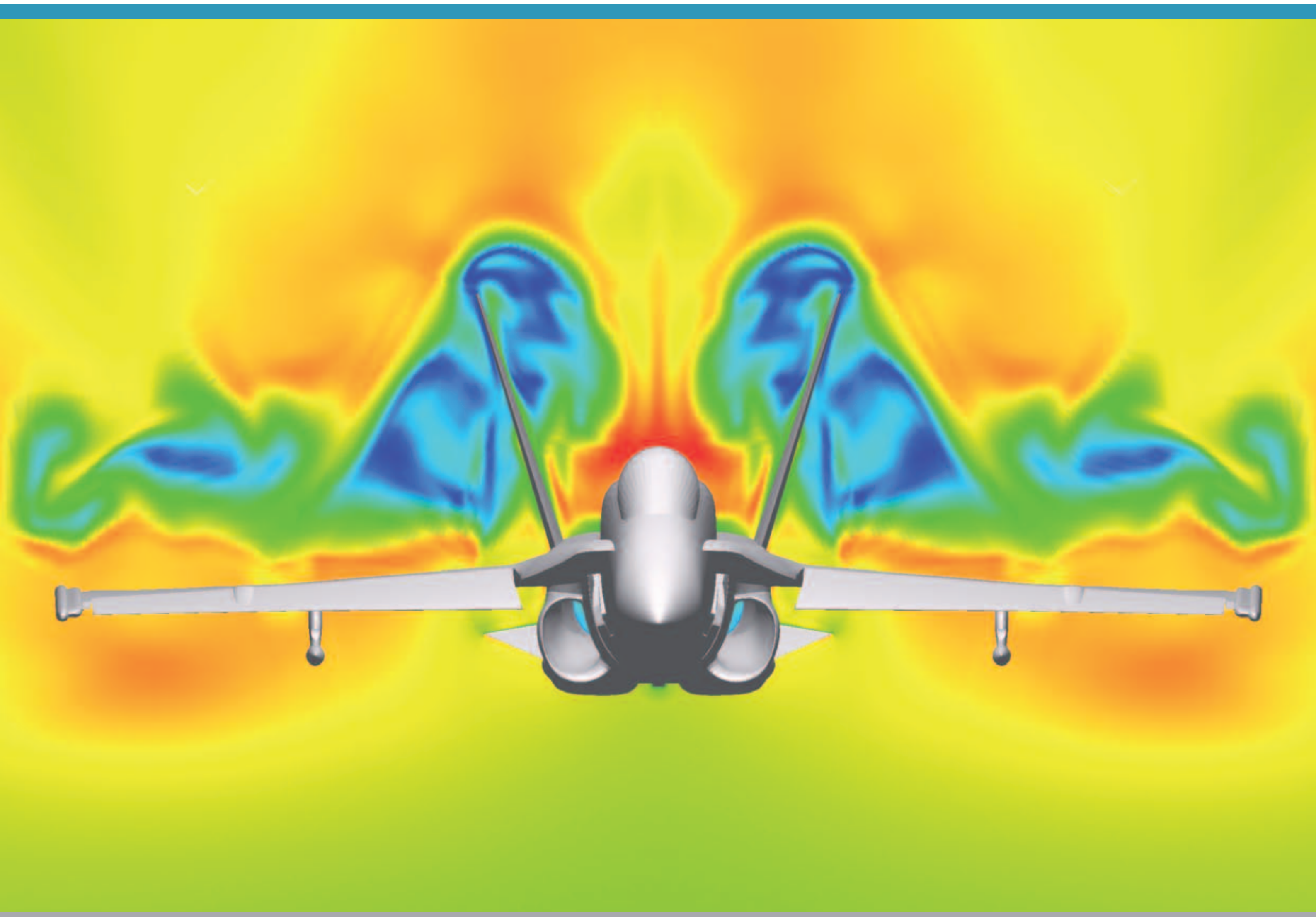
RUAG

Aerospace Defence Technology



Center Aerodynamics – Newsletter 11 – Fall 2005

In this edition

Dear reader, for the second time this year I have the honor to present you some selected projects from our Center Aerodynamics. Over the last years the technology of computational fluid dynamics (CFD) has strongly evolved. RUAG together with its partner CFS Engineering have further extended the capabilities of the Navier-Stokes Multi Block solver (NSMB). In addition to state-of-the-art steady state flow calculations, we are now focusing our interest on the fields of unsteady flow simulations and fluid-structure interactions. The first feature article describes the software tool which has been developed to combine structural and aerodynamic calculations, and shows an application for the F/A-18 fighter aircraft. How a combination of numerical and experimental means led to a highly optimized aerodynamic shape for the efficiency world record holder PAC-Car II is explained in the second article.

Please enjoy our second newsletter of 2005. Feedback and comments are always welcome.

M. Guillaume

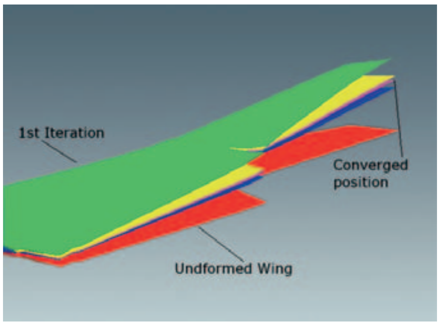
Michel Guillaume
Head Center Aerodynamics

Aero-elastic interactions on the Hornet

In a bid to create a practical maintenance schedule for the Swiss Air Force's F/A-18 Hornet fighter/attack jet, a full scale fatigue test rig was built and a wide variety of flight conditions simulated. Flight conditions were simulated by applying loads to different parts of