Coupling Momentum and Continuity Increases CFD Robustness

FLUENT technology introduces a pressure-based coupled solver to reduce computation time for low-speed compressible and incompressible flow applications.

By Franklyn J. Kelecy, Applications Specialist, ANSYS, Inc.

The FLUENT computational fluid dynamics (CFD) solver has undergone extensive development to extend its robustness and accuracy for a wide range of flow regimes. Since its initial release, the FLUENT solver has provided two basic solver algorithms: The first is a density-based coupled solver (DBCS) that uses the solution of the coupled system of fluid dynamics equations (continuity, momentum and energy); the second is a pressure-based algorithm that solves the equations in a segregated or uncoupled manner. The segregated pressure-based algorithm has proven to be both robust and versatile, and has been utilized in concert with a wide range of physical models, including multiphase flows, conjugate heat transfer and combustion. However, there are applications in which the convergence rate of the segregated algorithm is not satisfactory, generally due to the need in these scenarios for coupling between the continuity and momentum equations. Situations in which equation coupling can be an issue include rotating machinery flows and internal flows in complex geometries.

The ANSYS CFX solver relies on a pressure-based coupled solver approach to achieve robust convergence rates. ANSYS now offers a similar pressure-based coupled solver (PBCS) for the first time with version 6.3 of the FLUENT software. As its name implies, the algorithm solves the continuity and momentum equations in a coupled fashion, thereby eliminating the approximations associated with a segregated solution approach where the momentum and continuity equations are solved separately. While these approximations do not affect solution accuracy at convergence, they can hamper the convergence rate for certain classes of problems. With the coupled approach, removing the approximations due to isolating the equations permits the dependence of the momentum and continuity on each

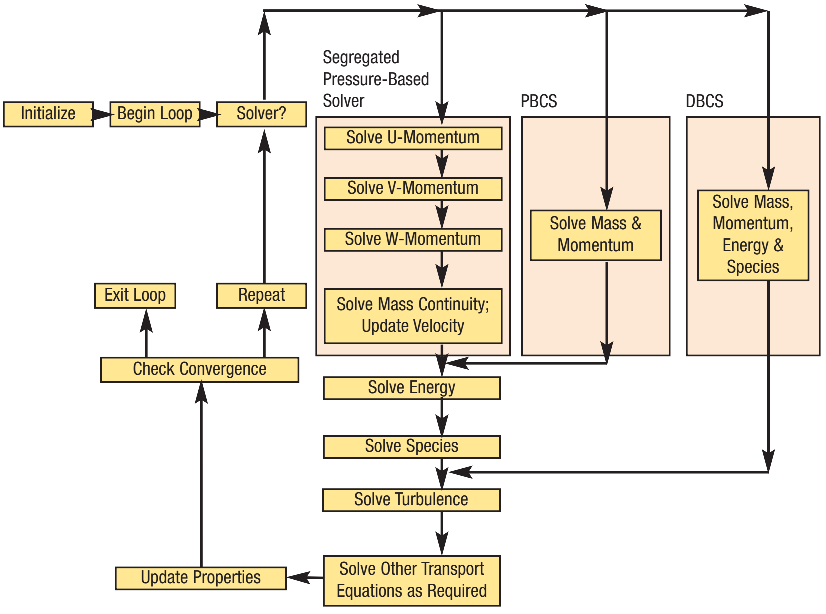

Figure 1. Flowchart illustrating FLUENT solver algorithms

other to be felt more directly. This leads to a more rapid and monotonic convergence rate and hence faster solution times. In addition, the coupling leads to improved robustness such that errors associated with initial conditions, nonlinearities in the physical models, and stretched and skewed meshes do not affect the stability of the iterative solution process as much as with segregated algorithms. The coupled algorithm can also be used with a wide range of physical models such as reacting flows, porous media, and many multiphase models, including volume of fluid (VOF) models.

Algorithm Overview

A flowchart illustrating the pressure-based and density-based solver algorithms is shown in Figure 1. This diagram depicts the process by which the pressure-based, segregated algorithm solves the momentum equations, for the unknown velocity components one at a time, as scalar equations and then solves a separate equation for mass continuity and pressure. The pressure solution is used to correct the velocities such that continuity is satisfied. When the flow equations are coupled together, the coefficients that are computed for each equation contain dependent variables from the other equations. In the segregated algorithm, these variables are supplied simply by using previously computed values, which introduces a decoupling error. The decoupling error can delay convergence in cases in which strong pressure–velocity coupling exists.

