ENGINEERING SIMULATION TAKES FLIGHT

Parker Aerospace uses ANSYS technology to reduce time and costs — as well as risk — in the design of aircraft systems.

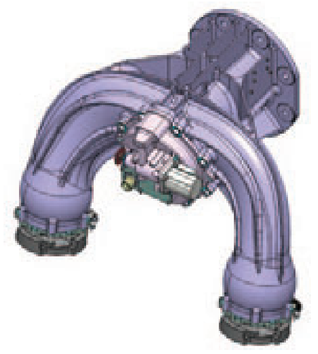
By Carsten Mehring, CFD Technology Lead, Parker Aerospace, Irvine, U.S.A.

As aircraft become more efficient, reliable and safe, they are also becoming more complex. Integrated systems design, architecting and systems control now take the spotlight in any new aircraft development project. Complex systems modeling requires accurate knowledge of subsystem and component performance. Although engineers can obtain component performance predictions using experiment, these test programs can be prohibitively expensive as the operating envelope continues to expand. The range of this operating envelope, characterized by the range of operating conditions (which include inlet pressures, flow rates, altitude and many others), continues to increase as companies extend the use of each aircraft component.

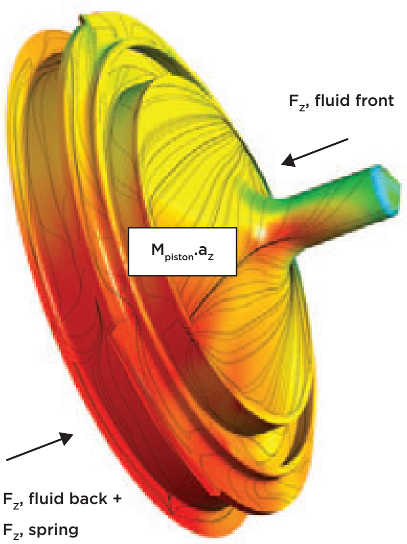
To analyze and evaluate design of components within a system and to support component test programs, the Central Engineering Group at Parker Aerospace uses ANSYS CFX for CFD and ANSYS Icepak for thermal packaging analyses. This helps the company to decrease development time and costs as well as to reduce risk.

Parker Aerospace is a global leader in flight control, hydraulic, fuel, fluid conveyance, thermal management, and engine systems and components used on virtually every commercial and military aircraft and aero-engine in production in the world today. The company's products are found on commercial transports, military fixed-wing planes, general aviation and business aircraft, helicopters, missiles, and unmanned aerial vehicles, as well as in other high-tech applications.

Simulation of Parker refuel coupling predicts pressure distribution at steady-state conditions and main piston dynamics based on transient FSI.



CAD model of refuel coupling



Streamlines and contour lines of static pressure on main piston

Contour lines of static pressure in two cross-sectional planes

Parker Aerospace uses ANSYS software to analyze and evaluate the design of components within a system, thereby reducing development time and cost as well as risk.

The Central Engineering Group at Parker Aerospace provides the company with broad expertise in a variety of engineering disciplines. The mechanical design team is responsible for delivering superior quality, cost- and time-effective analyses for aerospace systems and components. This includes providing premier analytical expertise by leveraging diverse problem-solving capabilities, including CFD and thermal packaging analysis using ANSYS tools.

AIRCRAFT/ENGINE FUELING

Parker's refuel coupling, used as a connection during refueling, might look like a part of a jet pack or Iron Man's suit, but, in reality, it controls the fueling process for one of the latest aircraft under development. This coupling, comprising two independent refuel lines that merge into one main port controlled by its own piston, is crucially important to meet aircraft safety as well as fuel fill and defuel time requirements.

Employing a mesh of approximately 12 million elements for numerous steady-state configurations (including piston settings), engineers used CFX to predict wall pressure distribution within the system to optimize the locations of pressure monitoring points. The system uses these points to control the closing and opening process of the piston in the main feed line. Experimentally measured overall pressure drop across the system agreed with the CFD results (within the limits of uncertainty associated with test data accuracy). Parker's CFD analysts are also deploying fluid–structure interaction (FSI) capabilities to tailor the dynamic behavior of the refuel coupling to specific customer requirements.

While full-blown FSI has its place within Parker's suite of modeling approaches, it does require considerable computational resources. To obtain quick turnaround and to conduct parametric design studies, the team often uses a combination of steady-state CFD analysis and 1-D simulation. If the system characteristic time scales allow, engineers take advantage of Parker's unique expertise in dynamic system modeling using SimuLink®, in which fluid dynamic

Analysts have successfully employed fluid dynamics to predict the performance characteristics of ejector pumps, especially their performance limits.

effects are incorporated into the 1-D simulation via lookup tables generated by steady-state CFD analyses.

