STAR-CCM+ Delivers “POWER with ease” to the Aerospace Industry

STAR-CCM+ has the aerospace industry in its genes.

Seven years ago, when we first released STAR-CCM+ - then an ambitious new simulation tool - one of our primary objectives was to fully support Aerospace and Defense applications. This was evident with the foundation of an excellent and innovative density-based (coupled) solver that is ideally suited for aerospace problems (from subsonic to hypersonic). The business reason behind CD-adapco’s commitment to Aerospace & Defense was the recognition of a gap in the market for a powerful yet intuitive simulation tool capable of addressing the full gamut of aerospace engineering challenges.

Needless to say, those early versions of STAR-CCM+ were an instant hit with the aerospace community, delivering both validated accuracy and ease-of-use. Released three times a year, each new version of STAR-CCM+ rewarded this loyalty by introducing new features and capabilities specifically targeted towards aerospace applications.

STAR-CCM+ has grown up. More than just a CFD code, STAR-CCM+ is now a fully integrated platform for multi-physics engineering simulation, spanning many disciplines and areas of application. From a single environment engineers can now include combustion; multiphase flow; heat transfer through solids and fluids; dynamic fluid body interaction; ice management, acoustics and solid stress capabilities.

The Aerospace Special Report is the perfect illustration of this, providing a selection of case studies and examples that demonstrate the tangible benefits that aerospace companies have achieved by deploying STAR-CCM+ as a principal tool in their engineering arsenal.

Please take time to review the report, these applications merely scratch the surface of the work we are currently doing with major aerospace and defense companies around the world.

I welcome you to contact us directly if you have questions on a current project or the potential outcome you could expect from the application of our software and/or services organization.

David L. Vaughn
VP Aerospace & Defense
CD-adapco

EMAIL dvaughn@us.cd-adapco.com

STAR-CCM+ V5
The STAR European Conference 2010 is the official release event for STAR-CCM+V5, which will introduce many productivity & automation enhancements, including a revolutionary new 3D Geometry Modeler.

www.cd-adapco.com/powerwithease

STAR-CCM+ delivers the entire engineering simulation process in a single integrated software environment.

This unique approach brings unrivalled ease-of-use and automation to CAD preparation, meshing, model set-up and iterative design studies, enabling your Engineers to deliver better results, faster.
Environmental engineers in the Aerospace industry continue to adopt STAR-CCM+ at an unprecedented rate. Since its launch, STAR-CCM+ has consistently grown in popularity in every facet of the Aerospace Industry. From UAVs to mega-airliners and from avionics cooling to hypersonic re-entry, organizations and companies such as NASA, Gulfstream Aerospace, Lockheed Martin, Airbus, Raytheon, MBDA, and many others have standardized on CD-adapco engineering simulation for a variety of applications.

David L. Vaughn, CD-adapco

So what makes STAR-CCM+ (or CD-adapco) so special? Why are so many companies switching from the competition or supplementing their testing efforts with engineering simulation for the first time? The interesting thing about the answer(s) to these questions is the fact that every user has different reasons which either relate to their specific business or to their established engineering practices. However, when you weave all the responses together, a pattern becomes obvious. The vast majority of CD-adapco customer remarks on STAR-CCM+ are related to accuracy, flexibility, productivity, and/or support. And, in fact, it was upon these “4 pillars” that CD-adapco’s commitment to the Aerospace industry was founded. One of the principle reasons for launching STAR-CCM+ was a commitment by CD-adapco’s founders to “...leverage our experience and expertise in order to provide solutions that have the flexibility to accurately solve the Aerospace industry’s broad range of flow, thermal, and stress problems with unprecedented efficiency.”

Accuracy is the first requirement for any engineering software used in the Aerospace Industry. In the past, adoption of commercial CFD software by aerospace engineers was typically hindered (at best) by this requirement. Early in the design of the STAR-CCM+ software architecture, CD-adapco recognized and fully appreciated the need for a variety of validated methods, models and technology in order to meet expectations of the aerospace community.

The most obvious manifestation of this software design goal is in the selection of numerical solvers/schemes provided in STAR-CCM+. The developers of STAR-CCM+ recognized that one size does not fit all, e.g. a pressure-based segregated solver could never satisfy the accuracy requirements for transonic flow applications. So they provided both a segregated (pressure-based) solver AND a coupled (density-based) solver. And more recently the AUUM+ differencing scheme was added as an option to the coupled solver to extend its applicability to the high hypersonic flow...
regime. It’s really OK if you don’t know what this means in terms of equations and algorithms. The point is that different numerical schemes are a better fit for different cases. And in order to provide full support for all facets of the industry, multiple numerical solvers and schemes are required.

If you’re going to make a commitment to provide accurate solutions, then you’d better be committed to validation, and CD-adapco recognizes the necessity of multiple levels of validation. Two very good examples are the Drag Prediction Workshop (DPW) and the Lockheed Martin JCM validation study.

The Drag Prediction Workshop is sponsored by the AIAA. The study shown here was completed for the DPW 3, but for validation we are showing the DLR-F6 wing-body configuration that was also the subject of DPW 2. The results shown here are for the “fine” mesh which consists of ~21 million cells including ~25 prism layers that provide a $y^+ \leq 1$ throughout the mesh. One key point for this study is that the mesh density was more optimized for lower angles of attack - specifically in the wake of the wing. The separation at the side of body can be seen in figure 1. This phenomena was well documented in the experimental results. The resulting drag polar for the fine mesh is shown in figure 2. As you see, the results match experiment quite well.

