Higher Order Workshop 3: Problem C3.3
Direct Numerical Simulation of the Taylor-Green Vortex

Presented by

BRIAN C. VERMEIRE
PHILIP ZWANENBURG
SIVA NADARAJAH

Computational Aerodynamics Group

McGill University
Montreal, Quebec, Canada
Summary

A high-order Collocation Penalty via Reconstruction (CPR) computational fluid dynamics code that solves the compressible Navier-Stokes equations was applied to the Taylor-Green vortex problem to test the accuracy and the performance of the direct numerical simulation of three-dimensional periodic and transitional flow. Based on analysis of the scheme’s dispersion and dissipation characteristics [1, 2], it was hypothesized that the CPR scheme can be used for Implicit Large Eddy Simulation (ILES), simulating transitional and turbulent flow without the addition of an explicit SGS model [3]. The third-order explicit Runge-Kutta scheme of Shu and Osher [4] is used to advance the simulation in time.

1 Code Description

The conservation laws are discretized by the correction procedure via reconstruction (CPR) scheme with DG correction functions. The divergence of the inviscid fluxes are determined either through a chain rule or Lagrange polynomial approach. The Roe flux is employed as the common interface flux and the BR2 scheme for the viscous flux. As for boundary conditions, Riemann invariants are used in the far-field, while either slip or adiabatic non-slip on the walls. The dynamic viscosity coefficient is either held constant throughout the computational domain or obtained from the Sutherland’s law. For the flat plate boundary layer case, the former is employed. Steady state solutions are obtained using a Newton-Krylov algorithm, which serves as the primary solver. The sparse linear system of equations are solved using GMRES included in the PETSc package version 3.2-p7, while preconditioning is provided by a block-Jacobi method. Before GMRES is employed, several block-Jacobi iterations are often performed. The solver is parallelized using MPI via Open MPI, version 1.4.3 where grid partitioning is achieved through ParMETIS. An implicit-explicit (IMEX) scheme serves as a secondary solver, where a three-stage diagonally implicit Runge-Kutta (DIRK) is used. Each stage is split between an explicit and implicit sub-stage, where the non-stiff regions are solved with an explicit RK, while the stiff portions are solved through the above stated Newton approach. Post-processing is typically performed with Tecplot 360 and/or Gmsh version 2.8.5.
1.1 Computations

The *Guillimin* cluster of the McGill high performance computing (MHPC) infrastructure, part of the Compute Canada HPC network, served for the computations on the hb architecture. Machine specifications and Taubench results are presented in Tables 1 and 2.

<table>
<thead>
<tr>
<th>Machine name</th>
<th>Specifications</th>
</tr>
</thead>
<tbody>
<tr>
<td>MHPC-(hb)</td>
<td>Dual Intel Westmere EP Xeon X5650</td>
</tr>
<tr>
<td></td>
<td>(6-core, 2.66 GHz, 12MB Cache, 95W)</td>
</tr>
</tbody>
</table>

Table 1: Computer specifications

<table>
<thead>
<tr>
<th>Machine name</th>
<th>Taubench CPU times</th>
</tr>
</thead>
<tbody>
<tr>
<td>MHPC-(hb)</td>
<td>9.5 (s)</td>
</tr>
</tbody>
</table>

Table 2: Taubench results

2 Case Summary

The following flow conditions were specified such that the flow be effectively incompressible:

<table>
<thead>
<tr>
<th>Quantity</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reynolds number, Re</td>
<td>1600</td>
</tr>
<tr>
<td>Ratio of specific heats, $\gamma$</td>
<td>1.4</td>
</tr>
<tr>
<td>Prandtl number, Pr</td>
<td>0.71</td>
</tr>
<tr>
<td>Bulk viscosity, $\mu_v$</td>
<td>0</td>
</tr>
<tr>
<td>Mach number, Re</td>
<td>0.1</td>
</tr>
<tr>
<td>Characteristic convective time, $t_c$</td>
<td>$\frac{L}{V_0}$</td>
</tr>
<tr>
<td>Final time</td>
<td>$20t_c$</td>
</tr>
</tbody>
</table>

Residual tolerances or other convergence criteria:
Simulation run until $20t_c$.

Machines used (number of cores if parallel):
384 processors using METIS [5].

3 Meshes

A regular cartesian mesh with $52^3$ nodes (for a P4 solution) was generated using a Matlab script.

Domain size (periodic cube):
$-\pi L \leq x, y, z \leq \pi L; L = 1$

Structured meshes with DOF/element = $(P + 1)^3$
For P4 results presented here with $52^3$ elements ($260^3$ total DOF), mesh spacing = $\frac{2\pi}{52}$.

4 Results

All results presented below compare the P4 CPR DNS results ($260^3$ DOF) with the reference data from from spectral DNS simulation ($512^3$ DOF).

4.1 Errors

The $L_2$ norms of the errors between the three measured parameters and the reference data are given in the table below; the error plots give the errors as a function of the non-dimensionalized time.

<table>
<thead>
<tr>
<th>h</th>
<th>TauBench</th>
<th>Total Kinetic Energy</th>
<th>Dissipation Rate</th>
<th>Enstrophy</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.85e-03</td>
<td>8.56e+07</td>
<td>1.46e-03</td>
<td>7.41e-03</td>
<td>7.55e+00</td>
</tr>
</tbody>
</table>

Table 3: $L_2$ Errors for measured parameters
Figure 1: Evolution of the dimensionless energy dissipation rate as a function of the dimensionless time: results of reference data and CPR code.

Figure 2: Evolution of the enstrophy as a function of the dimensionless time: results of reference data and CPR code.
Figure 3: Filled contours of $\frac{L}{V_0} |\omega|$ at $\frac{x}{L} = -\pi$ with contour lines from spectral DNS simulation at $t^* = 8$

Figure 4: Coloured contours of $\frac{L}{V_0} |\omega|$ at $\frac{x}{L} = -\pi$ with contour lines from spectral DNS simulation at $t^* = 8$
Figure 5: Evolution of the total kinetic energy as a function of the dimensionless time: results of reference data and CPR code.

Figure 6: Error in the dimensionless energy dissipation rate as a function of the dimensionless time: results of reference data and CPR code.
Figure 7: Error in the enstrophy as a function of the dimensionless time: results of reference data and CPR code

Figure 8: Error in the total kinetic energy as a function of the dimensionless time: results of reference data and CPR code
References


