

# 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

## 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 okruženja 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
```