When Lockheed Martin Missiles and Fire Control began investigating STAR-CCM+, their engineers worked with CD-adapco on a validation study of one of their full missile configurations. Every detail of the geometry was modeled as seen in the mesh images of figure 3. Also obvious in that image is the fact that this study employed wall functions since the boundary layer mesh only included 4-6 layers of prisms which resulted in $y^+$ of 30-60. Drag polars were calculated at 3 Mach numbers (0.5, 0.75, and 1.25). The resulting data is shown in figure 4. These validations (along with countless others) show that accuracy on aerospace applications is not a problem for STAR-CCM+.

PRODUCTIVITY is the goal of every engineering organization. Efficiencies in labor and computational resources are usually at the top of the list for anyone evaluating engineering simulation software, and it’s exactly what companies like Goodrich Aerostructures and Raytheon Space found when they tried STAR-CCM+ on their applications. The Aerospace Industry realizes that the objective of productivity must be applied to the entire CFD process. In short, it’s about quick turnaround from CAD import to post-processed reporting. The first productivity gains that most aerospace users recognize is in pre-processing and meshing. CD-adapco’s surface wrapping and polyhedral meshing technology is allowing applications that could never before be accomplished in a practical manner. A good example is thermal management. Managing the thermal environment is critical to many applications like avionics cooling problems, and most of thermal management cases involve extremely detailed geometric complexities. The STAR-CCM+ surface wrapper makes short work of “shrink-wrapping” the boundaries of a domain to provide a consistent manifold boundary for the computational domain - even if the imported CAD data contains gross inadequacies. Together with automated polyhedral meshing, surface wrapping makes it possible to analyze the thermal environment between a jet engine and the surrounding nacelle - a domain that is extremely complex and consists of hundreds of CAD parts.

FLEXIBILITY is the key to fully supporting an entire industry. Every company has a different organization and/or process that employs engineering →
simulation tools like STAR-CCM+. To satisfy those diverse needs a software must be flexible - like using it from within the CAD environment (the STAR-CAD front-end for STAR-CCM+) or importing any number of different CAD formats including native CAD files.

So you want flexibility in meshing? The polyhedral structure of STAR-CCM+ makes it independent of cell topology, so legacy and archived meshes can be solved. When generating a mesh from within STAR-CCM+ it’s your choice: traditional tetrahedral, advanced hexahedral, or polyhedral cells are all supported. Additionally, aerospace users can choose to generate meshes with or without an automated prism layer.

But it’s flexibility in the selection of physics models that really delivers the power of STAR-CCM+ across the Aerospace Industry. One of the more recent capabilities in STAR-CCM+ is the harmonic balance method which allows unsteady flows to be solved in the frequency domain. The ability to model chemically reacting gas mixtures in rocket plumes, to model sloshing liquids in fuel tanks, to model combustion within solid rocket motors, to model evaporating films on leading edges - these are just a few of the physics models that STAR-CCM+ users at NASA, MBDA, Liebherr Aerospace and many others are employing daily.

CD-adapco’s support organization has always been one of the strengths of the company. Customer support is at the center of every activity, and again this is a primary reason for the migration to STAR-CCM+ within the Aerospace Industry. This industry is very demanding, and it requires an experienced and accessible support team to address its needs.

The first experience a new aerospace customer has with CD-adapco support is usually through training. The breadth of training offerings is a significant advantage in meeting the wide range of needs from the Aerospace community. Regular training courses are offered throughout the world at CD-adapco offices, but on-site and web-based training is also available. The course selection provides a wide variety of topics from aerodynamics to heat transfer to fluid-structure interaction. There is even an option for customized training where a course can be designed to fit the specific needs of the customer, and will be delivered by a trainer experienced in the specified topic.

Training is only the beginning. CD-adapco provides every customer with a dedicated support engineer. This insures that the support team becomes familiar with the day-to-day requirements of their customers. And as one the world’s largest employers of CFD and CAE expertise, CD-adapco provides access to a collection of the world’s leading experts. Over 50% of the dedicated aerospace team is educated to PhD level, and the vast majority of them have worked for major aerospace companies from around the world.

Michael Guteres manages an engineering team at AAI, makers of the very successful Shadow UAV. Dr. Guteres has said “Our dedicated CD-adapco support engineer is always there ready to help. And when he can’t solve the problem, he immediately puts us in touch with the right expert to help us with our wide array of simulations.”

So again the question is why are so many aerospace companies choosing CD-adapco? Well, the short answer is accuracy, productivity, flexibility, and support/experience. For a more detailed answer just look around - it’s likely that you already know someone in the industry who is reaping the benefits of STAR-CCM+.

Over the past years, we have used STAR-CCM+ to predict the aerodynamic performance of both commercial and military aircraft.

In particular, we ran cases from the 2nd Drag Prediction Workshop and obtained excellent results compared with experiment. STAR-CCM+ yields accuracy at least as good as existing in-house codes and other commercial codes.

Matt Milne - QinetiQ

"
Whether it is lean manufacturing, six sigma, or some combination, quality improvement is now a primary focus of virtually every manufacturer in the aerospace industry. And now that the waste is squeezed out and quality is designed into manufacturing processes, visionary corporations are looking upstream toward the engineering of the product for further gains in quality.

One of the keys to ensuring manufacturability and efficient integration of component systems is the implementation of high fidelity Computer Aided Engineering (CAE) software during the early design stages.

One of the more popular methods for addressing quality improvement during the design process is known as Design for Six Sigma (DFSS). Like most quality improvement acronyms this process has different meanings for different organizations but, from a "big picture" perspective, DFSS simply means designing products that meet the customers’ requirements while maximizing the efficiency and robustness of the manufacturing and/or integration process. With modern production being centered on CAD/PLM data, it is imperative that engineering simulations, which predict the product performance, be tied directly to the same CAD/PLM data to ensure consistent quality. This also allows a more direct use of DFSS statistical tools for measuring quality.

