FEM IMPLEMENTATION FOR NAVIER-STOKES EQUATIONS FOR AN INCOMPRESSIBLE STEADY FLOW USING HIERARCHICAL BASIS FUNCTIONS

***Andrei G. Shvarts[1], Łukasz Kaczmarczyk[1] and Chris J. Pearce[1]**

[1]Glasgow Computational Engineering Centre, School of Engineering, University of Glasgow

*andrei.shvarts@glasgow.ac.uk

Summary

We present an implementation of the finite-element method with hierarchical basis functions of arbitrary order for simulating incompressible steady-state fluid flow governed by Navier-Stokes equations (NSE). To verify the proposed formulation, we solved a problem of the fluid flow around a rigid sphere considering different velocity magnitudes. For a low-Reynolds-number flow the result coincides with the analytical solution for the Stokes flow, while for a higher Reynolds number a vortex is formed behind the sphere. The developed framework can be used to study the fluid flow in contact interfaces between solids with rough surfaces (e.g. in fractures) and compare the effect of the model describing the fluid flow (Reynolds, Stokes, or full NSE) on the transmissivity of these interfaces. The development is undertaken in MoFEM [1].

Key Words: *hierarchical basis functions; Navier-Stokes equations; flow past a sphere*

Introduction

Navier-Stokes equations (NSE), governing the motion of a viscous fluid, are used in various applications: from simulations of the flow in blood vessels to studies of the air flow around airplane wings and rotor blades, scaling up to models of ocean and atmospheric currents. Even in the case of an incompressible steady flow, NSE are non-linear due to the effect of the inertia, which is more pronounced in case of a higher Reynolds number. In this study we discuss the implementation of a viscous fluid model in the framework of the finite-element code MoFEM [1], which incorporates hierarchical basis functions of arbitrary order. This approach permits us to locally increase the order of approximation, enforcing conformity across finite element boundaries, without the need to change the implementation of an element. Moreover, the requirement of different approximation orders for primal (velocity) and dual (pressure) variables, necessary for a robust simulation of the flow, can be easily satisfied.

Problem statement

An incompressible isoviscous steady-state flow in a domain Ω is governed by the following equations:

$$\begin{cases} \rho\,(\boldsymbol{u}\cdot\nabla)\,\boldsymbol{u} - \mu\nabla^2\boldsymbol{u} + \nabla p = \boldsymbol{f}, & \text{(1a)} \\ \nabla\cdot\boldsymbol{u} = 0, & \text{(1b)} \end{cases}$$

where (1a) are the Navier-Stokes equations, representing the balance of the momentum, and (1b) is the continuity equation; $\boldsymbol{u} = [u_1, u_2, u_3]^\intercal$ is the velocity field, p is the hydrostatic pressure field, ρ is the fluid mass density, μ is fluid viscosity and $\boldsymbol{f}$ is the density of external forces. The boundary value problem complements equations (1) by the Dirichlet and Neumann conditions on the boundary $\partial\Omega$:

$$\begin{cases} \boldsymbol{u} = \boldsymbol{u}_D & \text{on } \Gamma_D, \quad \text{(2a)} \\ \boldsymbol{n}\cdot[-p\boldsymbol{I} + \mu\,(\nabla\boldsymbol{u} + \nabla\boldsymbol{u}^\intercal)] = \boldsymbol{g}_N & \text{on } \Gamma_N, \quad \text{(2b)} \end{cases}$$

where $\boldsymbol{u}_D$ is the prescribed velocity on the part of the boundary $\Gamma_D \subset \partial\Omega$, and $\boldsymbol{g}_N$ is the prescribed traction vector on the remaining part of the boundary $\Gamma_N = \partial\Omega \setminus \Gamma_D$, $\boldsymbol{n}$ is an outward normal.