The pressure-based coupled solver differs from the segregated algorithm in that the continuity and momentum equations are solved in a fully coupled fashion. That is, a single matrix equation is solved in which the dependent variable is now a solution vector containing the unknown velocities and pressures. This is similar to the density-based implicit solver, except that the density-based solver also includes the energy equation in the coupled system and employs a different discretization of the flux terms, among other differences. The tradeoff with respect to computational resources is that the PBCS requires about twice the memory per cell as the segregated algorithm. This is due to the storage required for the coupled (matrix) equations. In general, the storage requirements are comparable to, though slightly less than, the density-based implicit algorithm.

It is important to note that, unlike the density-based schemes, the PBCS does not include the energy equation in the coupled system. This means that the density-based solver may still be preferable for high-speed compressible flow cases, in which coupling the energy equation is important. The PBCS can be used instead for all cases in which one previously would have used the segregated scheme, including low-speed compressible flows, incompressible flows or cases that require models available with only the pressure-based solver.

Using the Pressure-Based Coupled Solver

Activating the pressure-based coupled solver in FLUENT software is very straightforward. First, select the Pressure-Based option in the Define→Models→Solver panel as shown in Figure 2. (Users of older versions of FLUENT software should note that this manner of selecting solver algorithms is new in version 6.3.) Once this option is activated, the user then can select the coupled solver from the Pressure-Velocity Coupling list in the Solve→Controls→Solution panel, as shown in Figure 3.

Activating the pressure-based coupled solver exposes new solver

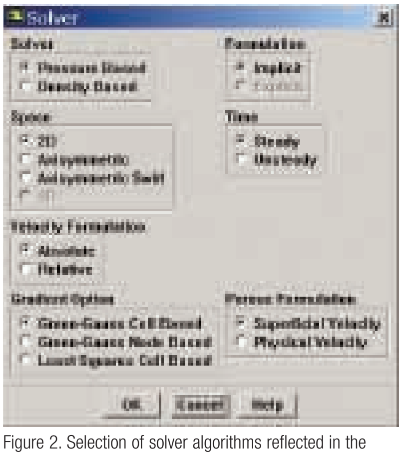

Figure 2. Selection of solver algorithms reflected in the Define→Models→Solver GUI (This panel was introduced as new in FLUENT 6.3.)

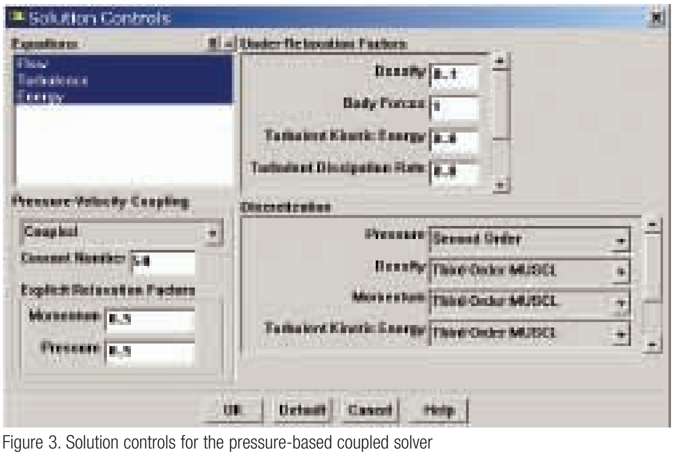

Figure 3. Solution controls for the pressure-based coupled solver

options in the Solve→Controls→ Solution panel. As shown in Figure 3, the user can supply values for a Courant number and explicit relaxation factors in addition to the usual under-relaxation factors and discretization options. The Courant number controls the diagonal dominance of the coupled system and works in a fashion similar to the Courant number employed in the density-based implicit solver. The default value is 200, but larger values may be used to accelerate convergence and smaller values to improve stability (for problems involving complex physics). If the pressure-based coupled solver is used for unsteady problems, the Courant number can often be set much higher (for example, 1 million). The explicit relaxation factors for momentum and pressure are essentially under-relaxation factors used to smooth changes in the velocities and pressures that are being solved for. The default values of 0.75 are suitable for most steady-state cases, but lower values (ranging from 0.25 to 0.5) may be required for problems with higher-order discretizations, skewed meshes or complex physics. For unsteady problems, the explicit relaxation factors can usually be set to 1.0.

