Coupled Density-Based Solver for High-Speed Compressible Flows

`dbnsFoam` and `dbnsTurbFoam`

Henrik Rusche, Wikki GmbH

Hrvoje Jasak, Wikki and FSB Zagreb

Karlo Šunjo, FSB Zagreb

9th OpenFOAM Workshop

23.06.2014, Zagreb, Croatia

Outline

Objective

- Describe the layout, capabilities, implementation and validation results for new coupled density-based FOAM solvers `dbnsFoam` and `dbnsTurbFoam`

Topics

1. Background
2. Review of governing equations
3. Compressible density-based coupled solver algorithm
4. `dbnsFoam` Solver Characteristics
5. Validation: Shock-wall boundary layer interaction
6. Validation: Swept ONERA M6 wing at transonic conditions
7. Summary

Compressible Navier-Stokes Equations

Compressible Formulation of Navier-Stokes Equations

- Solution variables: density ρ, momentum $\rho\mathbf{u}$ and energy ρe
- Continuity equation:

$$\frac{\partial \rho}{\partial t} + \nabla\bullet(\rho\mathbf{u}) = 0$$

  - Rate of change and convection: mass transport. The two terms are sometimes grouped into a **substantial derivative**
  - Mass sources and sinks would appear on the r.h.s.
  - Note the absence of a diffusion term: mass does not diffuse
  - Coupling with the momentum equation: rate of change of ρ depends on the divergence of momentum $\rho\mathbf{u}$
- Momentum equation:

$$\frac{\partial(\rho\mathbf{u})}{\partial t} + \nabla\bullet(\rho\mathbf{u}\mathbf{u}) - \nabla\bullet\left[\mu\left(\nabla\mathbf{u} + (\nabla\mathbf{u})^T\right)\right] = \rho\mathbf{g} - \nabla\left(P + \frac{2}{3}\,\mu\nabla\bullet\mathbf{u}\right)$$

  - Non-linear convection term: $\nabla\bullet(\rho\mathbf{u}\mathbf{u})$. This term provides the wealth of interaction present in fluid flows, e.g. vorticity, turbulence cascade
  - Diffusion term $\nabla\bullet(\mu\,\nabla\mathbf{u})$ contains viscous effects

Compressible Navier-Stokes Equations

Compressible Formulation of Navier-Stokes Equations

- Energy equation:

$$\frac{\partial(\rho e)}{\partial t} + \nabla\bullet(\rho e \mathbf{u}) - \nabla\bullet(\lambda \nabla T) = \rho \mathbf{g}\bullet\mathbf{u} - \nabla\bullet(P\,\mathbf{u}) - \nabla\bullet\left(\frac{2}{3}\,\mu(\nabla\bullet\mathbf{u})\,\mathbf{u}\right)$$
$$+\nabla\bullet\left[\mu\left(\nabla\mathbf{u} + (\nabla\mathbf{u})^T\right)\bullet\mathbf{u}\right] + \rho Q,$$

  - Weaker coupling to the rest of the system: e and T influence ρ and $\mathbf{u}$ through the equation of state

Density-Based Algorithm

Block-Coupled Density-Based Solver

- Noting that all governing equations fit into the standard form and all variables are fully coupled, the compressible Navier-Stokes system can be written as:

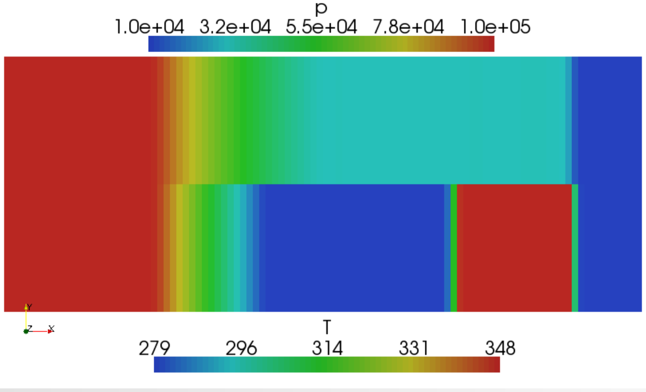
$$\frac{\partial U}{\partial t} + \nabla \bullet F - \nabla \bullet V = R$$

where the solution variable U is:

$$U = \begin{bmatrix} \rho \\ \rho \mathbf{u} \\ \rho e \end{bmatrix}$$

- Pressure appears in the convective flux F and V is the diffusive flux:

$$F = \begin{bmatrix} \rho \mathbf{u} \\ \rho \mathbf{u}\mathbf{u} + p\mathbf{I} \\ \rho(e+p)\mathbf{u} \end{bmatrix} \quad V = \begin{bmatrix} 0 \\ \boldsymbol{\sigma} \\ \boldsymbol{\sigma} \bullet \mathbf{u} - \mathbf{q} \end{bmatrix}$$



Density-Based Algorithm

Block-Coupled Density-Based Solver

- Coupled solver of this type is a standard work-horse of aerospace CFD: numerics is well understood and validated
- Basic implementation carries the Courant number limit
- Godunov-type compressible Navier-Stokes solver will evaluate F for each cell face directly from the state (U) left and right from the face, using approximate Riemann solver techniques

What About Equation Segregation?