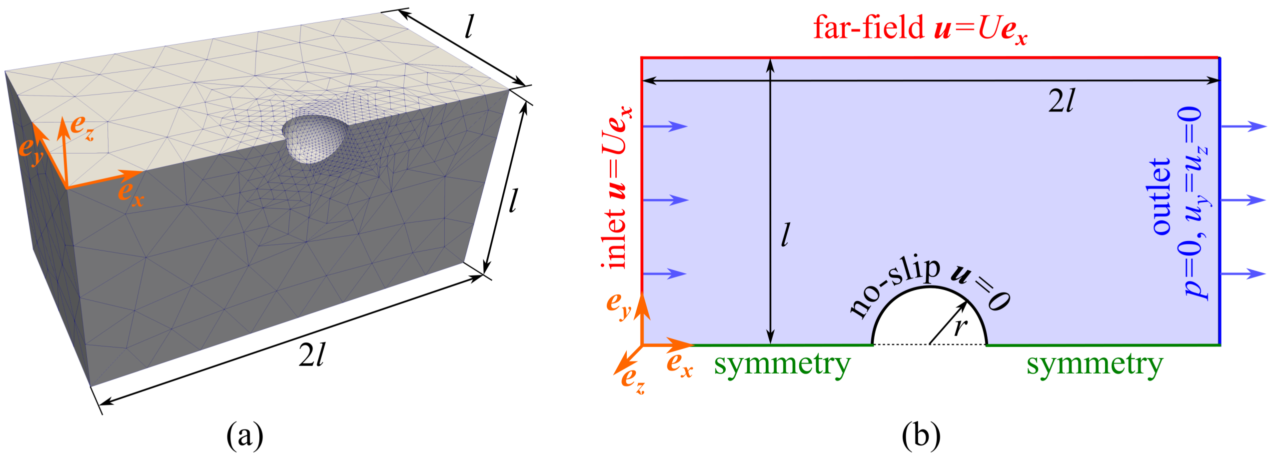

Figure 1: (a) Finite-element mesh used in simulations of fluid flow around a rigid sphere. (b) Sketch of the problem set-up on a section $z = 0$ of the mesh; parameters $r = 1$ m and $l = 7$ m were considered.

Finite-element implementation

The weak statement of the problem (1)-(2) reads: Find a vector field $\boldsymbol{u} \in \mathbf{V}$ and a scalar field $p \in \mathcal{P}$, such that for any test functions $\boldsymbol{v} = [v_1, v_2, v_3]^\intercal \in \mathbf{V}$ and $q \in \mathcal{P}$:

$$\int_\Omega \rho\,(\boldsymbol{u}\cdot\nabla)\,\boldsymbol{u}\cdot\boldsymbol{v}\,d\Omega + \int_\Omega \mu\,\nabla\boldsymbol{u} : \nabla\boldsymbol{v}\,d\Omega - \int_\Omega p\,\nabla\cdot\boldsymbol{v}\,d\Omega - \int_\Omega q\,\nabla\cdot\boldsymbol{u}\,d\Omega = \int_\Omega \boldsymbol{f}\cdot\boldsymbol{v}\,d\Omega + \int_{\Gamma_N} \boldsymbol{g}_N\cdot\boldsymbol{v}\,d\Gamma_N, \quad (3)$$

while the particular choices for the spaces $\mathbf{V}$ and $\mathcal{P}$ will be discussed below. Upon finite-element (FE) discretization of the domain Ω, we consider interpolation of both unknown fields introducing shape functions on each element:

$$u_i = \sum_{\alpha=1}^{n_{\boldsymbol{u}}} N_\alpha\, u_i^\alpha, \quad p = \sum_{\beta=1}^{n_p} \Phi_\beta\, p^\beta; \quad v_i = \sum_{\alpha=1}^{n_{\boldsymbol{u}}} N_\alpha\, v_i^\alpha, \quad q = \sum_{\beta=1}^{n_p} \Phi_\beta\, q^\beta, \quad (4)$$

where $n_{\boldsymbol{u}}$ is the number of shape functions associated with the velocity field, and n_p is the similar number for the pressure field. Using the hierarchical basis approximation, the vector of the shape functions can be decomposed into four sub-vectors, consisting of shape functions associated with element's entities: vertices, edges, faces and the volume of the element, e.g. for the velocity field:

$$\boldsymbol{N}^{el} = [N_1, \ldots N_\alpha, \ldots N_{n_{\boldsymbol{u}}}]^\intercal = \left[\boldsymbol{N}^{ver}, \boldsymbol{N}^{edge}, \boldsymbol{N}^{face}, \boldsymbol{N}^{vol}\right]^\intercal. \quad (5)$$

MoFEM [1] incorporates hierarchical basis functions of arbitrary order for tetrahedral meshes, based on Legendre [2], Lobatto [3] or Jacoby [4] polynomials, and approximations of $\mathbf{H}^1$, **H-div**, **H-curl** and $\mathbf{L}^2$ spaces can be considered. Furthermore, MoFEM provides functionality for iterating not only over all elements in the FE mesh, but also over shape functions associated with all entities of each element. Therefore, the resolution of a particular problem requires only the implementation of operators for computing the residual vector and the tangent matrix of each element, independent of the choice of the basis functions. Finally, the Newton method can be used for solving the non-linear problem.

