Published on *CD-adapco* (http://www.cd-adapco.com)

Home > Driving Down your Simulation Time with our Coupled Solver

Driving Down your Simulation Time with our Coupled Solver

Sabine Goodwin
August 11, 2013, 10:35 pm
Share This Post

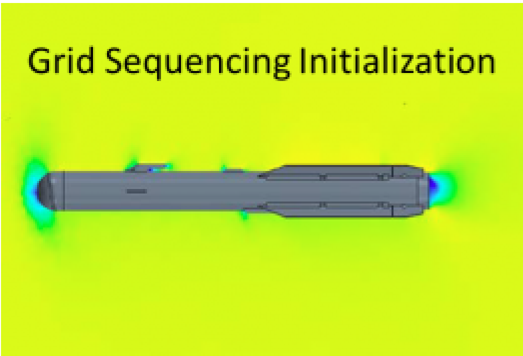
I wanted to take a moment and shine the spotlight on our coupled density-based solver. I must admit, I am perhaps a bit of a biased writer because after all, I have spent most of my career in industry analyzing and optimizing aircraft performance in flight regimes where the need for accurate shock capturing was part of daily life. Coupled solvers with a density-based approach have a proven track record for delivering robust solutions for these types of applications so it should come as no surprise that these numerical methods continue to spark my interest.

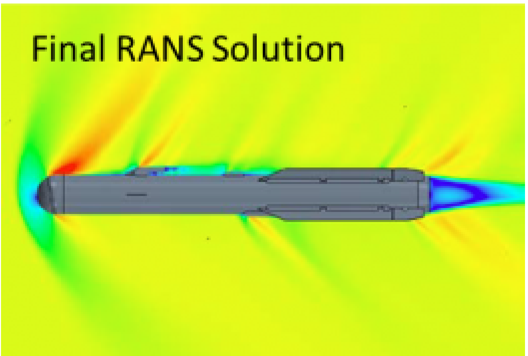
If you haven?t been performing your CFD simulations from under a rock, you have probably overheard your co-workers griping about coupled density-based approaches (paper-thin cubicle walls and open-plan offices are good for something!), with statements ranging from ?they don?t perform well at low speeds? to ?they just require too much memory?. Forget this for a minute because I am here to tell you that CD-adapco has made great strides in driving down the simulation times of our coupled solver for all speed regimes, resulting in a multi-purpose tool able handle a wide range of applications across many industries. Let me shed some light on how this is accomplished and what it means for our users.

Coupled density-based methods solve the conservation equations for mass, momentum and energy simultaneously using a (pseudo-) time-stepping approach. It is indeed true that solving this hyperbolic system of equations becomes ill-conditioned at low Mach numbers due to a disparity in convective and acoustic wave speeds, and it is well known that this leads to a significant deterioration in convergence. In addition, round-off errors are also larger at low Mach numbers resulting in a reduced accuracy. But there is a solution to this problem. To eliminate the numerical stiffness at low speeds, the coupled solver in STAR-CCM+ uses an automatic low-Mach pre-conditioning. To explain this in simple terms, think of it as the introduction of an artificial speed of sound (or compressibility) into the system and the effect of it is that it decreases the disparity among its eigenvalues (the system becomes less stiff). This powerful numerical tool not only enables fast convergence of solutions at low speeds, it also improves the accuracy of the results.

Problem solved? Not completely. Employing a low-Mach pre-conditioning technique can be problematic for some internal flows in which pressure differences must be established across regions of low speed flow, such as large vessels, long pipes, or flows with embedded recirculation zones. In these cases, the method may have problems with slow convergence of the mass balance (continuity) making it difficult to establish final mass flow. To alleviate this numerical problem, we have developed the novel Continuity Convergence Accelerator (CCA). This technique requires the solution of one additional elliptical equation, incurring a minimal increase in CPU time of the order of 5-10% per iteration. The result is that it greatly improves local and global balance of mass in cases where the pressure propagation through low-speed regions is important and leads to a significant acceleration in convergence (up to 50% in some cases).

There are several additional features in STAR-CCM+ that further enhance the stability and convergence of the coupled flow solver. For high supersonic and hypersonic flow, the AUSM+ (also known as awesome+) differencing scheme aids stability and insures accurate capturing of discontinuities. In addition, grid sequencing initialization provides a better initial condition for all speeds by solving an approximate initial solution via a series of coarsened meshes. It is like giving your CFD solutions a head start. It takes just a little computational effort (seconds to minutes) to perform the grid sequencing initialization and the pay-off is that it allows much more aggressive CFLs early on, resulting in a significantly faster convergence. Finally, our expert solutions driver combines a CFL ramp with corrections control/limiting and leads to a straight forward and robust convergence acceleration with little user input. This approach is hands-off (sit back and relax!) and helps to get to the results in 20 to 50% fewer iterations as compared to manual ramping.





One of the most significant advantages of using a coupled solver is that its performance is not sensitive to mesh density (meaning that CPU scales linearly with cell count). This characteristic of the coupled solver pays off specifically for cases where large meshes, high accuracy and quick turnaround are required. To illustrate this, consider Formula 1 design, an application where our coupled solver has been pushed hard (and there aren?t even any shocks to resolve!). F1 teams are regulated on the amount of CFD/wind tunnel time that that can be used and their meshes are usually massive. The key to success here is to get a lot of simulation throughput while ensuring high fidelity and accuracy. For these cases, using the coupled solver has allowed a significant reduction in computational cost, by 25%, resulting in many more computations per year.

It is true, one of the trade-offs of using a coupled density-based solver is that more memory is required, but in my opinion, this should not be the focus of your future water cooler discussions. The bottom line for the CFD community is a quick turn-around of accurate solutions for a complete range of speed regimes and applications and CD-adapco has been constantly improving on STAR-CCM+?s coupled solver to meet these needs.

Have you used our coupled solver? What do you think? If you have read this far, you must have an opinion, so please leave me your comments in the section below.