CD-adapco realized the benefits of upfront computational simulation and were the first to bring to market a true CAD embedded computational fluid dynamics (CFD) solution. The STAR-CAD Series of products embed state-of-the-art CFD technology directly in all the major CAD systems which allows engineers to perform full CFD simulations (pre-processing, meshing, and analysis) from within the CAD system.

In the past the product definition was simply tossed over the wall for detailed design and manufacturing. If production constraints were considered in the early design stages, they were typically considered as "rule-of-thumb." This lead to major headaches and unexpected expenditures once production actually began. More recently CAD/PLM systems have helped, but there is often still a wall between engineering analysts and designers, which prevent the direct inclusion of manufacturing constraints in the →
The STAR-CAD Series

product design cycle. Essentially analysts and designers must perform wasteful iterations to relate manufacturing constraints with performance goals. Now with CAD embedded simulation, manufacturing constraints can be considered directly when engineering analysts perform design studies. More often these studies are being executed as multidisciplinary design optimizations (MDO), which allow a six sigma approach to the product design. With direct access to parameterized CAD models, these MDO studies are much easier to setup, and manufacturing constraints are included in the formulation of the optimization problem. This results in the seamless inclusion of manufacturing requirements from the very inception of the product design. Of course the benefits of DFSS via CAD embedded analyses extends beyond the initial manufacture of the product. Component reliability is a major contributor to the total lifecycle cost, and nowhere is this truer than in the aerospace industry. Let’s look at the simple example of a flap actuation system. Using a CAD embedded CFD product such as STAR-CAT5 the static aerodynamic loads on the actuation system are easily obtained early in the design process. But perhaps of more value in terms of lifecycle costs is the early consideration of dynamic loads throughout the range of flap motion by integrating embedded CFD, FEA and multi-body dynamics simulations into a multidisciplinary design optimization. The entire actuation and flap track system can be optimized for lifecycle costs during early engineering stages. From the CAE perspective, there are many benefits associated with the STAR-CAD Series of CAD embedded CFD tools e.g., ease of use, superior accuracy, efficient numerical algorithms, etc. None of these, however, affect the bottom line more directly than the enabling of DFSS. CD-adapco refers to the STAR-CAD Series of products as “up front” solutions meaning that they bridge the gap between CAD and CAE analysis. Looking at the bottom line, however, perhaps the more meaningful metaphor is the gateway between quality improvement and upstream engineering design. It may be cliché to say that CD-adapco is breaking down barriers in the development and use of CAE technology, but the truth is - they've been doing it for over 25 years.
In full-scale fatigue testing of large aircraft, detailed information about the individual test apparatus can be used to optimize operation: saving both time and money! Such in-depth analyses can be dealt with easily and quickly using CFD, yielding much greater detail and understanding than from an experimental investigation alone.

Here, STAR-Design was used to predict the pressure losses in the hydraulic system used to exert large forces on the outer wing, causing the wing to displace and deform (replicating a specified operating condition). The results were compared against experiment and very accurate agreement was achieved.

**Testing environment**

In order to apply the loads to the wings, a complex hydraulic system with a large number of hydraulic cylinders is required. During normal test operation, the hydraulic cylinders are controlled by servo valves, which are part of the overall control system. During switch off operation, only passive valves control the hydraulic flow and consequently the hydraulic pressure. These have to be individually adjusted during commissioning, therefore obtaining detailed information of the pressure loss between the chambers on both sides of the actuator piston. How to adjust the valves is critical.

**CFD simulations**

The geometry of the hydraulic channels at the actuator, the valve manifold and the valve itself were designed using STAR-Design. The volume mesh, consisting of approximately 1 million cells, was obtained by →
triangulating the surface in pro-STAR’s surface meshing module and generating a trimmed mesh, with three extruded layers, in pro-STAR. A section of the mesh and a pressure distribution can be seen in Figure 1. In order to obtain an overview, a steady-state case was analyzed first. The flow was assumed to be steady, incompressible and turbulent. The standard k-ε model was chosen to model turbulence. The fluid under consideration was a special hydraulic fluid. A normal operation temperature of 55°C and the corresponding viscosity of the fluid were used. A typical flow field is shown in Figure 2. It is characterized by the strong production of swirl at the valve, while at other positions of interest in the fluid channel, the flow is only slightly influenced by turbulence effects. This means that the adjustable valve is the dominant cause of the pressure drop in the oil flow.

Comparison with experiments
Experimental investigations using the same setup have been performed in order to validate the CFD simulations. The pressure losses were measured at different flow rates and orifice diameters. The agreement between calculation and measurement is very good (Figure 3). Though the relation between flow rate and pressure loss seems to be almost quadratic, it was essential to understand the parameters that describe this relationship and to evaluate the situation at very low flow. As the main result, the effects of the flow of the hydraulic fluid during connection of both chambers of a loaded hydraulic actuator were understood to a much higher degree than it was possible only by experimental investigations. The effect of geometry alterations can be understood by numerical evaluation much faster than by experiment, which helps to act faster and reduce cost.

Conclusions
A very accurate prediction of the pressure losses caused by a geometrically complex hydraulic fluid channel including an adjustable valve can be achieved using STAR-CD. It was demonstrated that the combination of a numerical investigation with experimental validation saved time and money.
Safer UAV Landings

Dr. Ulf Specht, IABG mbH, Germany.

UAVs have been used in a reconnaissance & intelligence-gathering role since the '50s & more challenging roles are envisioned to the future - including combat missions. Since 1964 the US Defense Department has developed 11 different UAVs, though due to acquisition & development problems only three have entered production.

The US Navy has also studied the feasibility of operating Vertical Take-Off and Landing (VTOL) UAV’s since the early '60s, the QH-50 Gyrodyne torpedo-delivery drone being an early example. However, high cost and technological immaturity have precluded acquiring & fielding any operational VTOL UAV systems.