the aircraft. The values of these loads were computed by Boeing, the US Navy and from experiments in the center's own wind tunnel. To complete the required data aerodynamics engineers, together with numerical specialist partner CFS Engineering, computed the aerodynamic loads using the Navier Stokes Multi-Block solver (NSMB).

Setting up the calculations involved constructing a highly detailed, structured multi-block mesh comprising more than 1,000 blocks and some eight million grid points. More than 80 separate flow cases were computed. They were made firstly, to validate the calculated results with wind tunnel experiments; secondly, to compare the computed aerodynamic loads with load cases provided by Boeing; thirdly, to compare the load results with flight measurement data of the US Navy and, finally, to complete the load database. Calculations were made on a cluster of six PCs running under Linux. The calculated loads on the jet coincided with the load databases provided by Boeing and the US Navy. A significant effort was made to convert the aerodynamic loads on each aircraft component into structural loads, which were then imposed on the aircraft mounted on the fatigue test rig.

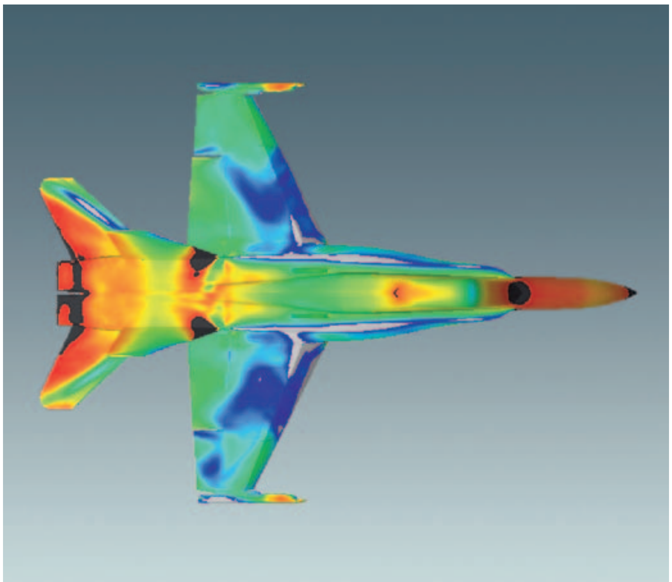
The numerical calculations clearly underscore the fact that, at high angles of attack, the influence of structural deformation of the aircraft wing cannot be dismissed and that flow unsteadiness (buffeting problems) become important. Additional studies were made to further analyze these two phenomena.