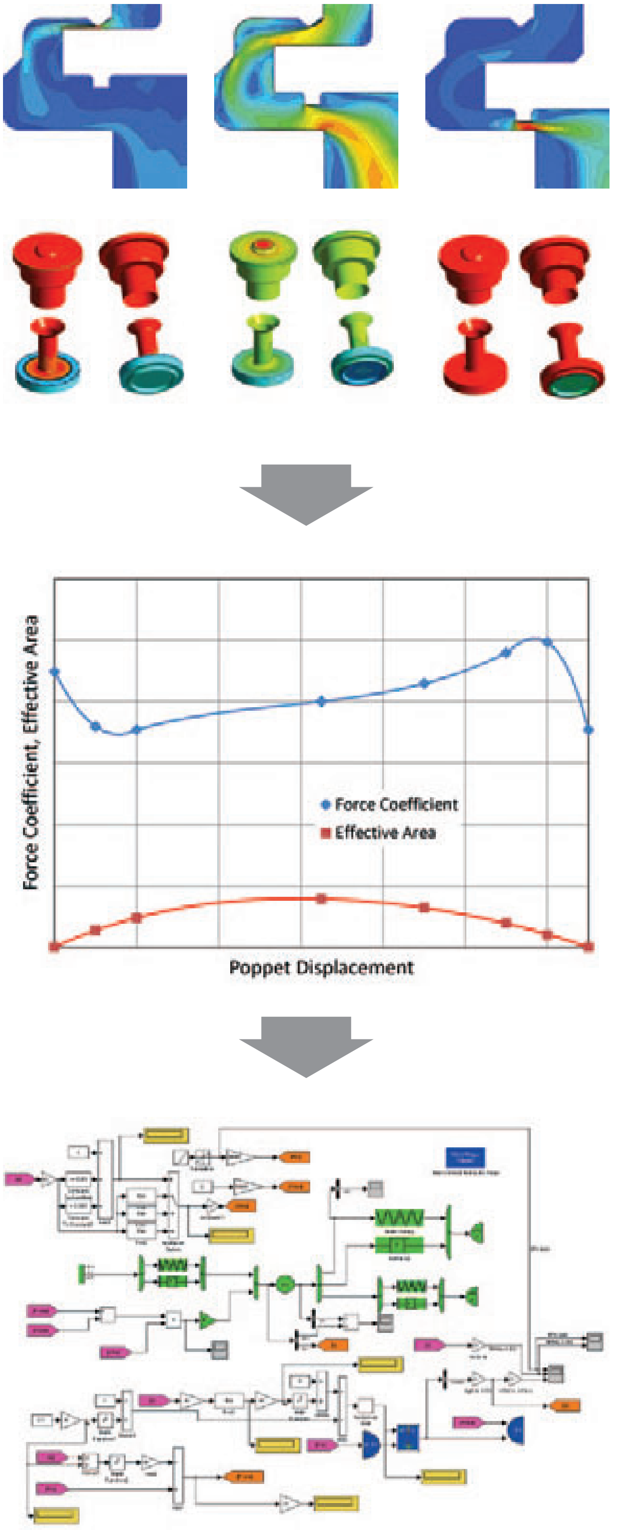
FUEL PUMPING

Aircraft fuel tanks are often housed in the wings within multiple compartments; the tanks contain baffles to prevent sloshing. Accessing this fuel requires the use of specialized pumps to feed the main fuel pump(s). Scavenging pumps, usually ejector pumps, are commonly used to collect fuel from remote corners or the bottom of fuel tanks and to discharge that fuel at the inlet to the main fuel feed pump(s). Fuel scavenged from the bottom of fuel tanks is often contaminated with considerable amounts of water, which was originally dissolved within the fuel or entered the fuel tank through condensation. Another important function of ejector pumps is to disperse that water into fine drops so that the engine can safely consume the fuel, with these finely dispersed water droplets, without any impact on performance.

Traditional ejector pumps do not have any moving parts and are highly reliable if used within their operating range. However, the operating range of fuel ejector pumps is limited by the onset of cavitation. Cavitation limits the pump's operating range, so the ability to identify this range early in the design process is important, even before building development hardware. Analysts at Parker Aerospace have successfully employed computational fluid dynamics to predict performance characteristics of ejector pumps — especially their performance limits — by utilizing the Rayleigh–Plesset cavitation model within CFX. The team is currently working to improve the existing cavitation model via customized scripts. Areas of focus include considering shear-induced cavitation effects and cavitation in multifluid ejector systems.