- This is feasible only for weakly coupled equation sets: in full coupling, either the shocks will not be appropriately resolved or the entropy condition will be violated
- Why does FOAM not already have a coupled density-based solver?
  We went to the wrong school!

`dbnsFoam` Solver

Objective: **Textbook Implementation of a Block-Coupled Density-Based Solver**

- Implementation of a Godunov-like solver
- 1st and 2nd order spatial accuracy
- Multi-stage Runge-Kutta time stepping
- Boundary conditions operate on primitive variables
  - Compatible with existing pressure-based solvers (segregated and coupled)
  - Riemann solver is used with boundary state vector to compute conservative fluxes at boundary
- Interface to thermo-physical modelling library
- Integration of standard turbulence modelling library: segregated from $\rho - \mathbf{u} - h$ coupling and implicit. Used without modification; diffusive fluxes are explicit in the $\rho - \mathbf{u} - h$ coupled system
- Support for coupled implicit boundaries: processor (HPC), GGI, mixing plane

`dbnsFoam` **Solver**

Objective: Textbook Implementation of a Block-Coupled Density-Based Solver

- Choice of approximate Riemann solver
  - Roe & Pike scheme with Harten's entropy fix
  - HLLC formulation by Batten et.al.
  - Beta flux scheme
  - Rusanov (Local Lax-Friedrichs) flux
- Choice of gradient limiter
  - Barth-Jespersen limiter
  - Venkatakrishnan differentiable limiter
  - Multi-dimensional limiter
- Choice of boundary conditions: static and rotating total state conditions

Density-Based Algorithm

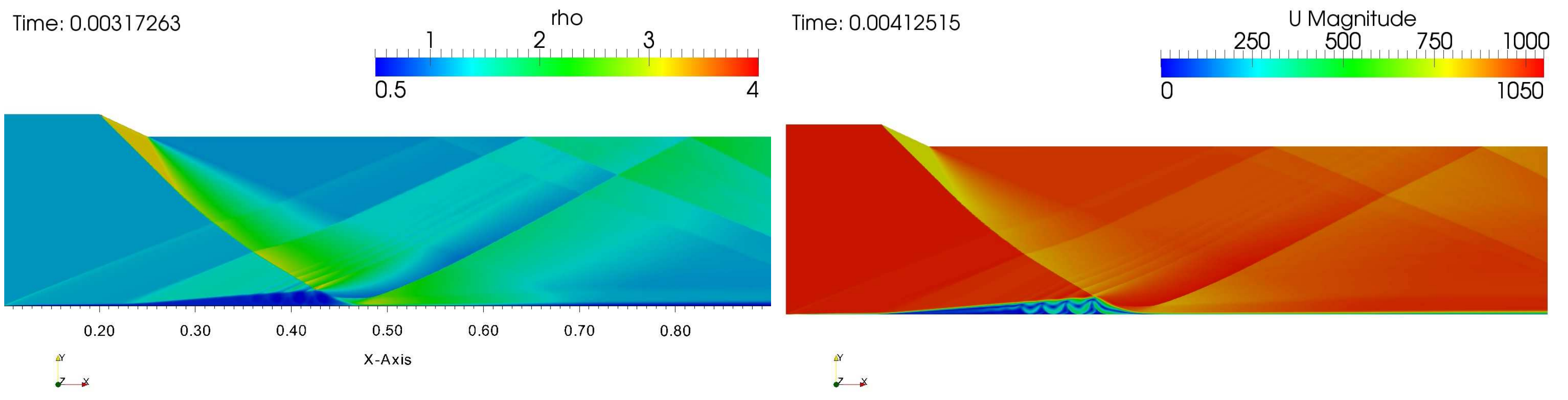
Block-Coupled Density-Based Solver: **Work-in-Progress**

- Extension of MRF and SRF models for compressible turbomachinery simulations
- Some of the flux formulations have been extended to moving meshes (ALE)
- **Acceleration techniques for steady-state**
  - Local and Dual Time Stepping
  - Implicit solution of $\rho - \mathbf{u} - h$ system using block matrix. Coupling coefficients obtained with automatic differentiation tools
  - Full Approximation Storage Geometric Multigrid
- Full run-time selection for choice of approximate Riemann solver, gradient limiters and other discretisation options
- Implementation of further approximate Riemann solvers and boundary conditions
- Testing and validation: in progress, University of Zagreb

