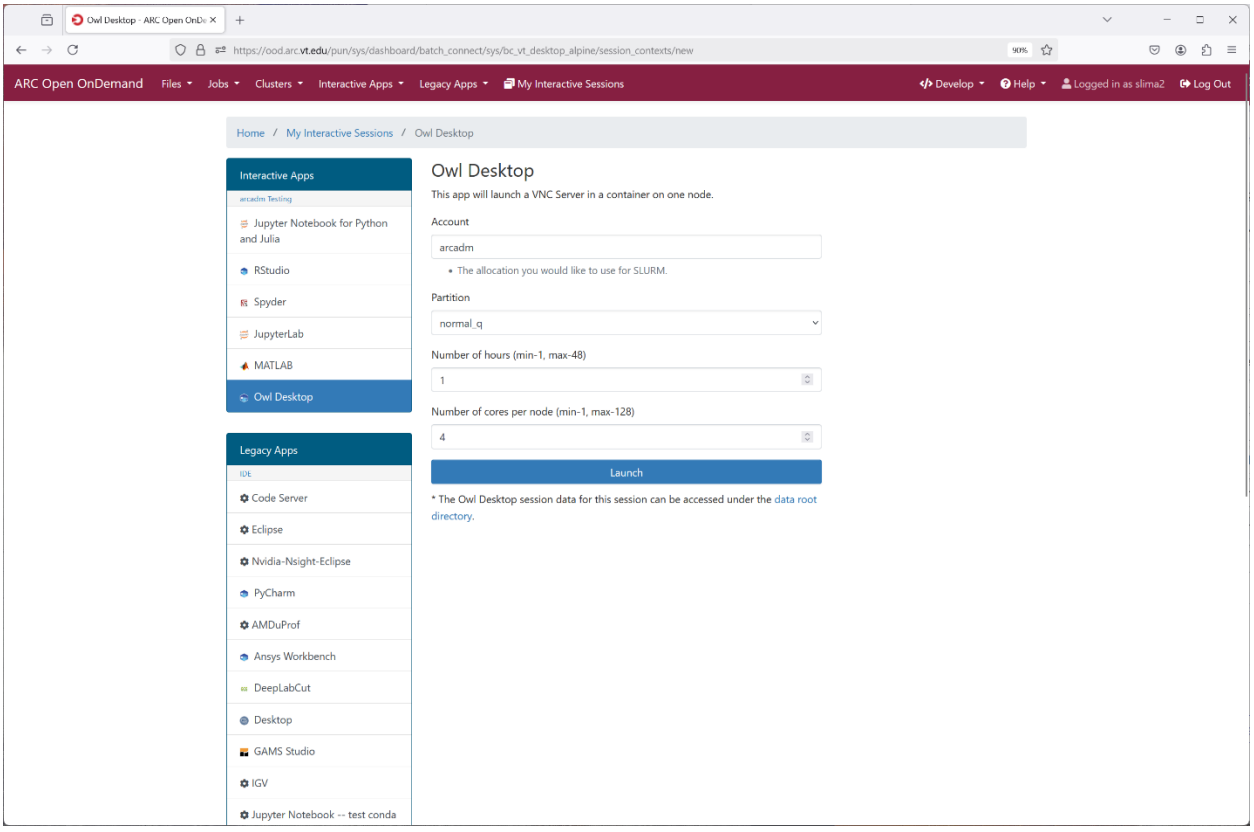
OpenFOAM-12 tutorial with Desktop App

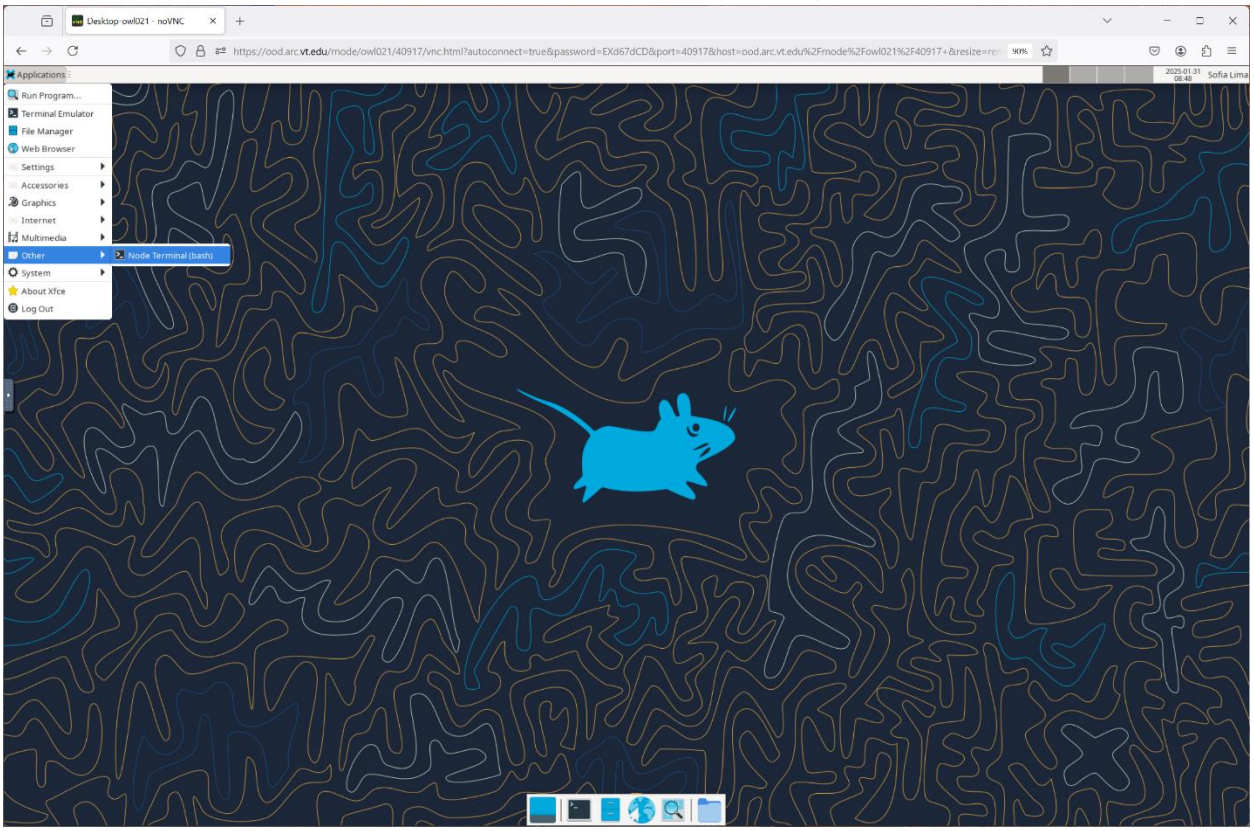
Using the Desktop App is one way to run OpenFOAM interactively. This can be a very useful tool during the early stages of design and development. It is recommended to run large simulations in batch mode.

Design your simulation

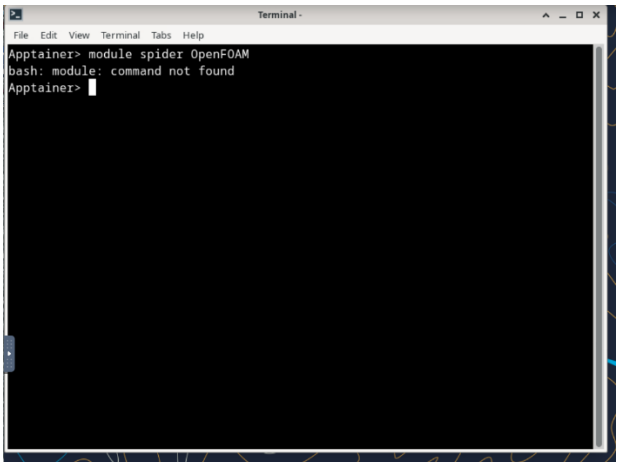
We will be using a simple incompressible Fluid example provided with the installation. We will copy the necessary data into our working directory. This example is designed to run with 4 processes in parallel; thus we will request 4 CPU processors (aka “cores” or “chips”).