A new software tool, known as the Fluid-Structure Interaction (FSI), has been developed to transfer information between the fluid mechanics and structural mechanics solver. The FSI converts the aerodynamic loads computed by Computational Fluid Dynamics (CFD) into structural loads, which are then used as input for a Computational Structural Mechanics Simulation (CSM). The result of this simulation is a displacement of the structure, which the FSI translates back into a displacement of the surface mesh used in the CFD calculation. This process repeats itself until a converged position of the geometry is obtained. The figure shows this process for the wing of the F/A-18 with deflected control surfaces.

One of the difficulties found in this process was the deformation of the CFD surface and the volume meshes. Tools were developed permitting very large CSM surface mesh deformation. As an extreme example, a mesh displacement for a 90 degree rotating NACA 0012 pitching airfoil was computed. It demonstrated a good mesh quality level.

The analysis of the results from the static aero-elastic deformation study showed that the flow around the wing and thus, the aerodynamic load was strongly affected by the deformation of the structure. The picture shows the pressure distribution on the non deformed (port) and deformed (starboard) wing.



Future perspectives

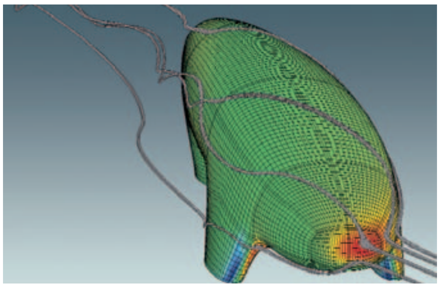
Such studies present several opportunities. The CFD code NSMB is now more than able to compute aerodynamic loads and perform unsteady calculations. It allows engineers to translate aerodynamic loads into structural loads and deformation. The new FSI software tool is presently under further development and the next step will be to compute dynamic fluid structure interaction problems. This year, a new F/A-18 grid with more than 14 million grid points has been prepared to model rather detailed aircraft components, such as the leading edge extension fence or missile fins.

PAC Car II

Cutting edge design

PAC-Car II is a highly efficient, single-seated hydrogen-powered fuel cell vehicle developed at the Swiss Federal Institute of Technology (ETH) in Zurich. In June 2005, the car took part in the Shell Eco Marathon in Ladoux, France: one of a series of fuel economy races which took place all over Europe. PAC-Car II broke the world record and achieved an equivalent of 5,385 km per liter of fuel (12,669 miles/US gal).

This was only possible by reducing the PAC-Car's driving resistance to a minimum. Major factors affecting the resistance of the three wheeled vehicle are weight, power train efficiency, the fuel cell system and aerodynamic drag. At an average speed of 30 km/h (18.6 mph), the minimum permitted by the rules, aerodynamic drag generates between 30% and 40% of the overall drag figure. The Center Aerodynamics supported the team to optimize the design. Both wind tunnel tests and CFD were applied to improve the aerodynamic modifications.



Wind tunnel tests were conducted at ETH in Zurich and in the Automotive Wind Tunnel (AWTE) using the rolling road feature. CFD simulations were performed using the Navier-Stokes Multi Block Solver NSMB.

Firstly, the car's outer shape was designed using CAD; taking space requirements for the pilot, power train and equipment into account. To validate the CFD code for the application, a 50% scale wind tunnel model was built and tested. The same geometry was also calculated with the NSMB (The picture shows streamlines and pressure distribution at a 15° yaw angle).