Validation: SWBLI

Validation: Shock-Wall Boundary Layer Interaction

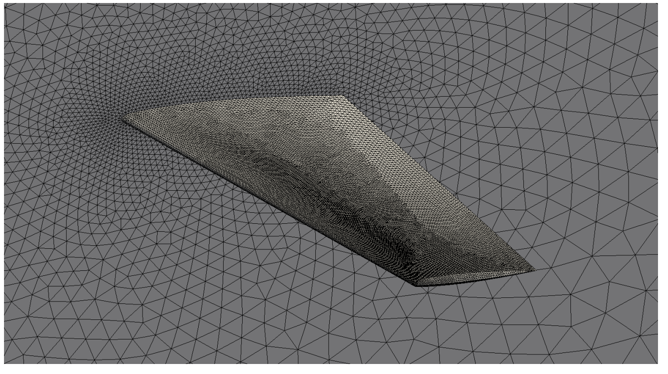
- Inlet Mach number = 2.94
- Re = 136 000, (assuming laminar flow)
- Interaction between the shock structure and viscous boundary layer creates complex flow structure: shock cannot touch the wall

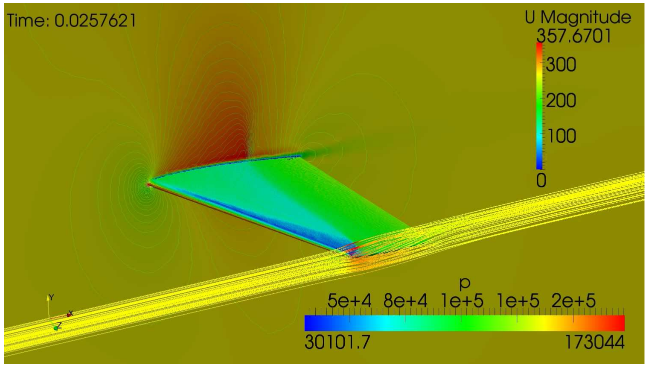


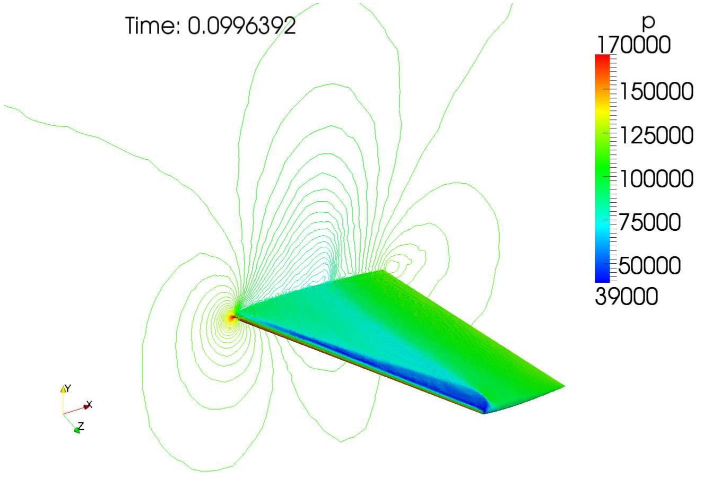
Validation: ONERA M-6 Wing

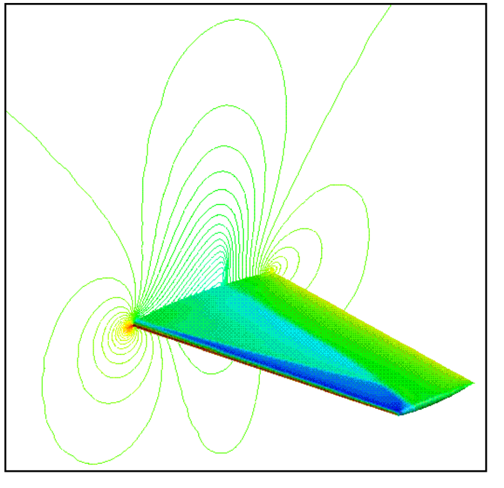
Validation: ONERA M-6 Swept Wing: NASA NPARC Validation example

- Tetrahedral mesh, 341 797 cells
- Air flow, Ma = 0.8395, AoA = 3.06 deg
- Comparison of wing pressure against reference CFD solution (bottom right)









Summary

Summary: First Release of the Coupled Density-Based Solver for High-Speed Flows

- The first version of the `dbnsFoam` and `dbnsTurbFoam` solvers released
- Standard discretisation options provided, first and second-order accuracy
- Integration with FOAM libraries: thermo-physical properties, turbulence models, boundary conditions
- Efficiency improvements in the development pipeline
- Further work and validation required: looking for partners and industrial validation cases

Credits

Work on the coupled density-based solver has been performed by a number of people mostly initiated through the NUMAP/FOAM Summerschool. Their contribution is hereby acknowledged.

- Oliver Borm
- Aleksandar Jemcov
- Sebastian Saegeler