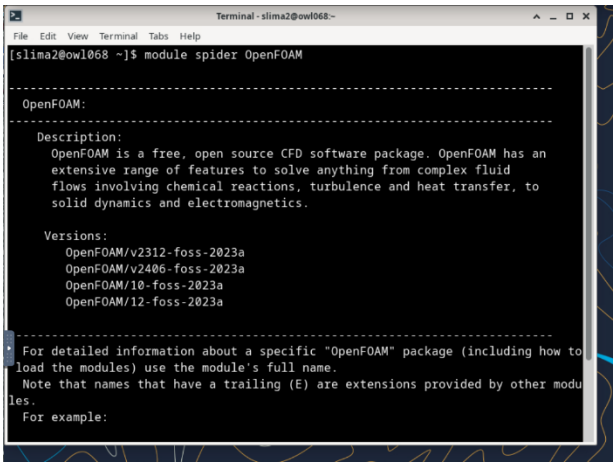


We will use the Node Terminal to execute commands interactively.



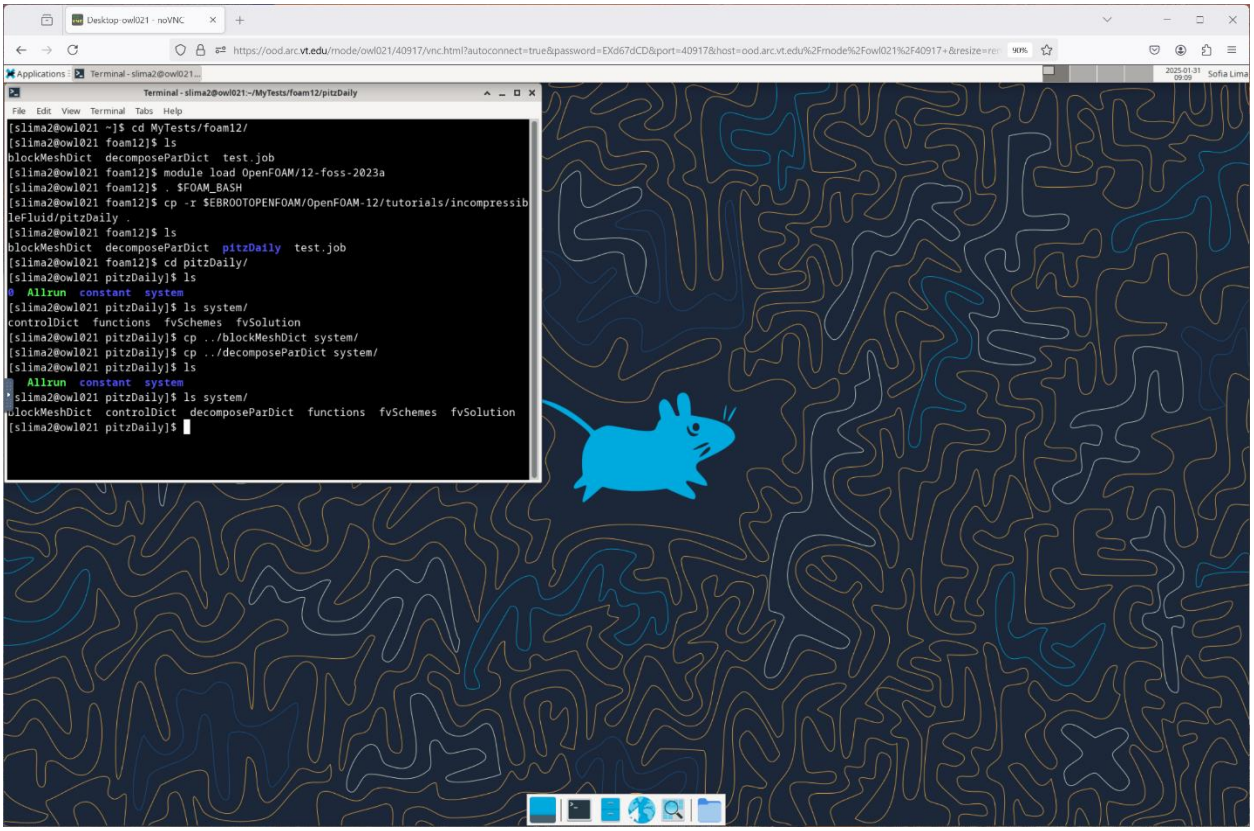
This terminal gives us access to a shell on the compute node with X-forwarding to allow for graphical rendering. This is different from the Terminal Emulator, which is a shell inside the container with “Apptainer>” on the command prompt.





