Comparison of various implicit, explicit, centered and upwind schemes for the simulation of compressed flows on moving mesh

Marc Buffat[1], Anne Cadiou, Lionel Le Penven, Catherine Le Ribault[2]
[1] *Université Claude Bernard Lyon I,*
[2] *Ecole Centrale de Lyon, 36, av Guy de Collongue, 69 134 Ecully, France*
Laboratoire de Mécanique des Fluides et d'Acoustique

26th February 2004

Compressed flows, such as flow inside internal combustion engines, represent an important challenge for numerical simulation. In addition to the unsteady and three dimensional features of these compressible flows, numerical difficulties arise from the moving domain and the low Mach number. Classical explicit centered scheme are unstable and either high order explicit scheme (Padé, ...) or upwind schemes (Roe, WENO, ...) should be used. However the CFL stability condition impose very small time step at low Mach number. Implicit second order centered schemes seems to be a promising alternative.

Several numerical schemes will be compared on two configurations typical of those low Mach number flows :

1. a mixed finite volume/finite element scheme using an explicit (Runge Kutta) time integration and a Roe-Turkel Riemann solver (NadiaLES software)
2. a finite volume centered scheme on unstructured meshes using an implicit time integration (BDF) (NadiaVF software)
3. a finite difference solver using an explicit (Runge Kutta) time integration and a high order centered (Padé) and upwind (WENO) schemes

The first test case is a viscous decay of a 2D Taylor vortex for 0.1 and 0.01 Mach number. An analytical incompressible solution allows to study the numerical behavior of these schemes at low Mach number.

The second test case is the compression of a 3D Taylor vortex with a compression rate of 5. At low Reynolds number the flow remains stable and an analytical solution exists in the limit of zero Mach number. However at higher Reynolds number, the flow becomes unstable and turbulent. Reference solutions have been obtained using a spectral solver for Reynolds numbers $Re = 1600$ (stable solution) and $Re = 6000$ (transition to turbulence). We will compare the numerical behavior of the various schemes on the 2D laminar flow at $Re = 500$, and on the 3D cases at $Re = 1600$ and $Re = 6000$.

The final paper will present an overview of these comparisons.