The pressure-based coupled algorithm is an important milestone in the development of the FLUENT solver, as it provides the user with a modern, fully coupled solution approach that is suitable for a wide range of flows. While it requires more memory to store the coupled coefficients, the improvement in performance can be dramatic, as illustrated in the example described in this article. ■

A Pressure-Based Coupled Solver Example

To demonstrate the efficacy of the pressure-based coupled solver, a standard compressible flow validation case, the RAE 2822 airfoil, was solved using the pressure-based segregated, pressure-based coupled and density-based implicit algorithms in FLUENT 6.3 technology. The airfoil was modeled as an external air flow problem with a freestream Mach number of 0.73 and a Reynolds number based on chord of 6.5x106. The 2-D numerical model utilized a quad mesh with 126,900 cells, as shown in Figure 4. The flow was assumed to be steady-state, viscous, turbulent and compressible, with the ideal gas law used for the equation of state. Turbulence was modeled using the realizible k-ε turbulence model with non-equilibrium wall functions. Second-order discretizations were used for all equations. For the pressure-based coupled solver, the CFL (Courant) number was set to 200 and the explicit relaxation factors to 0.5 for both momentum and pressure. Default solver settings were employed for the segregated and density-based implicit algorithms.

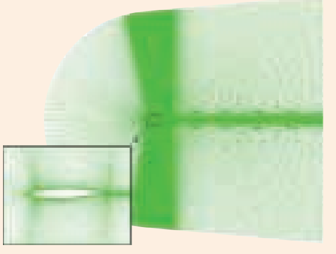

Figure 4. RAE 2822 airfoil test case mesh

The solutions obtained are illustrated in Figures 5 and 6. All three algorithms capture the suction surface shock wave crisply and show excellent agreement with the experimental data for this case. Figure 7 presents a table with the solver performance and resource requirements for each algorithm. As expected, the segregated solver uses the least memory. However, because of the close coupling of the momentum and continuity equations for this case, the data shows that the segregated solver required 2,570 iterations to reach convergence. In contrast, the pressure-based coupled solver required only 298 iterations to converge, which also compares favorably to the 976 iterations required by the density-based implicit solver.

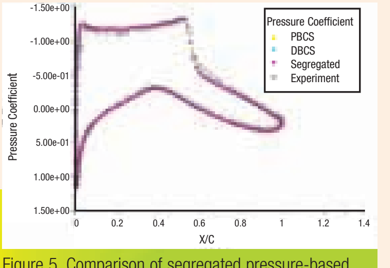

Figure 5. Comparison of segregated pressure-based PBCS and implicit DBCS solutions with test data for the RAE 2822 airfoil

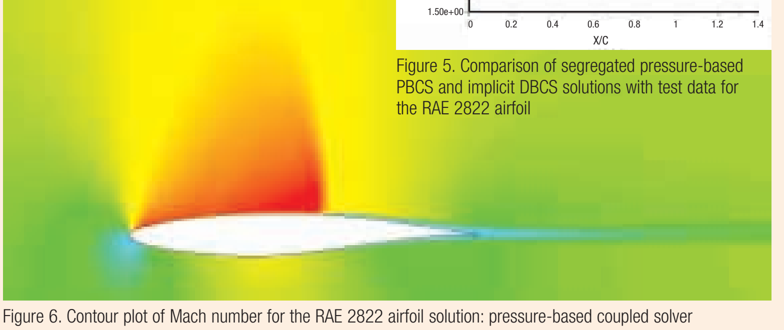

Figure 6. Contour plot of Mach number for the RAE 2822 airfoil solution: pressure-based coupled solver

Solver	Memory (MB)	Time per Iteration (seconds)	Iterations to Convergence	Time to Convergence (hours)
Segregated	172	2.10	2570	1.50
PBCS	259	3.26	298	0.27
DBCS	317	3.82	976	1.04

Figure 7. Comparison of computer resources and solver performance for the RAE 2822 airfoil