3D Benchmark "Incompressible Navier-Stokes' equations"

M. Ndjinga

July 21, 2022

Abstract

In view of the CANUM 2022 minisymposium "Saddle point problems in fluid dynamics", we propose a 3D benchmark to study the numerical simulation of an incompressible fluid modeled by time dependent Navier-Stokes' equations. The objective of the benchmark is to study the convergence of the Krylov solvers used in the numerical solution of the saddle point problem arising from the discrete Navier-Stokes' equations.

1 Introduction

We are interested in convection dominated flows in a pipe. This configuration is interesting for nuclear safety analysis of the coolant fluid in a reactor core. We would like to evaluate the behaviour of iterative solvers that are called at each time step of the simulation. The number of iterations can increase sharply as the mesh is refined. We are therefore looking for efficient preconditioning techniques.

The geometry of the domain in nuclear applications [4, 5] is sometimes tricky and can yield meshes with very poor quality as shown in the case of a nuclear assembly (section 5.2). We therefore provide mesh families that are representative of the difficulties encountered in nuclear thermal hydraulics applications. The main goal of this benchmark is to Asses various linear solvers for the saddle problem (16) arising from the discretisation of the Navier-Stokes system (7-8). The number of iterations tends to increase dramatically as the system size becomes large. Therefore efficient preconditioning techniques need to be designed. Another goal in the future will be to study the speed of convergence of the discretisations as the meshes are refined.

The benchmark consists in two different transient 3D problems where the fluid in a channel departs from the initial data to reach a stationary state. Each problem is based on a specific domain, and a specific sequence of refined meshes. In the first problem the domain is a duct with square cross section and allows a simple analytical stationary solution. In the second problem the cross section has an hexagonal shape with several holes. This geometry is typically encountered in nuclear reactors thermalhydraulics studies and does not allow an analytic expression for the stationary solution.

The PDE model The PDEs consist in either the Stokes' equations (section 2) or the incompressible Navier-Stokes' equations (section 3).

The domain The flow takes place in a tubular domain along the z axis : either a cube or a nuclear assembly (see section 5). The boundaries of the domain consist in an inlet, an outlet and walls. The inlet section is located at height $z = z_{min}$ and the outlet at height $z = z_{max}$.

The meshes In the case of the cube geometry, the meshes consist of uniform hexahedra gentle or bad tetrahedra, or more complex shapes (see section 5.1). In the case of the assembly geometry, the meshes consist of tetrahedra (see section 6.2).

The fluid The fluid is water at a constant temperature T_0. The fluid density ρ and friction viscosity μ are assumed constant. The numerical values for T_0, ρ and μ are given in equation (20).

Source term In the case of the cube geometry, an artificial source term (34) is used in the Stokes' equation (2) and in the Navier-Stokes' equation (8) in order to yield the exact solution (33).

2 The Stokes' equations benchmark

The equations We consider a 3D flow described by the Stokes' equations

$$\nabla \cdot \vec{u} = 0 \tag{1}$$

$$\vec{\nabla} P = \mu \triangle \vec{u} + \vec{f}, \tag{2}$$

with associated boundary conditions (constant fields)

$$P(z = z_{min}) = P_{inlet} \tag{3}$$

$$P(z = z_{max}) = P_{outlet} \tag{4}$$

$$\vec{u}(\vec{x} \text{ on wall }) = \vec{u}_{wall}. \tag{5}$$

The linear system Whatever the discretisation used for the system (1-2) the resulting linear system takes the saddle form

$$\begin{pmatrix} \mu\triangle & \vec{\nabla} \\ \nabla\cdot & 0 \end{pmatrix} \begin{pmatrix} \vec{u} \\ P \end{pmatrix} = \begin{pmatrix} 0 \\ \vec{f} \end{pmatrix}. \tag{6}$$

3 The incompressible Navier-Stokes' equations benchmark

The incompressible Navier-Stokes' equations We consider a 3D flow described by the Navier-Stokes' equations

$$\nabla \cdot \vec{u} = 0 \tag{7}$$

$$\partial_t \vec{u} + \nabla \cdot (\vec{u} \otimes \vec{u}) + \frac{1}{\rho}\vec{\nabla} P = \frac{\mu}{\rho}\triangle \vec{u} + \frac{1}{\rho}\vec{f}, \tag{8}$$

with associated initial conditions (constant fields)

$$\begin{aligned}\vec{u}(t=0,\vec{x}) &= \vec{u}_0 && (9)\\ P(t=0,\vec{x}) &= P_0, && (10)\end{aligned}$$

and boundary conditions (constant fields)

$$\begin{aligned}P(t,z=z_{min}) &= P_{inlet} && (11)\\ P(t,z=z_{max}) &= P_{outlet} && (12)\\ \vec{u}(t,\vec{x} \text{ on wall }) &= \vec{u}_{wall}. && (13)\end{aligned}$$

With $P_{inlet} > P_{outlet}$, the fluid will move from the inlet boundary towards the outlet boundary. In the absence of obstacles the flow will be directed along the z axis (case of the cube geometry). Otherwise the dynamics is much more complex (case of the nuclear assembly in section 5.2).