After this validation, the geometry was optimized in several steps, again using NSMB. The engineers found that the wheel fairing design had a crucial impact on the aerodynamic drag of the car, especially under yaw conditions. After the last iteration of the development calculations, a revised shape was again built as a 50% scale wind tunnel model.

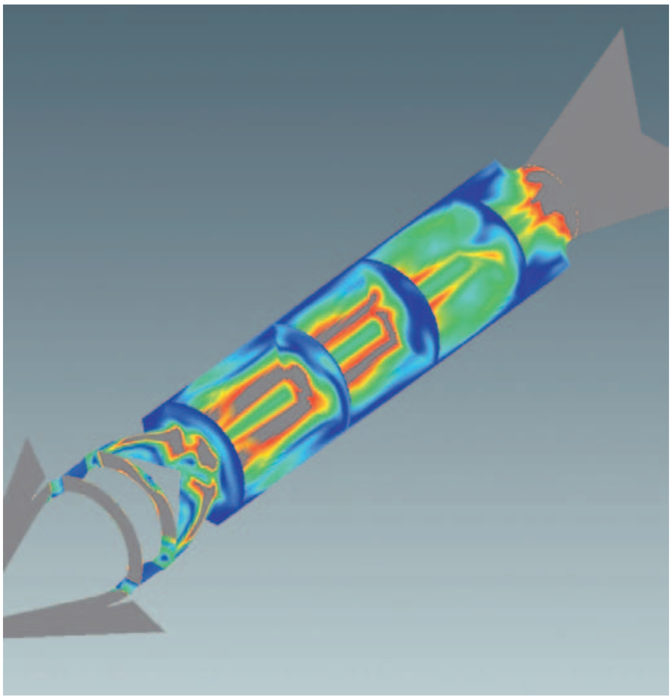
The design started with a drag coefficient (c_{bx}) for the initial shape of 0.095 and a drag area of 0.028 m^2. During the design iterations, c_{bx} was reduced below 0.08 and the drag area of the final shape did not exceed 0.019 m^2. Note that the drag area for the earlier PAC-Car I was as high as 0.069 m^2!

Recent Activities

Air intake studies including combustion

The objective of this study was to develop a solid fuel ramjet propelled projectile and major challenges surrounded the design of the air intake and the influence of viscous losses. Different intake geometries and exit nozzle diameters were then addressed, using the CFD code NSMB. Performance parameters included the total pressure at the nozzle outlet, the outflow mass flow and the requirement of subsonic flow in the combustion chamber to permit combustion.

The CFD results were validated with results from the free jet experimental test facility at Prins Maurits Laboratory TNO in the Netherlands. The CFD study permitted an increase in air intake performance of 30%. CFD and experimental results agreed on the pressure in the combustion chamber and the air intake un-start phenomena. Several unsteady simulations in respect to combustion were made to simulate the ignition process in the combustion chamber, leading to an increased understanding of the start-up of the ramjet.



Falling, with style

Body Flying is a sport which simulates the freefall experience of skydiving in a vertical jet of air. RUAG Aerospace's Center Aerodynamics is a long time consultant to Bodyflying Switzerland, a supplier of turnkey indoor and outdoor facilities for customers all over the world. These wind tunnels are used for professional skydiver training, as well as sports and entertainment functions.

The latest project involved converting a former MoD vertical wind tunnel located in Bedford, UK to the specific needs of body flyers. The main changes from an aerodynamic point of view were the addition of a new fan and a cooling system. The new facility opened its doors to the public in September 2005 and, with a flying area of almost 5 meters in diameter and 8 meters high, and vertical speeds up to 220 km/h, it is regarded as a world-class facility.

RUAG Aerospace
Center Aerodynamics
P.O. Box 301
CH-6032 Emmen
Switzerland

Tel. +41 412 683 801
Fax +41 412 683 897
aerodynamics@ruag.com
www.ruag.com

RUAG
Aerospace Defence Technology

11.05/xxxx