

Parallel simulations and using clusters

Fluidity Training Course 2014

Michael Lange

AMCG, Imperial College London

6th November 2014

Parallel Simulations

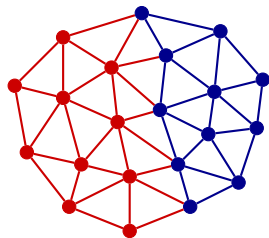
Large simulations require more than a laptop

- ▶ Fluidity runs in parallel via MPI
- ▶ Optional: OpenMP threading for hybrid runs
 - ▶ Configuration option `--enable-openmp`
- ▶ Fluidity runs on various clusters
 - ▶ Imperial College: CX1, CX2 (Intel)
 - ▶ UK national supercomputer: Archer (Cray CX30)
 - ▶ ANU/NCI: Raijin (Fujitsu Primergy)

Parallel Simulations

How to run Fluidity in parallel:

- ▶ (Almost) no changes to the .flml required!
 - ▶ Parallel compatible preconditioner (not eisenstat)
 - ▶ Optional: Remove fields from output (.stat and .vtu)
- ▶ Need to decompose the mesh
 - ▶ Tools provided:
`flredecomp`, `fldecomp`
 - ▶ Build both with: `make fltools`



Mesh decomposition

Mesh decomposition before the run:

- ▶ `mpiexec -n 4 flredecomp -i 1 -n 4 serial parallel`
 - ▶ Needs to be run in parallel!
 - ▶ Decomposes mesh named in `serial.flml`
 - ▶ Creates new parallel setup: `parallel.flml`
- ▶ Now run this on 4 processors:
`mpiexec -n 4 fluidity parallel.flml`

Mesh decomposition

Mesh decomposition with checkpoints:

- ▶ Switch on checkpointing!
 - ▶ `/io/checkpointing` in `.flml`
- ▶ `mpiexec -n 8 flredecomp -i 4 -n 8 par_4 par_8`
 - ▶ Needs to be run on `max(-i, -o)` processes
 - ▶ Enables re-starting on more/fewer processes
- ▶ Run as before:
`mpiexec -n 8 fluidity par_8.flml`

Running Fluidity on clusters

- ▶ Set up simulation on local machine!
- ▶ Install Fluidity on cluster
 - ▶ CX1/CX2: module load fluidity
 - ▶ Archer: instructions here
- ▶ Create .pbs submission script
 - ▶ Submit with qsub
- ▶ Copy results back to local machine
 - ▶ Visualisation as before

Archer

```
#!/bin/bash —login
#PBS -l walltime=01:00:00
#PBS -N jet_sim
#PBS -l select=2
#PBS -A n01-IC1

# Load Fluidity environment
module swap PrgEnv-cray PrgEnv-fluidity

# Make sure any symbolic links are resolved to absolute path
export PBS_O_WORKDIR=$(readlink -f $PBS_O_WORKDIR)
cd $PBS_O_WORKDIR

# Workaround for python environment
export LD_LIBRARY_PATH=$ANACONDA_LIB:$LD_LIBRARY_PATH

export FLUIDITYDIR=path/to/my/fluidity/install

# Decompose the mesh
aprun -n 48 -N 24 -d 1 -j 1 $FLUIDITYDIR/bin/flredecomp -i 1 -o 48 jet_sim set_sim_48

# Run simulation
aprun -n 48 -N 24 -d 1 -j 1 $FLUIDITYDIR/bin/fluidity -l -p -v2 jet_sim_48.flml
```