The time discretisation We propose the following backward Euler time discretisation of the Navier-Stokes system

$$\nabla\cdot\vec{u}^{n+1} = 0 \tag{14}$$

$$\frac{\vec{u}^{n+1}-\vec{u}^{n}}{\triangle t} + \nabla\cdot(\vec{u}^{n}\otimes\vec{u}^{n+1}) + \frac{1}{\rho}\vec{\nabla}P^{n+1} = \frac{\mu}{\rho}\triangle\vec{u}^{n+1} + \vec{f}, \tag{15}$$

The system (14-15) is an Oseen system that takes the saddle form

$$\begin{pmatrix} \frac{1}{\triangle t}\mathbb{I}_d + \vec{u}^n\cdot\nabla + \frac{\mu}{\rho}\triangle & \frac{1}{\rho}\vec{\nabla} \\ \nabla\cdot & 0 \end{pmatrix} \begin{pmatrix} \vec{u} \\ P \end{pmatrix}^{n+1} = \begin{pmatrix} 0 \\ \vec{f} \end{pmatrix}. \tag{16}$$

4 Numerical values

Physical parameters The fluid parameters correspond to water in normal conditions

$$\begin{aligned}T_0 &= 20°C && (17)\\ \rho &= 1000\,Kg/m^3 && (18)\\ \mu &= 10^{-3}\,Pa\,s && (19)\\ \vec{g} &= \begin{pmatrix} 0 \\ 0 \\ 10\,m/s^2 \end{pmatrix}. && (20)\end{aligned}$$

Initial data (ony for the Navier-Stokes' equations)

$$\begin{aligned}\vec{u}_0 &= (0,0,0) && (21)\\ P_0 &= 10^5\,Pa. && (22)\end{aligned}$$

Boundary data

$$
\begin{aligned}
P_{inlet} &= 10^5\, Pa &(23)\\
P_{outlet} &= P_{inlet} - \rho\|\vec{g}\|(z_{max} - z_{min}) &(24)\\
\vec{u}_{wall} &= (0,0,0). &(25)
\end{aligned}
$$

The fluid motion is generated by the pressure difference between the inlet and outlet sections. Since $P_{outlet} < P_{inlet}$ the fluir enters the domain at the inlet boundary and leaves the domain at the outlet boundary.

Numerical parameters

$$
\begin{aligned}
\triangle x_{min} &= \min_i diam(Cell_i) &(26)\\
\|P_h\| &= \frac{\max_i P_i}{P_0} &(27)\\
\|\vec{u}_h\| &= \max_i \|\vec{u}_i\| &(28)\\
\epsilon &= 10^{-6}. &(29)
\end{aligned}
$$

Time step (Naver-Stokes only)

$$
\Delta t = 10^3 \frac{\Delta x_{min}}{\max_i \|\vec{u}_i\|} \qquad (30)
$$

The stationary regime is reached when the time variation in every cell/node of the domain is negligible :

$$
\begin{aligned}
\max_i \frac{|P_i^{n+1} - P_i^n|}{P_0} &< \epsilon\Delta t, &(31)\\
\max_i \|\vec{u}_i^{n+1} - \vec{u}_i^n\| &< \epsilon\Delta t. &(32)
\end{aligned}
$$

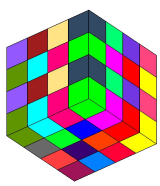
5 Domain and meshes

Two domains are considered : cube and nuclear assembly. Note that the mesh size is limited by file size limit on the github storage system (75Mb file $\approx$ 800000 cells).

5.1 Cube domain

The first problem is set in a simple cubic geometry that allows for an exact analytical solution.
We provide 4 mesh families of increasing difficulty :

- uniform hexahedra (compatible with finite difference or finite volume discretisations)



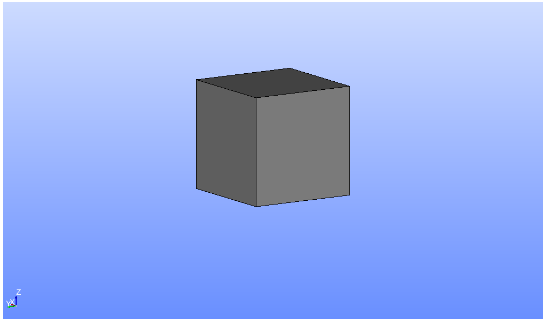
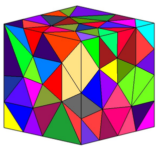


Figure 1: Cube domain

The meshes originate from the conference FVCA6 benchmark [2, 3] in Prague, 2011.

	mesh_hexa_1	mesh_hexa_2	mesh_hexa_3	mesh_hexa_4	mesh_hexa_5
Number of nodes	27	125	729	4913	35937
Number of cells	8	64	512	4096	32768

- gentle tetrahedra (compatible with finite elements or finite volume discretisations)

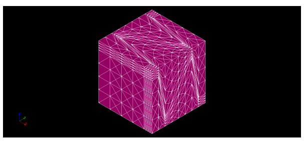


The meshes originate from the conference FVCA6 benchmark [2, 3] in Prague, 2011.

