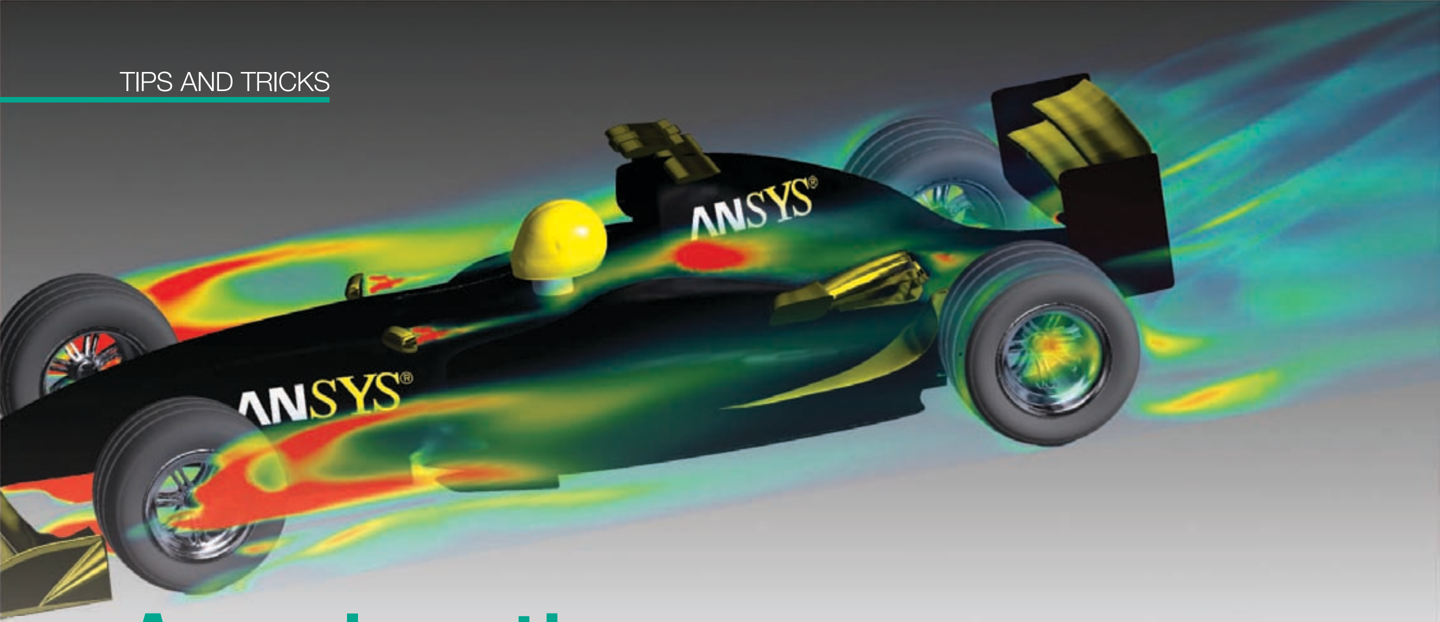


Accelerating CFD Solutions

Several recent enhancements in ANSYS FLUENT solver capabilities accelerate convergence and reduce solution time.

By Mark Keating, Principal Engineer, ANSYS, Inc.

Many solver performance enhancements have been introduced to ANSYS FLUENT fluid dynamics software over the past few releases. These capabilities can dramatically improve the speed and reliability of simulation. Running the solver out of the box does not always guarantee optimum solver settings for any particular application. So by understanding and using solver technology appropriately, a user can obtain faster results and better convergence.