ائق FACTS

QH-50
Performing landing operations of a helicopter or an UAV (unmanned air vehicle) on a helideck of a battle ship is a critical situation for both the helicopter pilot and the crew of the ship. A detailed understanding of the flow structures and the magnitude of the turbulent fluctuations is therefore necessary. Taking into consideration the very detailed superstructure of the complete ship, STAR-CD can be used, as it is in this case, to predict turbulence fields.

Background
Reconnaissance is one of the main tasks of the German Navy. An UAV will ensure identification of objects beyond the horizon of the ship even under insufficient optical conditions. Since the landing operation should be a completely automated task, the software of the control system has to take into account anything that might affect the smooth landing of the UAV, such as the environmental conditions. Therefore it is necessary to predict the flow field around the ship, by considering different wind speeds and wind directions relative to the ship.

CFD simulations
Starting from the water surface level, a narrow box was discretized using ICEM/Tetra containing the complete ship. This included the complicated structure with a refined region near the helideck. The mesh was completed by adding blocks of hexahedral cells in front of the bow, behind the stern, portside, starboard, and using STAR-CD’s ‘arbitrary couples’ methodology above the tetrahedral cells. The final mesh consisted of approximately 2 million cells. The flow was assumed to be steady, incompressible and turbulent. Turbulence is modeled by the standard high Reynolds $k-\varepsilon$ model.

Results
A general view of the flow field containing isosurfaces of the turbulent kinetic energy is shown in Figure 1. There is a remarkable production of the turbulent kinetic energy due to the superstructure. Since most of the turbulent kinetic energy has been dissipated before reaching the helideck these turbulent fields hardly affect the situation there. A large vortex generated by the main flow at the end of the hangar (similar to a backward-facing step) however, has a significant impact on the landing procedure (Figure 2). This vortex interacts with the main flow generating a shear layer and producing a region of high turbulent kinetic energy. Simulating many different flow directions, these computations can provide a “Best Practice Guide” on how to maneuver the ship prior to a landing operation. Furthermore the data sets of the velocities and the turbulent kinetic energy can be prepared as an input for a real-time UAV approach and touch-down simulation.

Conclusions
The detailed information of complex flow patterns obtained by STAR-CD simulations improves significantly the understanding of how the turbulent fields are generated. Furthermore the complete flow field, which is impossible to achieve in experiments, can be used as an input for an UAV simulation environment.
The Pro Observer Unmanned Aerial Vehicle (UAV) sounds like something straight from the pages of a James Bond story: a miniature aircraft, weighing less than a bag of sugar, that can be stored in a suitcase, ready to be launched on a “spying” mission within minutes of being unpacked.

Proving that truth is sometimes stranger than fiction, the Pro Observer is now in flight-testing, after having been designed extensively using CFD. It seems somehow fitting that this seemingly futuristic technology is designed with the aid of the world’s most futuristic CFD code: STAR-CCM+.

Although it is easy to associate UAVs with the covert world of international espionage, the truth is somewhat less glamorous. UAVs are really designed for work that is too “dull, dirty or dangerous” for the pilot of a manned aircraft. Put simply, for this type of mission, a pilot is simply a superseded component, representing an unnecessary burden on the aircraft. A manned aircraft must carry not only the pilot, but between two and five times their weight in equipment required to support them. Critical systems need also to have multiple levels of built in redundancy, which are required to bring the pilot home (hopefully alive) in cases of severe malfunction and failure. By removing the pilot, “pilot-error” can also be instantly eliminated as the most significant cause of aircraft failure.

One such UAV, which is currently in development, is the Pro Observer from the Italian company, Pro-S3, located in Turin, delivering high level innovation services and products. The Pro Observer is a low-cost UAV developed specifically for short-range observation missions. With a wingspan of just 80cm, and an overall length of 75cm, the aircraft can be stored inside a small suitcase and assembled and launched by a single person (whether a spy or not) in less than 10 minutes.

Powered by a 40W electric motor, the Pro Observer has →
The Challenge
In the fast paced and highly competitive world of UAV design, achieving optimal performance within short design cycles is absolutely critical.

The Solution
CD-adapco provides a unique solution consisting of highly efficient polyhedral meshes and CAD-embedded CFD to achieve accurate, efficient, and easy-to-use CAE software.

The Pro Observer can reach a maximum speed of 27 kts (50 kph) and maximum flight time of around 40 minutes, ensuring a 10 to 12 km radius of action. The miniaturized avionics on-board Pro Observer ensures a totally automated flight, from launch to recovery. The missions are planned on a normal laptop computer and uploaded to the aircraft through the wireless link. The flight path can be modified during flight and the mission re-tasked. During the flight, a small tiltable high-resolution camera is used to take live images of the ground that are immediately sent to the ground station.

Typical missions for the Pro Observer include the following as well as any other situation where a cheap aerial point of view is necessary:
- over the hill view (military and civilian applications)
- traffic monitoring
- power lines monitoring
- surveillance of dangerous areas
- events monitoring
- agricultural monitoring

The Pro Observer also has the distinction of being the first aircraft to be fully designed with the aid of STAR-CCM+, the next-generation CFD code from CD-adapco. For the tailless design of this UAV, accurate calculation of the aerodynamic center was critical, influencing directly the flight performances and stability of the aircraft. In the words of Daniele Camatti, CEO of ProS3 and designer of the Pro Observer: “CFD analyses performed with STAR-CCM+ were critical in determining if the initial airframe designs were good enough to ensure the required flight performance. With STAR-CCM+ we were able to estimate with great precision the position of the aircraft’s aerodynamic center, to such an extent that, from the first test flight onwards, we have completely avoided crashes.”

