OpenFOAM

OpenFOAM versions 2.3.0 and 17.12 are available on the cluster nodes. To use them, use the following commands:

```
module load openfoam/v1712
source $FOAM
```

If you wish to use OpenFOAM 2.3.0, replace v1712 with 2.3.0

After running these command the OpenFOAM commands are available for use.

From:
https://grid.fe.up.pt/dokuwiki/ - **GRID FEUP**

Permanent link:
https://grid.fe.up.pt/dokuwiki/doku.php?id=documentation:software:openfoam

Last update: **2018/06/25 18:12**