Abstract. Variations of the Navier–Stokes equation are standard models for the description of viscous liquid and gas flow. An application, used in many industrial fields such as automobile, aerospace, and hydrocarbon production industries. An application in the latter is the direct simulation of fluid transport through porous media at the pore scale, more precisely, on spatial domains that resolve the geometry of porous matrices of rocks. This poster presents a discontinuous Galerkin (DG) method for the compressible Navier–Stokes equation defined on voxel sets representing the pore space of rock samples at micrometer scale. The method exhibits optimal convergence and the simulated velocity fields compare well against the ones yielded by analytical solutions for simple geometries. The DG-based simulator also delivers intuitive velocity fields for complex pore geometries.

Mathematical model

- One-component single-phase momentum balance equation:
  \[ \delta_t (\rho v) + \nabla \cdot (\rho v v) - \nabla \cdot (\mu D(v)) = - \nabla p + f \text{ in } (0, T) \times \Omega. \]
- Initial and boundary conditions [1]:
  \[ v = v_i \text{ on } \Gamma \times \Omega, \]
  \[ v = v_0 \text{ on } (0, T) \times \partial \Omega^I, \]
  \[ \left( \gamma_2 D(v) + \lambda (v \cdot v) I - p I \right) v = 0 \text{ on } (0, T) \times \partial \Omega^O_{ \text{out}}. \]
- Deformation tensor: \( D(v) = (\nabla v + (\nabla v)^T)/2 \).
- Inflow boundary: \( \partial \Omega^I = \{ x \in \partial \Omega | v \cdot n < 0 \} \).
- Outflow boundary: \( \partial \Omega^O_{ \text{out}} = \partial \Omega \setminus (\partial \Omega^I \cup \partial \Omega^O_{ \text{out}}). \)

Voxel grid from CT imaging

- Micro-CT scan creates rock images at micrometer scale.
- Voxel sets represent the structure of porous medium.
- Voxel associated with the pore space build the computational domain (gray).
- Buffers are added to control boundary conditions (blue).

Porous domain flow simulation

Single-phase flow of an almost incompressible fluid occupying the pore space is modeled via direct numerical simulation approach. Shown is a steady-state velocity field, where high/low-flow regions are distinctly visible:

- One buffer: 10 × 80 × 80 voxels (64 000 elements).
- CT-scan image: 80 × 80 × 80 voxels (120 309 elements).
- Number of elements: \( N_{el} = 2 \times 64 000 + 120 309 = 248 309. \)
- Local degrees of freedom: \( N_\rho = 4. \)
- Unknowns per time step: \( 3 \times N_{el} \times N_\rho = 2 197 708. \)

Discretization

- Discontinuous Galerkin scheme for space discretization [2].
- Hierarchical basis achieving arbitrary order approximation.
  \[ v_h(x) = \sum_{\ell=1}^{N_\ell} \sum_{\alpha=1}^{N_\alpha} \sum_{\beta=1}^{N_\beta} v_{h, \ell, \alpha} \phi_{\ell, \alpha}(x), \]
- Implicit Euler method for temporal discretization.
- GPU-accelerated sparse iterative solvers from Paralution [3].

Numerical verification

- Zero errors for manufactured stationary solutions in polynomial spaces up to arithmetic accuracy.
- Convergence rates for prescribed non-polynomial solutions match theoretical rates.

<table>
<thead>
<tr>
<th>order</th>
<th>elements</th>
<th>steps</th>
<th>error SIPG</th>
<th>rate</th>
<th>error NIPG</th>
<th>rate</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3 × 3 × 3</td>
<td>1</td>
<td>1.76E–2</td>
<td>5</td>
<td>1.30E–1</td>
<td>2.12</td>
</tr>
<tr>
<td>2</td>
<td>2 × 2 × 2</td>
<td>4</td>
<td>1.91E–2</td>
<td>3.09</td>
<td>1.32E–1</td>
<td>1.97</td>
</tr>
<tr>
<td>4</td>
<td>3 × 3 × 3</td>
<td>12</td>
<td>3.36E–2</td>
<td>1.32</td>
<td>3.70E–1</td>
<td>2.49</td>
</tr>
<tr>
<td>8</td>
<td>8 × 8 × 8</td>
<td>64</td>
<td>6.14E–2</td>
<td>1.94</td>
<td>4.24E–1</td>
<td>2.05</td>
</tr>
<tr>
<td>12</td>
<td>12 × 12 × 12</td>
<td>1024</td>
<td>1.45E–2</td>
<td>1.74</td>
<td>7.14E–1</td>
<td>2.02</td>
</tr>
</tbody>
</table>

Future perspectives

- Coupling to mass transport equations.
- Extension to non-Newtonian fluids.
- Computation on larger voxel sets.
- MPI parallelization.