Numerical results

In order to verify the proposed formulation, we solved the problem of the fluid flow around a rigid

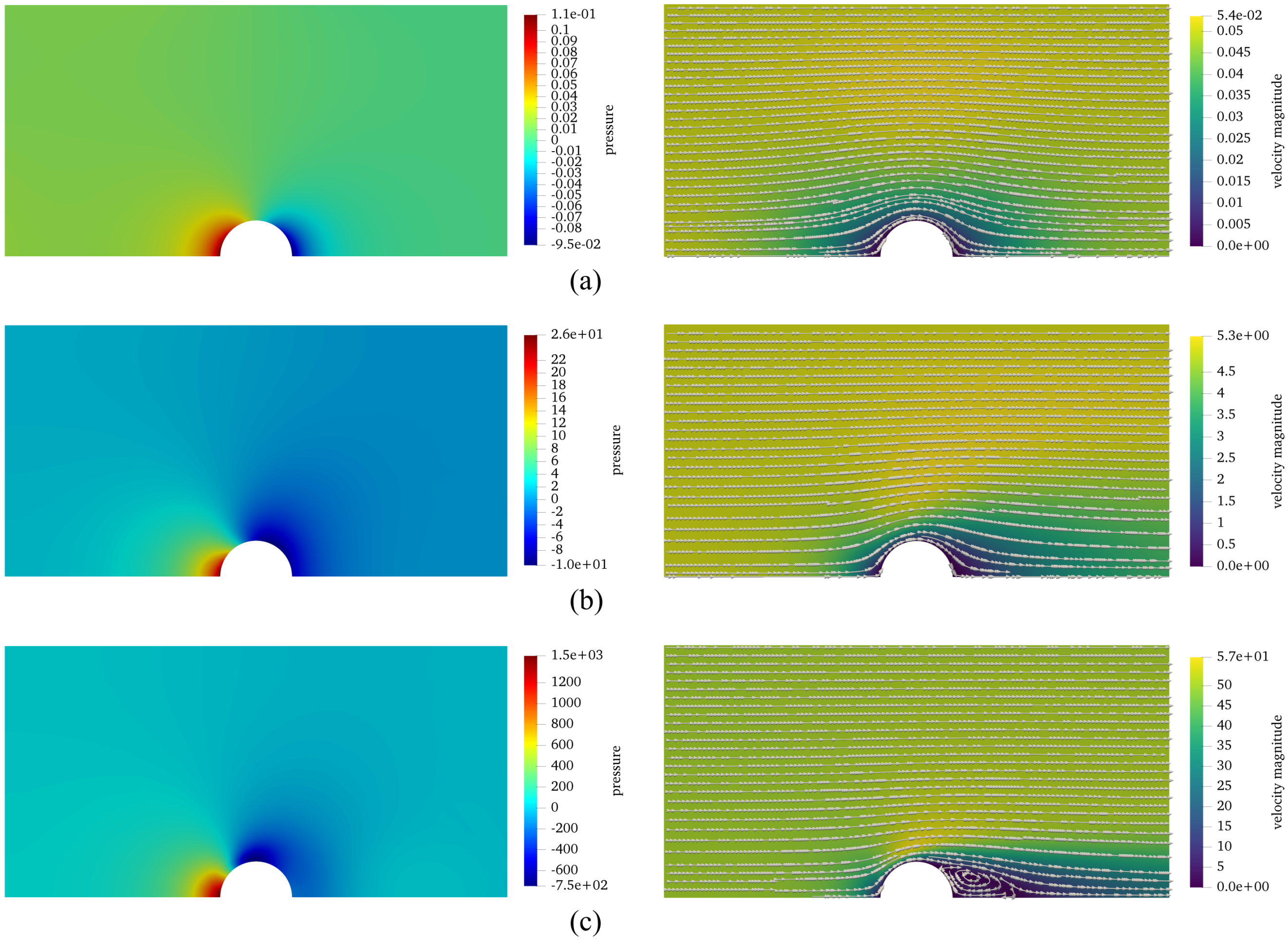

Figure 2: Results of simulations: the distribution of pressure (left) and velocity (right) on a section $z = 0$ of the mesh; (a) $U = 0.05\,\mathrm{m/s}, \mathcal{R} = 0.1$, (b) $U = 5\,\mathrm{m/s}, \mathcal{R} = 10$ and (c) $U = 50\,\mathrm{m/s}, \mathcal{R} = 100$.