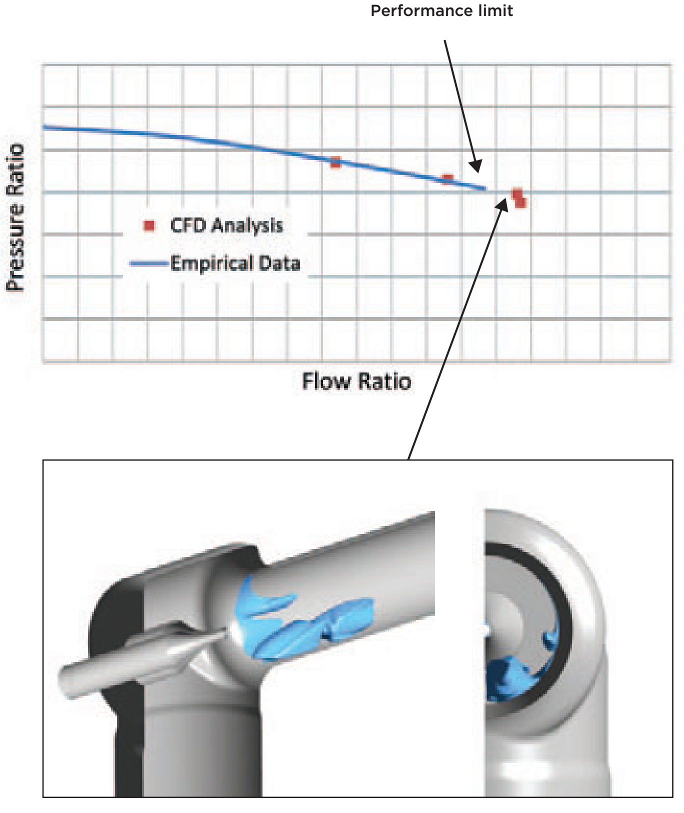
FUEL-TANK INERTING

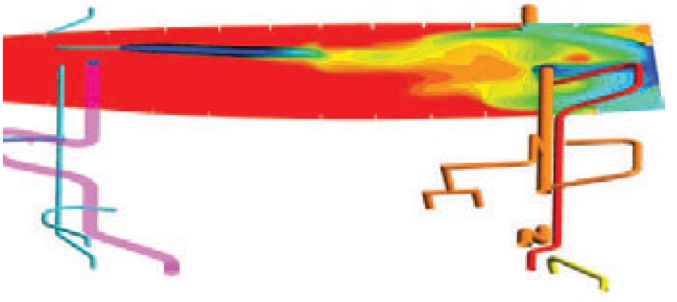
To prevent fuel tank explosion, fuel-tank inerting systems are deployed on commercial (and military) aircraft. One specific type of inerting system delivers nitrogen-rich (oxygen-deprived) air into the ullage (non–fuel-filled) space of the aircraft wing and center fuel tanks to reduce the likelihood of flammable fuel/air mixture formation, thus reducing the chance for ignition and fuel tank explosion. To demonstrate proper operation of the inerting system over the entire aircraft operating envelope, Parker engineers employ a comprehensive lumped parameter model of the tanks, vent lines and air flow within them in



Engineers at Parker combine steady-state CFD analyses and their unique lower-dimension simulation capabilities to analyze system dynamics of hydraulic valves. The example shows a fuel reprime valve. CFD analysis (top) generates lookup tables for force coefficient and effective area as a function of poppet position (center). These tables are later used within a 1-D simulation tool to predict overall system dynamics.

Comparison between jet pump performance prediction using ANSYS CFX and experimental data including onset of cavitation. Upon inception of cavitation, the cavitation region quickly expands as pressure ratio decreases. Iso-surface of vapor-phase volume fraction (=0.5) at the onset of cavitation is shown in blue.





Result from wing tank ullage space CFD analysis showing instantaneous oxygen concentration contour lines in a "spar" plane through the center of a nozzle injecting nitrogen-enriched air into ullage space for inerting purposes.

conjunction with a Monte Carlo method. This method generates statistical flammability exposure times (the time during which a spark would ignite a flame) for a range of unknown operational parameters that are bounded by known distribution functions and for a large number of theoretical flights (taking into account, for example, circumstances such as plane climb rate and weather conditions).

To validate and improve the prescribed lumped parameter model, Parker analysts conduct detailed CFD analyses for a number of characteristic flight sequences. The detailed information obtained from the CFD analyses is then utilized for benchmarking the lower-dimension lumped parameter model. Despite the large amount of computational resources required to conduct the CFD analyses, the cost of those resources is still minute compared to the expenses of a full-scale ground or flight test. While flight data is available at some stages of the development process, early CFD analysis provides an opportunity to deploy design improvements that cannot be determined by the lumped parameter model: for example, location and direction of nitrogen-enriched air (NEA) jets and location of air vents. In addition, detailed CFD analyses of the mixing within the ullage space and mass transfer between the various fuel tank compartments provides the opportunity to improve upon the lumped parameter model and, consequently, to improve predictions of flammability exposure times, all leading to a safer aircraft design.