The aircraft was modeled using STAR-Design, CD-adapco’s unique CAD-embedded design tool. STAR-Design is able to automatically create high quality polyhedral meshes, with enough extrusion layers around the aircraft to accurately predict the aerodynamic coefficients of the aircraft using a low-Reynolds number turbulence model. Through the power of associativity, changes to the CAD geometry in STAR-Design are reflected automatically in a revised CFD solution. This allows multiple design configurations to be tested both quickly and accurately. Daniele explains the benefits, “By using STAR-Design to run STAR-CCM+ we were able to quickly define the complete aircraft polar giving us a good starting point in choosing an appropriate power plant. Having completed a number of successful test flights, most of the flight parameters estimated with STAR-CCM+ have been confirmed by the test.”

More Information on Pro-S3 Visit http://www.pros3.it
For over a century mankind has devoted substantial resources toward achieving and improving flight. With the yearly threat of major wildfires and the ongoing conflicts in the Middle East as its primary focus, Propulsive Wing, LLC is developing unmanned aerial vehicle solutions for both domestic fire fighting support and military applications, covering a wide range of missions.

The Propulsive Wing is a completely new class of aircraft based on the integration of a cross-flow fan into the trailing edge of an airplane wing. The project began at Syracuse University with funding from NASA Glenn Research Center, and the technology is currently patent pending. Propulsive Wing LLC was founded to continue development and commercialize the platform as an unmanned airplane. Propulsive Wing is partnering with Elbridge, New York based engineering firm Allred & Associates, Inc. to accelerate this process.

With its unique design, for a given wingspan, Propulsive Wings are able to carry up to 3 times the payload weight and 10 times the internal payload volume of conventional systems. For this reason, the company calls its aircraft an aerial utility vehicle, or AUV for short. Propulsive Wing AUVs require shorter runways for take-off and landing (due to their extremely high lifting capability), generate low noise, and offer a high degree of user safety due to the elimination of external rotating propellers. This platform is applicable to many aircraft sizes ranging from small to large, unmanned and manned military, experimental, private and commercial passenger and cargo planes.
CFD was used throughout the design process to drive both aerodynamic and manufacturing decisions. For the current PW-4 prototype, which is currently undergoing flight-testing, the entire airframe was designed in CAD and modeled using STAR-CCM+ before fabrication even began. The ability of STAR-CCM+ to rapidly import a CAD model and create a new computational mesh allowed for multiple design iterations to be completed within a very short timescale. In addition to complete aircraft simulations of the PW-4 prototype, CFD was also extensively used to simulate sub-systems. In the design of the cross-flow fan propulsion system, simulation results were used to optimize the blade and housing geometry. Using CFD, Propulsive Wing designed a custom carbon-fiber cross-flow fan with excellent mechanical and aerodynamic performance.

At the next level, simulations were performed to investigate the integration of the cross-flow fan into a propulsive airfoil, these studies looked at the effect of propulsive airfoil design parameters, for example fan speed and sizing, duct inlet and outlet locations, and exhaust angle, on lift, drag, and pitching moment. Also, power requirements in various configurations were calculated. Full 3D unsteady simulations investigated the performance of the entire airplane. Understanding pitch stability characteristics and creation of a stable aircraft was a major technical hurdle. The Propulsive Wing configuration inherently involves complex coupling between the propulsion system and the wing pressure distribution, and can result in significant variations in pitching moment. For example, if fan speed is increased, this in turn increases the flow rate, the result is not only higher thrust, but, depending on the exhaust angle, changes the pitch-up or pitch-down tendency of the airplane. Simultaneously, however, the fan also produces greater suction at the air inlet, which alters the pressure distribution on the wing surface, while creating a nose-up reaction due to the direction of rotation. Using STAR-CCM+, Propulsive Wing was able to successfully understand these relationships and create a stable flying aircraft. The Propulsive Wing fits several niche applications where there currently is no solution. One is large payload, short duration sensor deployment for the military. Also, as the platform scales up, the large cargo and short takeoff and landing capabilities lend the AUV to missions where food, water, or other supplies need to be transported to remote or high altitude locations, both for military, as well as civilian emergency relief.

In addition to military use, one of the primary missions at Propulsive Wing is to develop the aircraft into a frontline component in the suppression of wildfires, which endanger people, wildlife, and agriculture. Annually the US Government spends over $1 billion fighting wildfires, which destroy about 8 million acres.

The Propulsive Wing is a completely new class of aircraft based on the integration of a cross-flow fan into the trailing edge of an airplane wing. The project began at Syracuse University with funding from NASA Glenn Research Center, and the technology is currently patent pending. Propulsive Wing LLC was formed in 2006 to commercialize a newly-developed high-lift, high-payload flying wing platform. Instead of external propellers, the design utilizes partially-embedded cross-flow fans for thrust and boundary-layer control. Based on 6 years of research and development, the aircraft is readily scalable and reconfigurable to meet specific mission requirements.

PW-4 PROTOTYPE COMING IN FOR LANDING

Construction of the PW-4 prototype is complete, and the model is now undergoing a flight test program. The vehicle has a wingspan of 7 feet, an empty weight of 24 pounds, and has an estimated payload capacity of over 30 additional pounds. Initial phases of the flight test program, which explored pitch stability and control and lateral stability and control, have now been completed, and we have verified a stable platform. The current testing phases involve expanding the flight envelope, and will ultimately involve exploring higher load carrying capabilities.

Over the next month, Propulsive Wing will begin releasing videos of the plane, which will be downloadable from: http://www.propulsivewing.com
aerial utility vehicle, or AUV for short. Propulsive Wing AUVs require shorter runways for take-off and landing (due to their extremely high lifting capability), generate low noise, and offer a high degree of user safety due to the elimination of external rotating propellers. This platform is applicable to many aircraft sizes ranging from small to large, unmanned and manned military, experimental, private and commercial passenger and cargo planes.

