

# OpenFOAM

## Cavity

- double the resolution

## Dam break

- copy the dam break tutorial and run it on the cluster (up to post-processing)

<https://cfd.direct/openfoam/user-guide/v6-dambreak/>

- do a parallel run (running in parallel section – increasing resolution is optional)

## ParaView

- extracting just girder
- extracting a plane
- glyphs
- graphs (multiple displays)
- plot over line
- plot over time (sample data)
- enabling LIC plugin and showing on plane
- clipping off the sides
- enabling volume rendering for low pressure regions
- stream tracers, tubes, and glyphs