	mesh_tetra_0	mesh_tetra_1	mesh_tetra_2	mesh_tetra_3	mesh_tetra_4
Number of nodes	80	488	857	1601	2997
Number of cells	215	2003	3898	7711	15266

	mesh_tetra_5	mesh_tetra_6
Number of nodes	5692	10994
Number of cells	30480	61052

- bad tetrahedra (compatible with finite elements or finite volume discretisations)



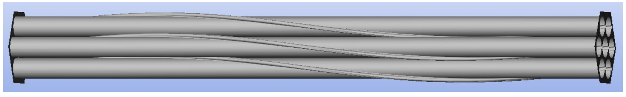


Figure 2: View of a wire wrapped nuclear assembly

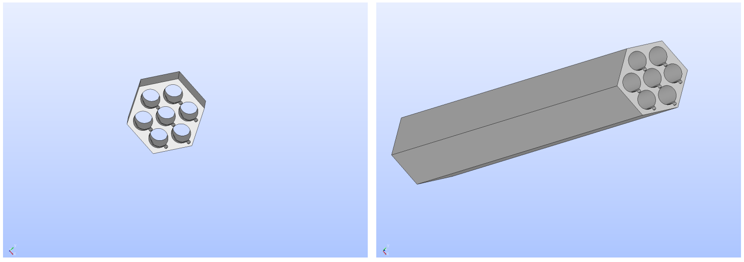
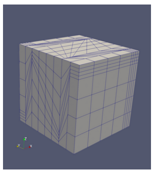


Figure 3: CAD modeling of the fluid domain around a 7-pin nuclear assembly

The meshes where obtained by tetrahedrisation of the Kershaw meshes from the conference FVCA6 benchmark [2, 3] in Prague, 2011.

	KershawTetra1	KershawTetra2	KershawTetra3
Number of nodes	3865	31793	258145
Number of cells	11072	93440	766976

- Kershaw polyhedra (compatible with finite volume discretisation)



The meshes originate from the conference FVCA6 benchmark [2, 3] in Prague, 2011.

	Kershaw1	Kershaw2	Kershaw3	Kershaw4
Number of nodes	729	4913	35937	274625
Number of cells	512	4096	32768	262144

5.2 Nuclear assembly

The second problem is set in a more complex geometry than the first and does not admit an exact analytical solution. The fluid flows around a 7-pin nuclear assembly wrapped with wires [4, 5]. The cross-section is hexagonal with 7 holes (see figure 5.2).

We need meshes of the fluid domain (figure 5.2) in order to perform CFD analysis. We provide a single mesh family made of tetrahedra (figure 4 and table 1).

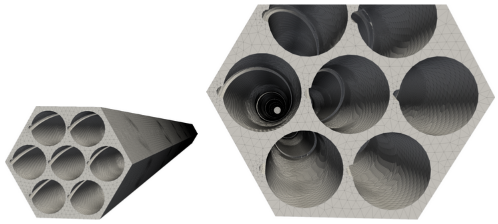


Figure 4: Nuclear assembly mesh

	AssemblyMesh1	AssemblyMesh2	AssemblyMesh3	AssemblyMesh4
Number of nodes	4999	15272	40976	63608
Number of cells	18122	49571	131979	218307

	AssemblyMesh5	AssemblyMesh6
Number of nodes	114915	209460
Number of cells	435021	808837

Table 1: Mesh names and sizes

6 Source term and stationary regime

6.1 Cube domain

In the case of the cube geometry we consider an exact solution in the form of a Poiseuille flow velocity.

$$\vec{u}_{stat,cube}(x,y,z) = \begin{pmatrix} 0 \\ 0 \\ x(x-1)y(y-1) \end{pmatrix}. \tag{33}$$

Injecting (33) into the Stokes' equation (1-2) or into the Navier-Stokes' equation (7-8) we find

$$\vec{f}_{cube} = -2\mu(x(x-1)+y(y-1)) + \rho\vec{g}. \tag{34}$$

6.2 Assembly domain

In the case of the nuclear assembly, there is no simple expression of the stationary solution and the source term is zero :

$$\vec{f}_{assembly} = \vec{0}. \tag{35}$$

References

[1] https://github.com/ndjinga/NavierStokes3DBenchmark

[2] https://github.com/ndjinga/FVCA_Meshes

[3] Fořt Jaroslav, Fürst Jiří, Halama Jan, Herbin Raphaèle, Hubert Florence, Finite volumes for complex applications VI - problems & perspectives, Volume 2, Springer Proceedings in Mathematics Series, Vol. 4, 2016

[4] U. Bieder, H. Uitslag-Doolaard, B.Mikuž, Investigation of pressure loss and velocity distribution in fuel assemblies with wire-wrapped rods by using RANS and LES with wall functions, Annals of Nuclear Energy, Volume 152, March 2021

[5] Min Seop Song, Jae Ho Jeong, Eung Soo Kim, Numerical investigation on vortex behavior in wire-wrapped fuel assembly for a sodium fast reactor, Nuclear Engineering and Technology, Volume 51, Issue 3, June 2019, Pages 665-675