CFD was used throughout the design process to drive both aerodynamic and manufacturing decisions. For the current PW-4 prototype, which is currently undergoing flight-testing, the entire airframe was designed in CAD and modeled using STAR-CCM+ before fabrication even began. The ability of STAR-CCM+ to rapidly import a CAD model and create a new computational mesh allowed for multiple design iterations to be completed within a very short timescale.

In addition to complete aircraft simulations of the PW-4 prototype, CFD was also extensively used to simulate sub-systems. In the design of the cross-flow fan propulsion system, simulation results were used to optimize the blade and housing geometry. Using CFD, Propulsive Wing designed a custom carbon-fiber cross-flow fan with excellent mechanical and aerodynamic performance.

At the next level, simulations were performed to investigate the integration of the cross-flow fan within a propulsive airfoil, these studies looked at the effect of propulsive airfoil design parameters, for example fan speed and sizing, duct inlet and outlet locations, and exhaust angle, on lift, drag, and pitching moment. Also, power requirements in various configurations were calculated.

Full 3D unsteady simulations investigated the performance of the entire airplane. Understanding pitch stability characteristics and creation of a stable aircraft was a major technical hurdle. The Propulsive Wing configuration inherently involves complex coupling between the propulsion system and the wing pressure distribution, and can result in significant variations in pitching moment. For example, if fan speed is increased, this in turn increases the flow rate, the result is not only higher thrust, but, depending on the exhaust angle, changes the pitch-up or pitch-down tendency of the airplane. Simultaneously, however, the fan also produces greater suction at the air inlet, which alters the pressure distribution on the wing surface, while creating a nose-up reaction due to the direction of rotation. Using STAR-CCM+, Propulsive Wing was able to successfully understand these relationships and create a stable flying aircraft.

In addition to military use, one of the primary missions at Propulsive Wing is to develop the aircraft into a frontline component in the suppression of wildfires, which endanger people, wildlife, and agriculture. Annually the US Government spends over $1 billion fighting wildfires, which destroy about 8 million acres. Furthermore, due to unfavorable weather trends and other factors, forest fires have been growing in number, getting larger, and gaining in intensity.

The goal is to supplement the use of large tanker-sized aircraft with swarms of smaller unmanned drones that will each drop up to 200 pounds of water or fire-retardant chemicals. The short-field capability, compact size, and large cargo-carrying ability of the Propulsive Wing units, will allow rapid deployment from remote areas to help extinguish the thousands of small wildfires reported each year before they grow larger and become a threat to people and property. The US Forest Service estimates that up to 80% of the total resources expended each year are allocated to the small number of large fires. The goal, then, must be to put out all of the small fires before they become large, thus permitting the Forest Service the time and energy to focus more on controlled burns and forest management to help prevent future fires. Propulsive Wing envisions hundreds of cross-flow fan AUV systems based strategically around the West and Mid-West regions of the United States fighting wildfires on a 24/7 basis at a fraction the cost, as well as a fraction the risk to the pilots operating the small contract general aviation aircraft used for the same purpose. Such a system will improve the world’s ability to combat forest fires by an order of magnitude.
Virtual Design Enables a Revolutionary Flying Technology to Take Off

David Lasserre, ENTECHO, Australia

The challenge of designing a next generation compact Vertical Take Off and Landing (VTOL) craft has been addressed with the combination of a novel radial fan technology and the use of unique lifting and control surfaces.

CFD Techniques

The design process started with a searching aerodynamic analysis in order to establish the best interaction between the lifting surfaces and to set the parameters of the propulsion system that satisfy the lift requirements. During the concept creation stage, no prototype was built, and the geometry development relied only on the CFD results. To ensure a high turnover of results for each configuration, automation scripts were written to create, mesh and run a matrix of geometries and boundary conditions.

In the next step where the flight mechanics and stability are analyzed, all surfaces relevant to the control of the craft are modeled in detail. However, to reduce the complexity, rotor and stator blades are simulated through a momentum generator, using the user subroutines capability of STAR-CCM+. The radial momentum added to the system converges on the value of the power input needed to hover in each case. An additional swirl can also be added to accurately simulate any residual tangential flow.

The mesh generation and model setup is controlled by a script that implements the CD-adapco automatic meshing feature when running a series of cases at different control surface configurations and flight orientations.

The flight control system analysis has proved essential in the optimization of the performance of the attitude control system. For example, the flight performance of the manned platform in particular required detailed analysis of its behavior in ground effect. This flight control system analysis returns accurate aerodynamic
forces and pitch, roll and yaw torque inputs to the flight controls system lookup tables, with up to four configurations being run daily on the solving cluster.

Simultaneously, the propulsion system and lifting surfaces are analyzed in greater detail. Sector meshes are set up for the Moving Reference Frame (MFR) method to analyze the propulsion turbo machinery. Special attention is paid to the rotor and stator interactions with the blades optimized to satisfy the dual requirements of efficient lift generation and rotor torque cancellation.

During the same design loop, the yaw control surfaces capabilities can be evaluated in order to complete the range of information needed for the flight controls systems. Mesh size and setup parameters have been optimized to allow at least one configuration to be run overnight. This stage closes the aerodynamic design loop, as shape and dynamic loads are then known for the CAD/FEM team to finalize the model.

Benefits and Achievements
The STAR-CCM+ simulation process is fully integrated into the virtual design process and interacts strongly with the CAD design and software development for the control system. The design loads predicted by the CFD analysis make the choice of the composite materials in the craft’s structure much easier, leading to significant weight reductions and further improvements in the payload and endurance capabilities of the flight platform.

The flight control system CFD analysis has proven to be a powerful tool. One of its most important outputs is the data that is fed into a flight simulator that delivers realistic attitude response and lift characteristics. It has also made it possible to identify, quantify and address an unusual ground effect response and therefore avoided putting the prototype craft or personnel at risk.