CONTROL ELECTRONICS

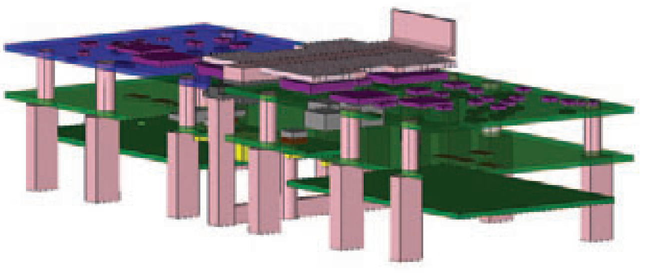
With the development of the more electric aircraft, the number of electronic control units on board increases, while the space available for housing those components does not (a result of efforts to maximize cargo or passenger space). Additionally, increasing usage of composite materials in aircraft design reduces the opportunity to utilize the airframe surrounding electronics boxes as heat sinks. At the same time, energy spent to cool electronics components should be as low as possible to reduce the impact on overall system efficiencies. Because of these factors, electric components run hotter than ever before, and accurate predictive tools are needed to guarantee their proper function even before in-flight testing commences. At Parker, analysts extensively use ANSYS Icepak to fulfill those needs. Using this tool, engineers are able to gather information on temperature levels early in the design process to assure that electronic components are operable both under cold-start conditions (meeting warm-up requirements) and in continuous operation under hot ambient conditions. The analyses conducted at Parker span a broad spectrum, varying in the level of fidelity by which the components are modeled (ranging from very rough models with components lumped together to more detailed approaches verifying that maximum component junction temperatures are not exceeded). Analyses also vary in the type of heat transfer used to dissipate the majority of energy away from the electronics components (forced convection via fans and cooling channels or conduction-dominated heat transfer through PCB boards and casings that are enhanced by board vias and board/casing conductive bracings, for example).

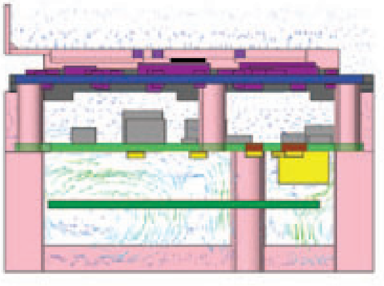
KNOWLEDGE COLLECTION

A wealth of other Parker products have benefited from analyses using ANSYS tools, including fuel system solenoid banks, liquid ring pumps, pneumatic butterfly valves, centrifugal fuel and gear pumps, and electronics cooling plates. Similar to the path FEA analysis took a couple of decades ago, CFD has matured to become an integral part of the product design process at Parker Aerospace, reducing development time and cost while achieving optimum product performance. The wealth of information obtained from CFD analysis provides detailed insight into systems behavior and facilitates generating knowledge and correlation databases, thereby reducing development risk, cost and time.

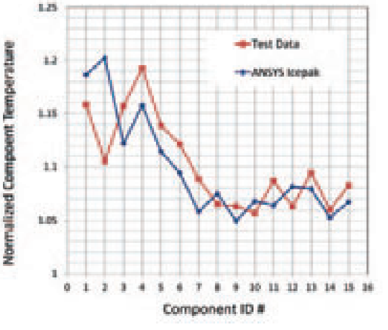
ANSYS Icepak predicts buoyancy-driven flow and PCB board surface temperature distribution within an electronics box at specific ambient and operating conditions. Comparison with test data identified faulty thermocouple measurement on a component.

Multiboard electronics control unit

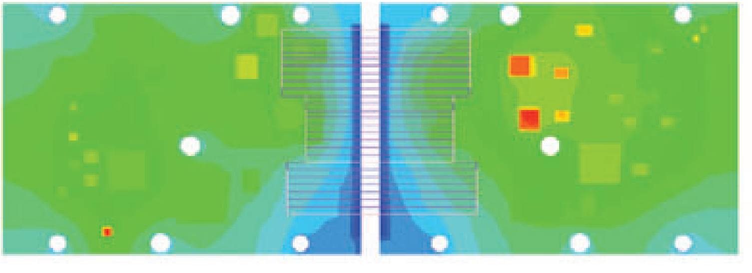




Buoyancy-driven flow between PCB boards



Model benchmark using available test data



Board temperature contour lines on one side of PCB board

The wealth of information obtained from CFD analysis provides detailed insight into systems behavior and facilitates generating knowledge and correlation databases.