Setup your working directory

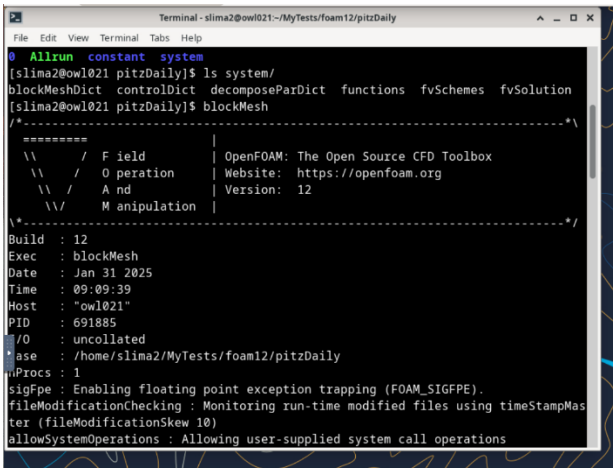
First we will load the OpenFOAM software. Loading this module will define some environment variables, including $FOAM_BASH and $EBROOTOPENFOAM. OpenFOAM is unusual because we must source (.) the bash script $FOAM_BASH in order to call the openfoam executables like blockMesh, decomposePar, and foamRun. Then, we will the copy the example data into our working directory.