Once the CFD calculations of aerodynamic performance and attitude control met the prerequisite targets, a prototype flight platform was constructed. The successful test flights of this “MuPod” UAV (unmanned aerial vehicle) has confirmed the value of CD-adapco products in providing accurate flight characteristics early in the development process. It was very rewarding to witness the technology at work as the prototype took off for the first time and behaved as predicted by the flight simulator.

Evolution
The virtual design environment approach has provided an early and thorough understanding of the potential and capabilities of this innovative flight technology. The number of hardware variants selected for construction has been reduced significantly by using the right tools and the right techniques and significant savings in time and cost have been realized as a result. We are currently updating the implementation methodology to use STAR-CCM+ with very promising results so far.

MORE INFORMATION http://www.entecho.com.au
Flapping Airfoil Analysis of Micro Air Vehicles using STAR-CD

D.Funda Kurtulus 1,2, Alain Farcy 1, Laurent David, Nafiz Alemdaroglu
Ecole National Superieur de Mechnaque et d’Aérotechnique, LEA, Poitiers, France
Middle East Technical University, Ankara Turkey

Flapping Motion Description Inspired from Birds and Insects
Hummingbirds and several insects use normal hovering where the wings are moving through a large angle in an approximately horizontal plane making a figure-of-eight motion with a symmetrical half-strokes.

The flapping motion is divided into 4 regions with the first region corresponding to half of the downstroke where the leading edge is pointing in positive direction and second one to the half-upstroke. While the third and fourth regions, are the mirror images of these two regions, corresponding to the second half of upstroke and downstroke respectively. Figure 2 shows a detailed description of the flapping motion in one period.

In order to use a simplified model, the symmetrical NACA 0012 airfoil is chosen for the present study. The nomenclature of the parameters is such that represents the angle of attack, c is the airfoil chord, V is the translational velocity, xa is the position of change of angle of attack, xv is the position of change of velocity. In the figure, the center of rotation (denoted by a) is at chord location of the airfoil. The motion is implemented into the program using the moving grid option and user defined subroutines of STAR-CD.

A total of 216 cases were investigated numerically, by changing the parameters described in Figure 2. The vorticity contours at the half-amplitude location for different center of rotations are given in Figure 3 for α=30°, Re=1000, xa=2c, xv=2c. During this parametric study it is found that the most influential parameter for the aerodynamic force coefficients is the angle of attack and the second is the center of rotation.

During the beginning of 2nd and 4th regions of the flapping motion, the aim of the flapping airfoil analysis is to understand the aerodynamic phenomena and the vortex topology of this highly unsteady motion. Instead of the use of real insect/bird wing geometries and motions, which are highly complex and difficult to imitate by an exact modeling, a simplified model is used to understand the unsteady aerodynamics and vortex formation during the different phases of the flapping motion.
the airfoil enters the trace of the leading edge vortex, translational vortex and rotational vortex that were generated before the return. At angles of attack higher than 30°, the effects of the downwash due to these vortices are stronger. For small angles, the lift coefficient has both negative and positive values, which give average force coefficients close to zero. As the angle of attack is increased to 30°, these negative peaks disappear and at 45° positive peaks are observable.

Vortex identification by considering Q-criteria and first and second eigenvalues are visualized. The demonstrated results are obtained by using STAR-CD for a 2-D, unsteady, laminar flapping motion. High positive values of Q imply vortex regions where the rotation rate is dominant compared to the strain rate. The vorticity contours and pressure values are also represented for comparison to the different techniques. The streamlines are visualized in both inertial reference frame and body fixed reference frame.

The whole computational domain is moving, with the motion imposed by user subroutines, defining the flapping motion angular and translational velocities. The motion of the grid domain close to the airfoil is shown in Fig.2. The whole domain is a 20c radius O-type computational grid. Macros were developed for the calculation of the second invariant of velocity gradient (also called as second invariant of the mean rate-of-displacement tensor) Eq.1 and eigenvalue of the sum (called the μ criteria):

\[ Q = \frac{1}{2} (\Omega^2 - S^2 \dot{S}) = - \frac{1}{2} \frac{\partial u_j}{\partial x_i} \frac{\partial n_j}{\partial x_i} \]  

(Eq.1)

The results are also compared with the experimental visualization techniques as Particle Image Velocimetry (PIV) and laser sheet visualizations (Figure 5). The numerical solutions are very satisfactory compared to experimental results although the problem is highly unsteady.

Conclusion
In this study, the flapping motion aerodynamics was considered for a symmetrical hovering case for use in future Micro Air Vehicle applications. MAV’s resembling insects and small birds with flapping wings. The complexity of the problem raises the necessity of a simplified model, so a two-dimensional model was investigated with a symmetrical airfoil with variable velocity and angular velocity laws.

The analysis tools used for the description of the phenomena are the numerical simulations and the experimental investigations. The numerical simulations were performed with STAR-CD using the moving grid capability. STAR-CD was used for the parametrical study to get a first idea of the parameters which influence the flapping motion study. It was concluded that the most influencing parameter is the starting angle of attack. The second important parameter is the center of rotation. The other parameters as the change of position of the velocity and angle of attack and Re (or Reynolds number) were found to be less important.
In this article, we describe a recent work in which we compared the advantages and disadvantages of the steady and transient approaches to the analysis of a four-bladed aircraft propeller. In the study, we examined a concept model of a ground based turbo-prop engine operating within an enclosure. We used STAR-CD to consider several facets of the propeller design, principally:

1. Maximum torque load on the propeller blades
2. Time varying cyclical loading of the blades
3. Mass flow through the system
4. Engine outlet temperature
5. Flow over the tip of the propeller blades

Our overriding question concerned the trade off between the expense of the numerical calculation technique and the accuracy of the solution it predicted. In order for a computational method to qualify as a valid and useful simulation technique, calculations are required to be both accurate and practical. We needed to understand whether the Implicit MRF approach could meet the technical challenge and whether the transient moving mesh approach could meet the schedule requirements of the project.

