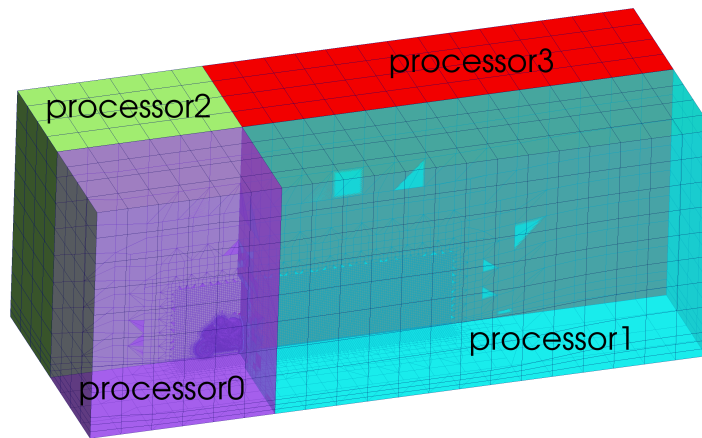


Laboratory Exercise: Parallel Simulation with OpenFOAM

We will use the `motorBike` tutorial case to investigate the parallel execution capabilities in OpenFOAM. Parallel grid generation and simulation is achieved using the concept of domain decomposition. To perform a parallel computation, the following steps are typically taken:

- (1) Decompose the mesh and fields (velocity, pressure, etc)



- (2) Run the application in parallel (`icoFoam`, `snappyHexMesh`, etc), for example:

```
foamJob snappyHexMesh -parallel
```

- (3) Post-process the decomposed case or reconstruct the mesh and fields at the latest time using:

```
reconstructPar -latestTime
```

A summary of parallel execution with OpenFOAM can be found at:

<https://cfd.direct/openfoam/user-guide/v6-running-applications-parallel>

- (1) Use the `motorBike` case from the grid-generation lab and generate the grid in parallel. The typical CAEN machine has 4 processors. Compare the time necessary to generate grid in serial and in parallel using 2 and 4 processors.

- (2) Use the `motorBike-stampede2` case to generate the grid and solve the flow in parallel using a high-performance computing (HPC) center Stampede2. For details on the usage of Stampede2 visit <https://portal.tacc.utexas.edu/user-guides/stampede2>. Experiment with the number of processors used to evaluate the case. Reconstruct the results and copy them to your local workstation to visualize in `paraFoam`.

Stampede2 101:

- (1) Log in to Stampede2 using SSH in a terminal:
`ssh TACCusername@stampede2.tacc.utexas.edu`
 - Enter your TACC account password and TACC token code for two-factor authentication
- (2) Navigating the Stampede2 file system:
 - `cdh`: enter home space (storage up to 10GB, backed up, not purged, not intended for parallel or high-intensity file operations)
 - `cds`: enter scratch space (large capacity without a quota where you should run your cases; not backed up, subject to purge if access time is more than 10 days old)
 - `cdw`: enter work space (large capacity of 1TB, not backed up, not purged, not intended for high-intensity file operations or jobs involving very large files)
- (3) Transferring files:
 - To transfer file or directory to your home space on Stampede2 from your local workstation:
`scp -r pathToFileOrDir TACCusername@stampede2.tacc.utexas.edu:~/.`
 - To transfer from Stampede2 to your local workstation:
`scp -r TACCusername@stampede2.tacc.utexas.edu:pathToFileOrDir ~/.`
- (4) Load modules on Stampede2 just like on CAEN workstations:
`module load module-name`
- (5) Load OpenFOAM by sourcing the `openfoam` file:
`source openfoam`
 - This is a shortcut to load all the necessary modules required to use OpenFOAM-5.0 (note this is a newer version than OpenFOAM-2.4.x on CAEN workstation)
- (6) Running jobs on Stampede2: Submit the `job.sub` submission file to the scheduler from your case directory in the scratch space:
`sbatch job.sub`
 - Edit `job.sub` to change the job name (`#SBATCH -J`), number of processors and nodes (`#SBATCH -n`, `#SBATCH -N`), time allocation (`#SBATCH -t`), queue (`#SBATCH -p`), email, applications to execute, etc
- (7) Monitor job status in the queue:
`showq -u`
- (8) Cancel job in queue:
`scancel jobid`

Example of copying the `motorBike-stampede2` case and the `openfoam` source file to Stampede2 and submitting a parallel job:

```
scp -r motorBike-stampede2 openfoam tg854073@stampede2.tacc.utexas.edu:~/.  
ssh -Y tg854073@stampede2.tacc.utexas.edu  
source openfoam  
cp -r motorBike-stampede2 $SCRATCH/.  
cds  
cd motorBike-stampede2  
sbatch job.sub
```