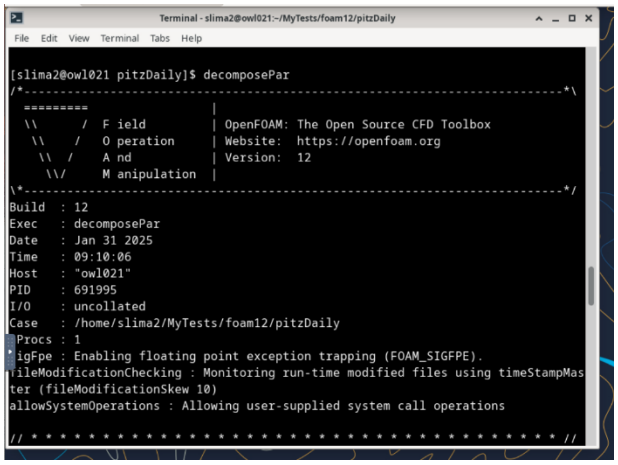


Run OpenFOAM

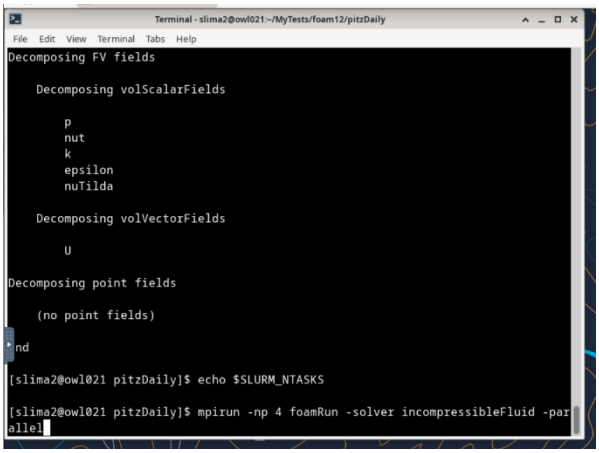
We will use openfoam commands to achieve our computational goals. We will follow the process to accelerate runtime by executing our simulation across multiple cores in parallel. This experiment is designed to run on 4 blocks of the mesh with 1 processor for each.

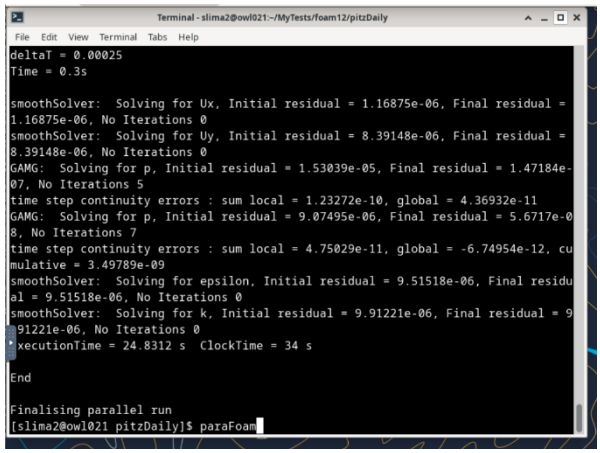
First we will run openfoam to block the mesh and decompose for parallel execution.



