

Agenda Programming CFD in OpenFOAM

Teachers:

Simone Bnà, Francesco Bottau, Ettore Fadiga, Giuseppe Giaquinto, Tommaso Zanelli

Day 1

- **09:30 – 10:00 HPC introduction (Tommaso Zanelli):** ssh connection, vscode configuration (vi for advanced user), module load, login on reservation nodes
Tutorial 1: connection to the cluster, run vscode and check installation
- **10:00 – 10:45 Getting started with C++ (Giuseppe Giaquinto):** creating a C++ program, compilation (wmake), macro, function overloading, scope, namespace, header files, class, inheritance, alias, template programming
Tutorial 2: write your own first program (Hello World)

Break 15 min

- **11:00 – 12:00 OpenFOAM core classes (container) (Giuseppe Giaquinto):** introduction, scalar, vector, tensor, List, Field, word (optional: pTraits, One, Zero)
Tutorial 3: examples of algebra operations on foam native objects, example of forAll to fill in array, parallelism between List and array in C
- **12:00 - 13:00 Optional models (Giuseppe Giaquinto):** autoPtr, memory allocation, pointer de-referencing, pointer validity
Tutorial 4: examples with tmp objects using algebra operations

13:00-14:30 Lunch

- **14:30 – 15:30 OpenFOAM init case (create OpenFOAM context) (Ettore Fadiga):** setRootCaseList, create Time, create fvMesh (OpenFOAM mesh description)
Tutorial 5: Hello word with a simple time loop without geometric Field initialization

Break 15 min

- **15:45 – 16:45 Data construction (Ettore Fadiga):** I/O registry, Dimensioned Field, GeometricField (volume and surface), boundaryField, fvPatchField (few notes on processorPatchField)
Tutorial 6: creation of a geometricField reading input from disk

Break 15 min

- **17:00 - 18:00 Interfaces (Ettore Fadiga):** library-application interface, model update and delivery, virtual functions, run-time selection
Tutorial 7: add a patch with run-time selection (e.g. parabolic inflow)
-

Day 2

- **09:30 - 10:30 Data access (Francesco Bottau):** data access in loops (face addressing, loop over faces, loop over cells)
Tutorial 8: examples of traversing algorithms

Break 15 min

- **10:45 - 11:45 Data operations (Francesco Bottau):** reduce, sum, average, scan (few notes on parallel aspects)
Tutorial 9: examples of usage of the scan algorithm to compute the *faceStart* list
- **11:45 - 13:00 FV equations (Francesco Bottau):** lduMatrix, fvMatrix
Tutorial 10: operations with matrices

13:00-14:30 Lunch

- **14:30 – 15:45 Derivatives and algebra (Tommaso Zanelli):** fvm, fvc, Laplace operator, Div operator, Time-derivative operator, Grad operator
Tutorial 11: assembly momentum equation

Break 15 min

- **16:00 - 16:15 Introduction to Tutorial 12 (Tommaso Zanelli)**
 - **16:15 – 18:00 Tutorial 12 (Tommaso Zanelli):** Write a custom grad operator using loop over faces (faceList previously computed) instead of a loop over cells
-

Day 3

- **09:30 – 10:45 Solution algorithms for incompressible Navier-Stokes (Simone Bnà):** Navier-Stokes, segregated vs coupled: splitting U-p, PISO, SIMPLE, PIMPLE
Tutorial 13: add PISO loop to the previous solver

Break 15 min

- **11:00 - 12:30 Solvers (Simone Bnà)**
 - **Solvers (I):** lduMatrix::solver, Amul (few notes on parallel aspects), PCG, preconditioning
 - **Solvers (II):** csrMatrix, ldu2csr conversion, AmgX4Foam e petsc4foam design aspects

12:30-14:00 lunch

- **14:00 - 15:00 OpenFOAM GPU porting (Simone Bnà):** OpenFOAM design limitations, zeptoFOAM design concepts (e.g Memory pool)
- **15:00 - 18:00 Tutorials (Simone Bnà)**
Tutorial 14: Convert the matrix from LDU to CSR and write your own PCG solver using a user-defined CSR-based Amul
Tutorial 15: Solve the SIMPLE/PISO loop using our custom grad and custom solve