OpenFOAM

blocked URL

Opis

OpenFOAM je besplatan i otvoreni softver za numeričko rješavanje problema u računalnoj dinamici fluida i srodnim disciplinama. OpenFOAM ima široku primjenu u industriji, akademskim i istraživačkim institucijama. Može se koristiti za modeliranje protoka fluida u raznim sustavima.

Verzije

Verzija	Modul	Red	Supek	Padobran	MPI
2212	module load scientific/openfoam/2212	cpu	✅	❌	cray-MPICH - 8.1.20
2312	module load scientific/openfoam/2312	cpu	✅	❌	openMPI - 5.0.1
10	module load scientific/openfoam/10	cpu	✅	❌	cray-MPICH - 8.1.20
11	module load scientific/openfoam/11	cpu	✅	❌	openMPI - 5.0.1

Službena dokumentacija

- https://doc.cfd.direct/openfoam/user-guide-v10/index
- https://www.openfoam.com/documentation/user-guide
- https://www.openfoam.com/documentation/tutorial-guide

Primjer pokretanja posla (Supek)

Svaki OpenFOAM posao započinje sa učitavanjem OpenFOAM modula i postavljanjem odgovarajućeg okuženja naredbama:

module load scientific/openfoam/2212

source $OF_activate

. ${WM_PROJECT_DIR:?}/bin/tools/RunFunctions

OpenMPI-5.0.1

Ukoliko koristite verzije openfoam-aa prevedene sa openMPI-5.0.1, prilikom pozivanja naredbe mpiexec potrebno je dodati i **--hostfile $PBS_NODEFILE** opciju.

Primjer snappyHexMesh:

mpiexec --hostfile $PBS_NODEFILE snappyHexMesh -parallel

U primjeru niže pokrenuti će se posao na 128 jezgri, u "cpu" redu čekanja.

primjer_OF.pbs

```
    #!/bin/bash
#PBS -N run-OF
#PBS -q cpu
#PBS -j oe
#PBS -l select=128
#PBS -l place=pack

cd ${PBS_O_WORKDIR}

module load scientific/openfoam/2212
source $OF_activate
. ${WM_PROJECT_DIR:?}/bin/tools/RunFunctions

surfaceFeatureExtract
blockMesh
decomposePar

mpiexec snappyHexMesh -parallel
mpiexec topoSet -parallel
restore0Dir -processor
patchSummary
mpiexec potentialFoam -writephi -parallel
mpiexec checkMesh -writeFields '(nonOrthoAngle)' -constant -parallel
mpiexec simpleFoam -parallel
reconstructParMesh -constant
reconstructPar -latestTime
```