Fluid Flow through Porous Media Simulation Scalability with OpenFOAM and MPI

Erik Jensen\textsuperscript{1}, Mayank Tyagi\textsuperscript{2,3}, Zhi Shang\textsuperscript{3}

\textsuperscript{1}Department of Modeling, Simulation, and Visualization Engineering; Old Dominion University; Norfolk, Virginia 23529
\textsuperscript{2}Craft and Hawkins Department of Petroleum Engineering; Louisiana State University; Baton Rouge, Louisiana 70803
\textsuperscript{3}Center for Computation and Technology; Louisiana State University; Baton Rouge, Louisiana 70803

Fluid Flow through Porous Media

- Petroleum engineers study the motion of hydrocarbons in porous media (e.g., sandstone).
- Although rocks appear to be solid, they contain a significant amount of pore space that can harbor fluid (e.g., crude oil).
- When oil is extracted, pore space facilitates transport.
- Porosity quantifies the amount of pore space.
- Permeability describes resistance to fluid flow.

Computational Fluid Dynamics (CFD)

- The velocity and pressure of a Newtonian fluid are governed by the Navier-Stokes equations.
- Momentum: $\frac{\partial p}{\partial t} + \frac{\partial}{\partial x_i} \left( \rho u_i u_j \right) = \frac{\partial}{\partial x_i} \left( \mu \frac{\partial u_i}{\partial x_j} \right) + \frac{\partial p}{\partial x_i} + S_{M_{ij}}$.
- Solutions must be found computationally.
- The momentum equations are approximated by discretized equations (e.g., for $u_1$):
  $a_{ij} u_{1,j} = \sum a_{i,j} (u_{1,i} - p_{1,i}) + b_{ij}$.
- The domain is discretized into a staggered grid that stores velocity and pressure information, separately, at specific points in and between cells.
- Velocities are stored at cell faces.
- Pressure is stored at cell centers.
- In practical applications, the grid contours to the model surface. Cells can assume non-cubic shapes.

OpenFOAM and Distributed Simulation

- OpenFOAM (Open source Field Operation and Manipulation) is a CFD software package for Linux.
- Open MPI (Message Passing Interface) is used to run simulations in parallel.
- Parallelization is running a simulation on more than one processor at a time.
- Scalability is a measure of how well a simulation parallelizes.
- CFD simulations are parallelized by splitting up the physical domain and distributing the pieces to different processors or computers.

Procedure

1. Code a script (Python) to generate a three-dimensional porous media model. The geometry format is Stereolithography (STL). The file format is binary. Run the script to generate the geometry.
2. Develop a base OpenFOAM simulation for laminar fluid flow.
3. Add the STL file to the base OpenFOAM simulation. Run blockMesh to generate the background mesh. Run snappyHexMesh to embed the porous media model and refine the mesh.
4. Use the Linux shell to create job-submission scripts that automate the simulations. Each script will:
   a. Allocate one or more nodes on SuperMIC.
   b. Edit the number of subdomains in the decomposeParDict file.
   c. Decompose the domain with decomposePar.
4. Run icoFoam in parallel using OpenMPI over the allocated nodes.
5. Submit the job scripts to the cluster. Visualize the results in paraFoam.

Results

- Simulation Wall Time, 8M Cells
- Total CPU Time, 8M Cells

Discussion

Experimental results show a maximum speedup of about 800 percent. The physical implications of decomposition are evidenced by irregularities in the simulation time and total CPU time trends.

References

- \textsuperscript{3} OpenFOAM Foundation. \textit{OpenFOAM User's Guide, version 4.0}.

Acknowledgements

Dr. Becky Carmichael, CEC Coordinator, LSU
Dr. Feng Chen, IT Consultant, HPC User Services, LSU
Aaron Harrington, Graduate Student, Department of Chemical Engineering, LSU

This material is based upon work supported by the National Science Foundation under award OCI-1533236 with additional support from the Center for Computation & Technology at Louisiana State University.