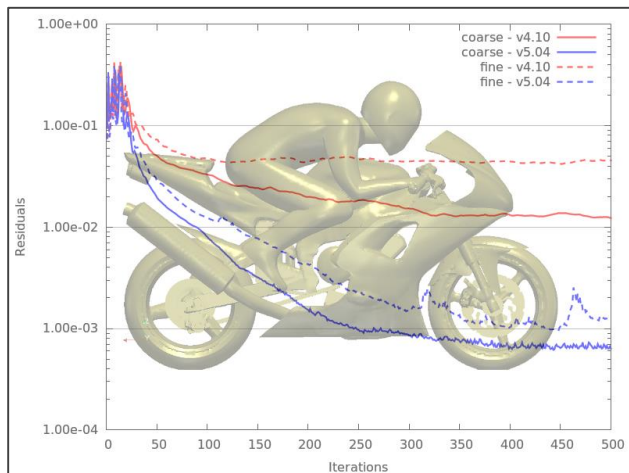


## Version 5.04 (April 27, 2015)

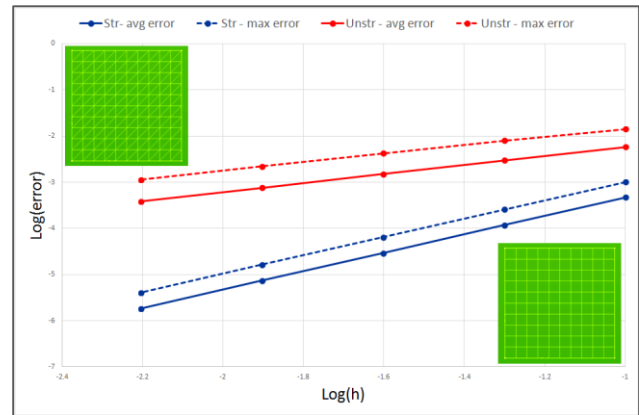
### Library enhancements

- New multi-dimensional TVD interpolation schemes. 'dc' denotes deferred correction version useful for meshes of marginal quality. The letters following 'linearUpwind' denote the particular limiter used.
  - linearUpwindBJ
  - linearUpwindDL
  - linearUpwindMC
  - linearUpwindMG
  - linearUpwindMinmod
  - linearUpwindUMIST
  - linearUpwindVanAlbada
  - linearUpwindVanLeer
  - dcLinearUpwindBJ
  - dcLinearUpwindMC
  - dcLinearUpwindMinmod
  - dcLinearUpwindUMIST
  - dcLinearUpwindVanAlbada
  - dcLinearUpwindVanLeer

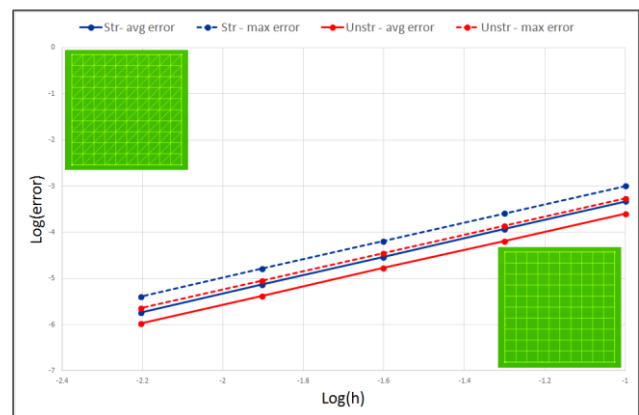


Pressure residuals for motorbike tutorial comparing *linearUpwind* scheme from v4.10 and *linearUpwindDL* scheme in v5.04

- Fixed non-orthogonal correction on boundaries



Fixed value boundary condition error convergence with v4.10



Fixed value boundary condition error convergence with v5.04

- Added new gradient scheme called 'corrGauss' that enables better accuracy for the Gauss gradient reconstruction method with fixed gradient boundary conditions
- Arbitrary Mesh Interface (AMI) upgraded to recent version (OpenFOAM 2.3)
- Sample upgraded to recent version (OpenFOAM 2.3)
- 300+ bug fixes to library and applications

### Python execution environment

- Does not require sourcing (Linux and Mac) or calling (Windows) an environment file to set up Caelus environment
- Applications executed by pre-pending 'caelus.py'
- 'caelus-log.py' utility added to extract data from log file

- Additional utilities in Caelus ./bin directory
- Derived from the FreeFoam project.

Example:

```
// Output to screen
$> caelus.py blockMesh
// Output to log file
$> caelus.py -l decomposePar
// Execute application in parallel
$> caelus.py -l simpleSolver -parallel
// Extract information from solver log file
$> caelus-log.py simpleSolver.log
```

## swak Library and Utilities

- Ported swak4Foam library and utilities to Caelus
- Expression-based specification of, for example, boundary conditions, initial conditions and source terms. Documentation and examples are provided in ./external/swak/ directory of the Caelus installation or visit the swak4Foam wiki:  
<https://openfoamwiki.net/index.php/Contrib/swak4Foam>
- Bug fixes to enable compilation on Windows

## Solvers

### *Incompressible solvers*

- `pimpleDyMSolver`: unsteady, moving mesh solver provided by the OpenFOAM Foundation

### *Multiphase solvers*

- `vofSolver`: transient, Volume of Fluid (VOF) phase-fraction based solver for 2 immiscible fluids provided by the OpenFOAM Foundation

## Models

### *Turbulence*

- Scale Adapted Simulation (SAS) based on `kwSST` (Menter et al 2003)