineering+Flows