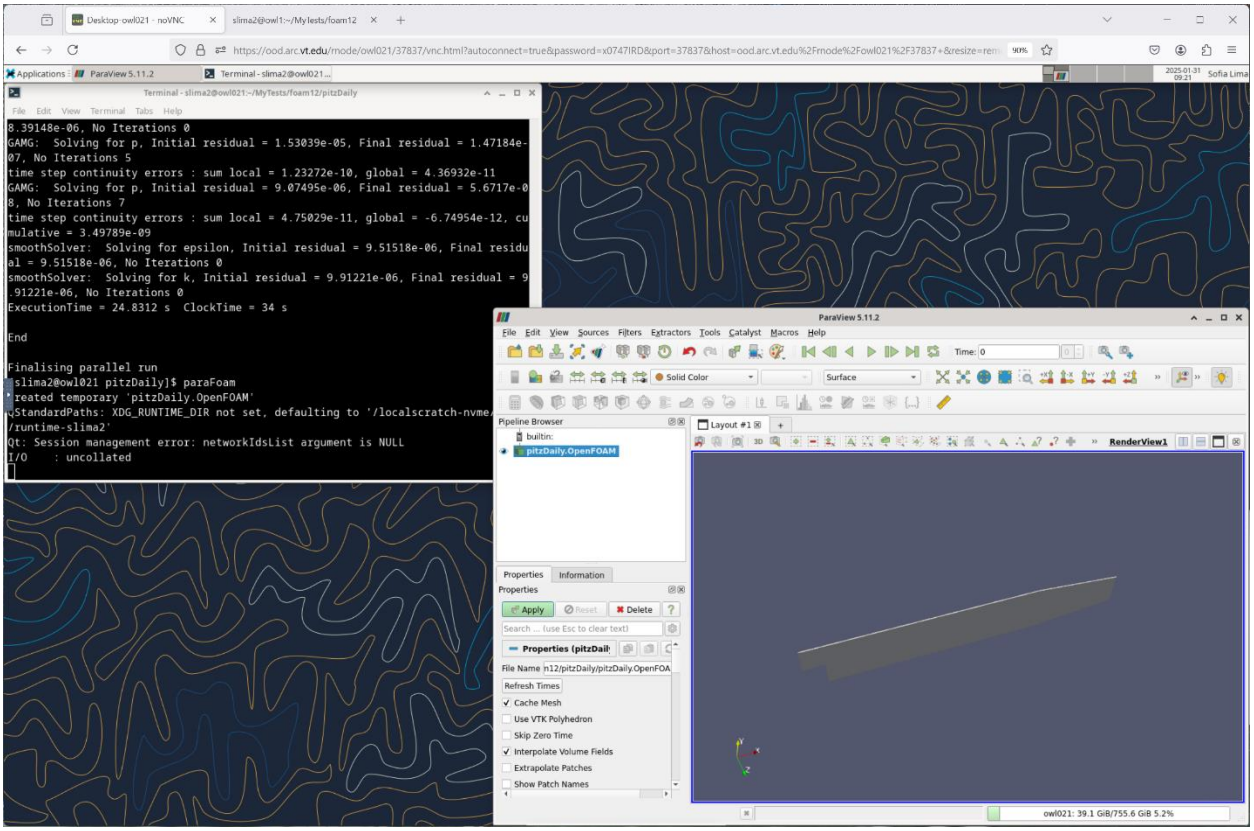


Now we will run our simulation in parallel with OpenMPI. We will see the log in standard output; we could have redirected the output to a file by adding `> run_output.log` to the end of the command.





Because we are on the Desktop app which gives us the access to a shell with X-forwarding, we can use the paraFoam command to visualize the results.


Desktop-owl021 - noVNC
slima2@owl1:~/Mytests/foam12
https://ood.arc.vt.edu/mode/owl021/37837/vnc.html?autoconnect=true&password=x0747IRD&port=37837&host=ood.arc.vt.edu%2Fnode%2Fowl021%2F37837+&resize=rem
Applications
ParaView 5.11.2
Terminal - slima2@owl021...
2025-01-31 09:21
Sofia Lima
Terminal - slima2@owl021:~/MyTests/foam12/pitzDaily
File Edit View Terminal Tabs Help
8.39148e-06, No Iterations 0
GAMG: Solving for p, Initial residual = 1.53039e-05, Final residual = 1.47184e-07, No Iterations 5
time step continuity errors : sum local = 1.23272e-10, global = 4.36932e-11
GAMG: Solving for p, Initial residual = 9.07495e-06, Final residual = 5.6717e-08, No Iterations 7
time step continuity errors : sum local = 4.75029e-11, global = -6.74954e-12, cumulative = 3.49789e-09
smoothSolver: Solving for epsilon, Initial residual = 9.51518e-06, Final residual = 9.51518e-06, No Iterations 0
smoothSolver: Solving for k, Initial residual = 9.91221e-06, Final residual = 9.91221e-06, No Iterations 0
ExecutionTime = 24.8312 s ClockTime = 34 s
End
Finalising parallel run
slima2@owl021 pitzDaily]$ paraFoam
Created temporary 'pitzDaily.OpenFOAM'
StandardPaths: XDG_RUNTIME_DIR not set, defaulting to '/localscratch-nvme/runtime-slima2'
Qt: Session management error: networkIdsList argument is NULL
I/O : uncollated
ParaView 5.11.2
File Edit View Sources Filters Extractors Tools Catalyst Macros Help
Time: 0
Solid Color
Surface
Pipeline Browser
builtin:
pitzDaily.OpenFOAM
Layout #1
RenderView1
Properties
Information
Properties
Apply
Reset
Delete
Search ... (use Esc to clear text)
Properties (pitzDail
File Name n12/pitzDaily/pitzDaily.OpenFOA
Refresh Times
Cache Mesh
Use VTK Polyhedron
Skip Zero Time
Interpolate Volume Fields
Extrapolate Patches
Show Patch Names
owl021: 39.1 GiB/755.6 GiB 5.2%