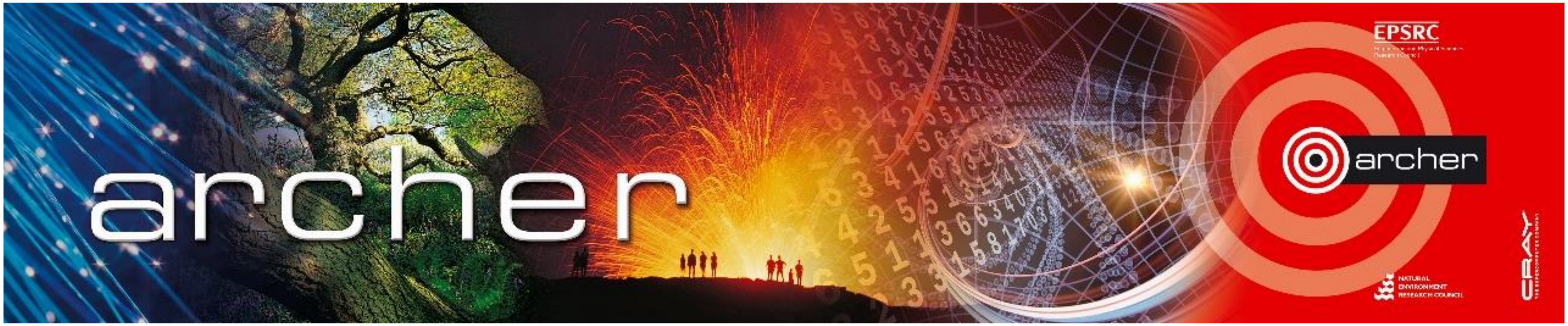


Hints on using OpenFOAM on ARCHER

David Henty

11th October 2017

Overview

- OpenFOAM is installed centrally
  - see http://www.archer.ac.uk/documentation/software/openfoam/
  - pre/post-processing are non-parallel versions for serial nodes
  - contact ARCHER helpdesk if you need other versions

OpenFOAM version	Compute nodes	Pre/post-processing nodes	Compilation instructions
2.1.1	no	no	yes
2.1.X	no	no	yes
2.2.2	yes	yes	yes
2.3.0	yes	no	no
2.4.0	yes	no	yes
3.0.1	yes	no	yes
4.1	yes	yes	yes

Serial or parallel?

- Compute nodes have MPI + vectorisation (AVX) - use for
  - parallel solvers (e.g. icoFoam)
  - parallel utilities (e.g. snappyHexMesh)
  - serial utilities as part of a job (e.g. blockMesh)

- Pre/post-processing nodes have lots of memory – use for
  - pre- and post-processing that requires large amounts of memory
  - e.g. reconstructPar

- See documentation for how to access different versions

OpenFOAM performance

- Major issue on ARCHER is file IO
- Each process in an OpenFOAM parallel simulation writes one file for each output field at each output time:
  - *number of files = number of output fields* x *number of output times* x *number of processes*
  - a large number of small files
- But ARCHER has a Lustre file system
  - optimised for reading and writing small numbers of large files
  - opening and closing large numbers of files can be slow
  - large numbers of processes reading or writing files can contend for access to the file system

Suggestions

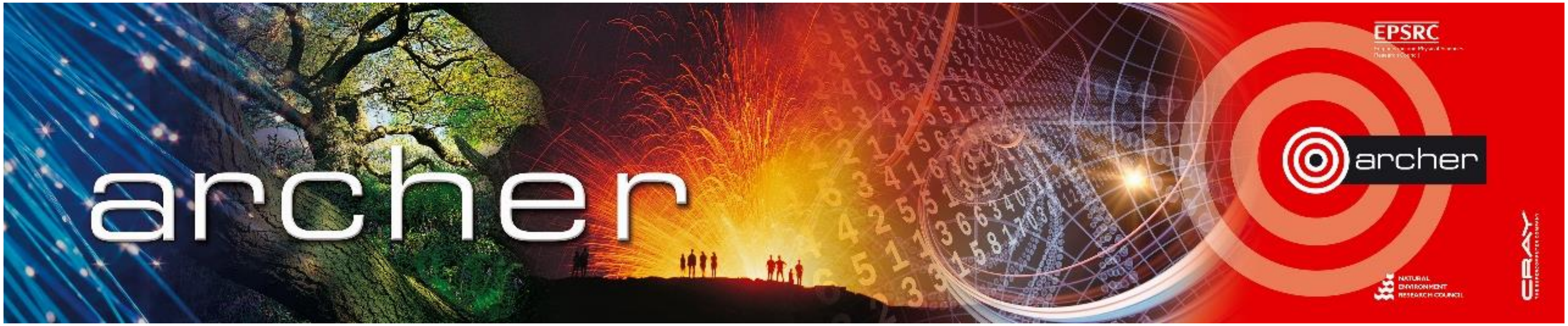
- Always clear up after a run to remove unnecessary files
  - ARCHER filesystem filling up after many years' usage ...
  - *don't* stripe files across multiple disks (now the default setting)

- Alter settings in **`controlDict`**
  - increase **`writeInterval`**
  - binary format for the fields: **`writeFormat binary`**
  - for steady-state solutions, overwrite each time: **`purgeWrite 1`**

- May be slow to start if many shared libraries
  - contact helpdesk to investigate DLFM package

If in doubt

- Check documentation
  - http://www.archer.ac.uk/documentation/software/openfoam/
- Contact helpdesk
  - support@archer.ac.uk



Goodbye!