Pressure-Based Coupled Solver

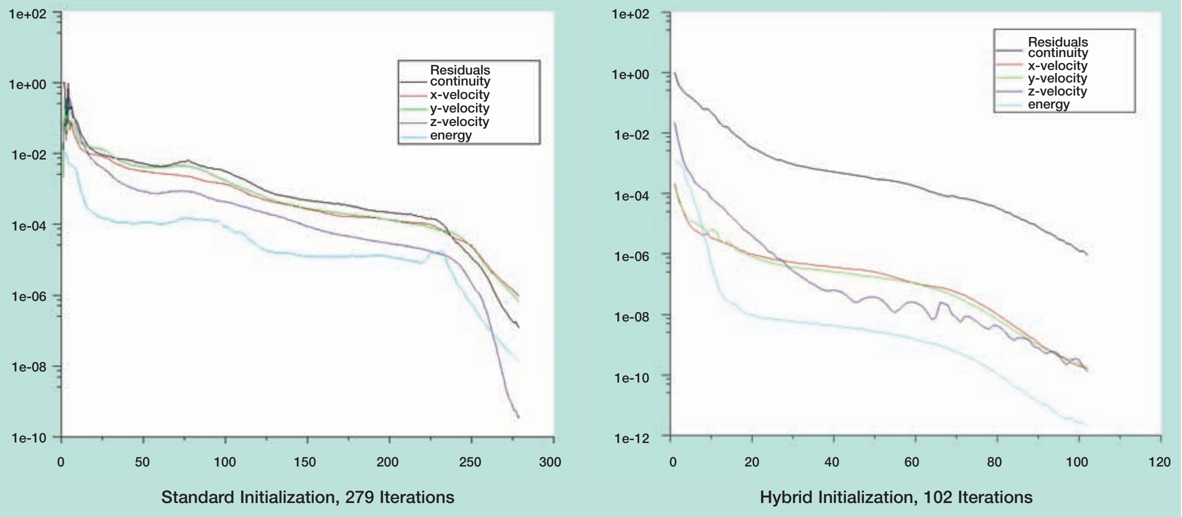
The pressure-based coupled solver (PBCS) was introduced in 2006, and its usage is growing. This solver reduces the time to overall convergence, by as much as five times, by solving momentum and pressure-based continuity equations in a coupled manner. Though there is a slight increase in associated memory requirements for using this solver, its benefits far outweigh the drawbacks. The PBCS is becoming the solver of choice for subsonic applications. When using it, re-ordering the grid is always advisable. The default explicit under-relaxation factors (URFs) for pressure and velocity of 0.75 are generally robust values, but they should be reduced for skewed meshes, when oscillatory convergence is experienced, or when higher-order discretization is employed to about 0.4 to 0.5. Taking the turbulence URF up to 0.95 to 0.99 can help to accelerate viscous cases. To access the PBCS, change the p–v coupling in the drop-down list from SIMPLE to Coupled.

Pseudo-Transient Method

The pseudo-transient solution method, introduced in version 13.0, is a form of implicit under-relaxation for steady-state cases. It allows users to obtain solutions faster and more robustly than previous versions of ANSYS FLUENT software, especially for highly anisotropic meshes, when using PBCS and density-based (DBNS) implicit solvers. This method uses a pseudo-transient time-stepping approach. In general, the time per iteration is slightly higher, but in some extreme cases the number of iterations required for convergence using this method has dropped by an order of magnitude or more. Usually, overall speedups of 30 percent to 50 percent can be expected. Cases with multiple reference frame (MRF) zones should benefit from using this method. The table shows the levels of improvement possible.

Cases	Courant number-based coupled (iterations)	Pseudo-transient coupled (iterations)
Backward facing step (turbulent: SST)	750	75
Film cooling benchmark (turbulent: SA)	2,300	1,350
Flat plate, SST transition model	1,200	100
Rotor/stator with mixing plane model	500	250
Centrifugal pump	220	50
Axial compressor stage	400	110

Solver speedups achieved using the pseudo-transient coupled solver in ANSYS FLUENT 13.0



Residual plots for a heat exchanger case show the effect on iteration count of hybrid initialization. This case does not converge with default under-relaxation and standard initialization, but it converges well with the default under-relaxation factors. It is more than two times faster when hybrid initialization is used.

Initialization Methods

Providing an initial data field that is close to the final solution for steady-state cases means the solver has to do less work to reach the converged result. Therefore, this reduces simulation time. Typically, many users employ standard initialization; some use patching for localized control, especially for moving domains or multiphase analyses. Interpolation files are used to initialize cases, and the interpolation workflows available in ANSYS FLUENT have been improved in recent releases. There are also other initialization techniques for further accelerating the simulation convergence.

Full multigrid initialization (FMG), introduced in 2006, provides the initial and approximate solution at a minimum cost to overall computational expense. The feature is accessed via the text user interface (TUI) and re-orders the grid as part of the process. The commands are:

```
Solve>initialize>set-fmg-initialization
Solve>initialize>fmg-i
```

The overall initialization time using this approach is much longer than that using standard initialization by zone, but it allows a much quicker solve. FMG solves Euler equations and is available for single-phase flows only. FMG initialization provides the best-guess initial solution. It is particularly suited to turbomachinery as well as external and compressible flow problems.

Hybrid solution initialization was introduced at version 13.0 and uses a collection of recipes and boundary interpolation methods to efficiently initialize the solution based purely on simulation setup — so the user does not need to provide additional inputs for initialization. The method can be applied to flows ranging from subsonic to supersonic. It is the recommended method when using PBCS and DBNS for steady-state cases in ANSYS FLUENT 13.0. This initialization may improve the convergence robustness for many cases. Unlike FMG, this initialization method can be used for multiphase flows.

Summary

A number of solver settings are available to aid solution acceleration and convergence within the ANSYS FLUENT solver. Individually, these techniques can be used to reduce solution times; combined, they offer even greater capability. For example, in some cases, moving from a segregated solver to PBCS with FMG and the pseudo-transient method has resulted in up to 100 times speedup. These features show the benefits of investigating and taking advantage of new solver technologies.