The CAD geometry of the concept design examined in the study was built in STAR-Design (Figure 1). All the components were created and meshed separately, using trimmed cell technology (Figure 2) before being assembled into a single model (Figure 3). The final assembly consisted of 1.75 million computational cells.
Fixed pressure boundaries were prescribed at inflow and outflow regions, and rotating wall boundaries to the surface of the propeller (Figure 4). The flow was considered compressible, consisting of large temperature gradients in the system due to the hot exhaust of the gas turbine engine. Identical flow properties, solver settings and geometric configurations were simulated for both the MRF and moving mesh approaches.

Our analysis revealed that both the steady MRF and the transient moving mesh approaches proved meritorious. The steady analysis was computationally stable and converged monotonically in a timely fashion. The steady simulation captured the basic flow structure across the propeller tips, as well as the temperature mixing of the engine exhaust. The MRF predicted a fixed torque loading on the blades; however, due to the steady nature of MRF, the analysis is not able to predict the cyclic torque loading that the blade experiences naturally during rotation. The steady analysis provided quick, general results in which the gross flow structure was predicted (Figure 5).

The transient moving mesh analysis provided more than the gross flow structure; the analysis additionally provided critical engineering data concerning the effects of the blade rotation in time. Specifically, we noticed a high torque loading experienced by all blades as they passed a particular point in the 360° revolution (Figure 6 & 7). The time accurate results of transient blade loading provided torque spike magnitudes; the results allowed us to determine if additional engineering of the engine mounting system was warranted to mitigate the high cyclic loading. The transient analysis also captured the temperature mixing as did the steady analysis, and predicted a system mass flow rate 4% higher than that of the steady case. The transient simulation required approximately 4-6 times more computational runtime to establish a “cyclically steady” solution, yet the analysis provided more insight for understanding the flow physics of the system.

From our examination, we conclude that the transient moving mesh analysis more appropriately captures high resolution, high accuracy flow behavior and cyclic fatigue characteristics. Although MRF is less expensive and acceptable for understanding the basic flow structure, the steady state MRF approach is not able to provide potentially critical time accurate information.
**STAR-CAT5 helps to design “coolest business jet ever”**

Stephen Ferguson, CD-adapco

Recently cited by Forbes magazine as “the coolest business jet ever”, the Falcon 7X is the latest in a long line of high-class executive jets from Dassault Aviation. As well as being cool, the Falcon 7X is also revolutionary - being the first civil aircraft ever to be designed using virtual rather than physical prototypes. Designed entirely using Dassault Systèmes CATIA Product Lifecycle Management (PLM) tools, every component on the Falcon 7X was subjected to rigorous CAE analyses.

Dassault Aviation is committed to providing the very highest standards of comfort for Falcon 7X passengers. The aircraft features Dassault Aviation’s breakthrough Environmental Control System, that maintains a constant cabin temperature and an adequate level of pressurization as the aircraft passes through extremes of external conditions. The Environmental Control System also manages safety critical aspects such as wing de-icing and avionics cooling. Central to the operation of the Environmental Control System is the mixing jet pump that mixes the air sent to the cabin from bleeds on different stages of the engine compressors. The pump’s job is to maintain the pressure and temperature within the cabin air-conditioning system independent of the regime in which the engines are operating.

Dassault Aviation’s current CAE process utilizes one-dimensional network analysis for sizing the mixing jet pumps. Although the process works well across a wide range of operating conditions, under certain extreme conditions - such as supersonic flow in the mixer - the accuracy of the network model was often found wanting due to the complex flow within the pump. In order to remedy this, Dassault Aviation adopted CD-adapco’s STAR-CAT5 CFD software to better characterize the performance of the pump under extreme conditions. STAR-CAT5 is the first industrial-strength CFD software to be fully embedded within the CATIA V5 PLM system.

The advanced capabilities of STAR-CAT5 enabled meshing of the real geometry of the jet pump without exiting CATIA V5. The mesh was exported to be optimized within STAR-CCM+, CD-adapco’s next generation CFD solver. In order to obtain the adapted refinement level, STAR-CCM+’s advanced capabilities were used to cut, combine, or fuse different parts of the mesh. Several parts of the mesh were created with STAR-CAT5 and were cut at the desired place with STAR-CCM+, then all the parts were combined and the interfaces were joined. Finally the resulting mesh was adapted in terms of size and shape for all areas of the flow.

The results of the STAR-CCM+ calculations were compared with laboratory tests and have very good agreement for all studied cases of flow, even for the more complex ones. These very high quality results now allow Dassault Aviation to optimize the in-house 1D code for extreme conditions of flow and encourage Dassault Aviation to continue in this direction in order to study more complex geometries.

The advanced capabilities of STAR-CAT5 enabled meshing of the real geometry of the jet pump without exiting CATIA V5. The mesh was exported to be optimized within STAR-CCM+, CD-adapco’s next generation CFD solver. In order to obtain the adapted refinement level, STAR-CCM+’s advanced capabilities were used to cut, combine, or fuse different parts of the mesh. Several parts of the mesh were created with STAR-CAT5 and were cut at the desired place with STAR-CCM+, then all the parts were combined and the interfaces were joined. Finally the resulting mesh was adapted in terms of size and shape for all areas of the flow.

The results of the STAR-CCM+ calculations were compared with laboratory tests and have very good agreement for all studied cases of flow, even for the more complex ones. These very high quality results now allow Dassault Aviation to optimize the in-house 1D code for extreme conditions of flow and encourage Dassault Aviation to continue in this direction in order to study more complex geometries.