sphere (see Fig. 1) of a radius r positioned in the centre of a cubic domain, the side length $2l$ of which is considered sufficiently large compared to r, so that a uniform far-field velocity on the exterior boundaries is valid (note that the body forces are neglected). Exploiting the symmetry of the problem, we used a quarter of the domain in our simulations. In Fig. 2 we present the numerical results obtained for three different values of far-field velocity magnitude: $U = 0.05, 5, 50\,\mathrm{m/s}$, which permits us to study the flow at different Reynolds numbers:

$$\mathcal{R} = \frac{2r\,U\,\rho}{\mu}, \tag{6}$$

where the diameter of the sphere $2r$ and the uniform far-field velocity magnitude U represent the length and the velocity scales, respectively. For a Reynolds number $\mathcal{R} = 0.1$ the solution is symmetric with respect to the plane $x = l$, in accordance with the analytical solution of the Stokes equation, which can be obtained neglecting the non-linear terms of NSE [5]. With $\mathcal{R} = 10$ the symmetry is broken, however, the flow remains laminar. For a higher Reynolds number $\mathcal{R} = 100$, we observe formation of a vortex behind the sphere. Note that in the presented example we considered Legendre polynomials [2] of the third order for velocity and of the second order for pressure, while the space $\mathbf{H}^1$ was used for both fields. Alternatively, the space $\mathbf{L}^2$ can be considered for the pressure field, however, in order to fulfil the necessary conditions of the patch test [6], a polynomial of even lower order must be used for approximating the pressure.

Perspectives

The proposed framework can be used to study the fluid flow in contact interfaces between solids with rough surfaces, which is relevant for numerous engineering and geophysical applications. The thin fluid flow is often described by the Reynolds equation, defined on the *lubrication surface*:

$$\nabla \cdot \left[\frac{h^3}{12\mu} \nabla p \right] = 0, \tag{7}$$

where h is the thickness of the fluid film, and the pressure p is assumed constant across the thickness, while the velocity profile is parabolic [5]. However, it has been pointed out that the Reynolds equation overestimates the fluid velocity (and, consequently, the fluid flux) in fractures, compared to the solution of full NSE, if the roughness of the surfaces is taken into account [7]. At the same time, the resolution of NSE becomes difficult if a fine discretization is used, necessary for a representative description of rough surfaces. The problem is further complicated if the contact between these surfaces is considered, resulting in a decrease of the film thickness under increasing external load. Nevertheless, the hierarchical basis approximation provides a possibility to consider the interfacial flow governed by NSE in a single layer of *prism elements*, using polynomials of an arbitrary order to interpolate the velocity profile across the thickness. Moreover, utilization of the hierarchical basis functions facilitates the use of more effective solvers (e.g. the multi-grid method), crucial for problems involving large number of unknowns, in particular, for simulations of fluid flow in rough contact interfaces.

Conclusions

In this study we showed how the model of a viscous fluid, governed by the Navier-Stokes equations, can be implemented in the finite-element framework using hierarchical basis approximation. To verify the proposed formulation, we solved a problem of the fluid flow around a rigid sphere. For a low-Reynolds-number flow, the solution coincides with the analytical solution for the Stokes flow, which neglects the non-linear terms, while under a higher Reynolds number a vortex is formed behind the sphere. Finally, we discussed a perspective application of the developed framework for studying the fluid flow described by Navier-Stokes equations in contact interfaces between solids with rough surfaces.

References

[1] L. Kaczmarczyk, Z. Ullah, K. Lewandowski, X. Meng, X.-Y. Zhou, C. Pearce, I. Athanasiadis, H. Nguyen, C.-A. Chalons-Mouriesse, and E. Miur. Mofem-v0.8.16, November 2018. http://mofem.eng.gla.ac.uk.

[2] M. Ainsworth and J. Coyle. Hierarchic finite element bases on unstructured tetrahedral meshes. *International Journal for Numerical Methods in Engineering*, 58(14), 2003.

[3] H. Bériot, A. Prinn, and G. Gabard. Efficient implementation of high-order finite elements for helmholtz problems. *International Journal for Numerical Methods in Engineering*, 106(3), 2016.

[4] F. Fuentes, B. Keith, L. Demkowicz, and S. Nagaraj. Orientation embedded high order shape functions for the exact sequence elements of all shapes. *Computers & Mathematics with Applications*, 70(4), 2015.

[5] G.K. Batchelor. *An introduction to fluid dynamics*. Cambridge University Press, 1967.

[6] O.C. Zienkiewicz and R.L. Taylor. *The finite element method: the basis*, volume 1. Butterworth-Heinemann, 2000.

[7] S.R. Brown, H.W. Stockman, and S.J. Reeves. Applicability of the Reynolds equation for modeling fluid flow between rough surfaces. *Geophysical Research Letters*, 22(18), 1995.