OpenFOAM

Purpose: Computational Fluid Dynamics
Latest version: v2112
Licence: Free of use
GNU GPL _ext-link
Website: *http://www.openfoam.com/ _ext-link*

OpenFOAM is a software environment for computational fluid dynamics (CFD) and related computations.

OpenFOAM provides a C++ library and tools for setting up, meshing, solving, computing properties, and post-processing continuum mechanics, especially computational fluid dynamics. OpenFOAM includes solvers for a number of applications and provides an environment in which users can build their own.

SLURM Submit script example

More information about the submit script can be found using the ***Job Script Generator***.

Sbatch options:

The options shown in the example are detailed below. For more information and a more comprehensive list of available options, see the ***sbatch command page***.

- **-J**: Name for the job's allocation.
- **-e**: Name of the stderr redirection filename.
- **-o**: Name of the stdout redirection filename.
- **-p**: Name of the partition (queue) where the job will be submitted.
- **-n**: Number of tasks.

- **-c**: Number of cores per task.
- **-t**: Set the job's time limit. If the job don't finish before the time runs out, it will be killed.
