Unsteady/Steady Continuous Adjoint Method Using a Block Coupled Solver in OpenFOAM: Application on the Drivaer Vehicle

Christos K. Vezyris
Evangelos M. Papoutsis-Kiachagias
Kyriakos C. Giannakoglou
National Technical University of Athens, School of Mech. Eng.,
Parallel CFD & Optimization Unit, Athens, Greece

In this work, shape optimization is performed using the unsteady and steady continuous adjoint method. An implicit numerical solver, programmed in OpenFOAM, for both the Navier-Stokes and the corresponding continuous adjoint equations, is used.

The implicit, block-coupled solver is based on the pseudo-compressibility approach and foam-extend-3.1 is used as the programming environment. The block-coupled solver computes the solution for the flow variables simultaneously, leading to faster convergence and an implicit treatment of the numerically stiff Adjoint Transpose Convection (ATC) term.

A drag minimization problem is considered. The Drivaer vehicle is studied, which is an AboutFlow Adjoint Optimisation Benchmark test case. The objective function is the drag minimization by optimizing the side view mirror.

I. Introduction

The lack of a pressure term in the continuity equation makes the numerical solution of the incompressible Navier–Stokes flow equations challenging. Commonly, using appropriate transformations a Poisson like equation is derived to allow the pressure calculation. OpenFOAM©, among other CFD packages, uses the SIMPLE algorithm to numerically solve the momentum and continuity equations (cast in the form of a Poisson equation) in a segregated manner. An advantage of this approach is the overall low memory requirement[2].

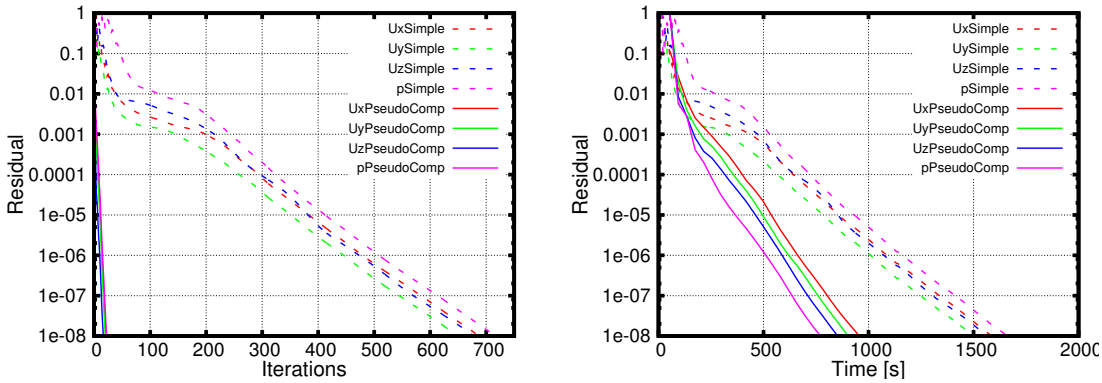
The simultaneous numerical solution that implicit block-coupled solvers undertake, promise faster convergence and lower total computation time. Moreover, potentially greater stability could arise by block-coupled solvers, though in the expense of increased memory requirements.

II. Development of an Implicit Block Coupled solver in OpenFOAM

The physical decoupling between the continuity and momentum equations is overcome by implementing the pseudo-compressibility approach[1]. Both the Navier–Stokes and the corresponding adjoint equations are discretized using the Roe flux scheme. The implicit, block-coupled solver is programmed in foam-ext-3.1 environment. The existing block matrix infrastructure is used, though altered when needed and in conjunction with developed code which allows the optimization process to be conducted.

III. Results

The developed solver is first tested in steady state problems. It is compared with the standard, Navier–Stokes solver for incompressible flows which exists in foam-extend-3.1. A 3D S-bend duct, being among the EU-funded AboutFlow project benchmark test cases, is chosen for demonstration purposes. The computational mesh consists of 446×10^3 cells and the Reynolds number is $\sim$400. In figure 1a and 1b the comparison is illustrated. The number of iterations needed by the block-coupled solver to converge is approximately 25, as seen in figure 1a, while the segregated solver requires approximately 680 iterations. In figure 1b the total computational time for each solver is shown. It is observed that the speed-up resulted by the block-coupled solver is $\sim 2.2\times$.



(a) Number of iteration.

(b) Computational time needed.

Figure 1: Developed block-coupled solver compared with the segregated SIMPLE algorithm based solver. The test case is a 3D, S-bend duct.

References

[1]A. J. Chorin. A numerical method for solving incompressible viscous flow problems. *Journal of Computational Physics*, 135(2):118 – 125, 1997.

[2]M. Darwish, I. Sraj, and F. Moukalled. A coupled finite volume solver for the solution of incompressible flows on unstructured grids. *Journal of Computational Physics*, 228(1):180 – 201, 2009.