Discrete Projection Methods for Incompressible Fluid Flow Problems and Application to a Fluid-Structure Interaction Problem

Jörg-M. Sautter
Mathematisches Institut, Universität Düsseldorf, Germany, sautter@am.uni-duesseldorf.de

Abstract. Numerical simulations of transient incompressible fluid flow in arbitrary (two- or three-dimensional) domains, possibly coupled with other effects, are of great importance in many industrial applications. Discrete projection methods are efficient time integration schemes for incompressible fluid flow simulations. It is shown how the time-dependent Navier-Stokes equations can be implemented using Femlab’s general form. The discretization is performed in such a way that external algorithms, such as discrete projections methods (including an algebraic multigrid solver, etc.), can be applied to solve the discretized system of equations and that complex and highly nonlinear problems such as fluid-structure interaction, e.g. fluid flow coupled with rigid body motion, can be numerically simulated.

1 Introduction

Interaction of an incompressible fluid and rigid bodies belongs to the class of fluid-structure interaction problems. The coupling between the fluid and a structure can be twofold, i.e. one-way or two-way: The coupling is one-way if the motion of the structure influences the fluid flow but the action of the fluid flow upon the structure is negligible. The other way around is of course also possible. A rotating fan or a ship’s propeller are examples of fluid structure interaction problems with one-way coupling; at least in a first order approximation. The situation is more complicated when the motion of the structure influences the fluid flow and at the same time the structure’s motion is influenced by the fluid flow. This is called a two-way coupling. Examples are wind power plants and a boulder sinking in water.

In the following, the simulation of the motion of a rigid obstacle in a reservoir filled with an incompressible fluid will be considered. This is a fluid-structure interaction problem with two-way coupling in the sense described above. In section 2, the underlying Navier-Stokes initial boundary value problem will be introduced and its spatial discretization with Femlab is described in section 3. The basics of the discrete projection method for the time integration and the solution of the (non-) linear subproblems are described in section 4. Finally, a numerical example of a fluid-structure interaction problem is presented in section 5.

2 Incompressible fluid flow

Transient flow of isothermal incompressible Newtonian or non-Newtonian fluids is governed by the following Navier-Stokes initial boundary value problem in velocity-pressure form:

\[
\begin{align*}
& \rho \left( \frac{\partial u}{\partial t} + (u \cdot \nabla) u \right) = \nabla \cdot \left( -\rho I_{d \times d} + \eta (\nabla u + (\nabla u)^T) \right) + b & \text{in } \Omega \text{ for } t \in [0, t_{\text{final}}] \\
& \nabla \cdot u = 0 & \text{in } \Omega \text{ for } t \in [0, t_{\text{final}}]
\end{align*}
\]
where \( u \) is the fluid velocity, \( p \) the pressure, \( \eta \) the viscosity, \( \varrho \) the density, and \( b \) a given body force. \( \Omega \) denotes the \( d \)-dimensional area occupied by the moving fluid and \( \Gamma = \Gamma^D \cup \Gamma^N = \Gamma_{\text{in}} \cup \Gamma_{\text{out}} \cup \Gamma_{\text{solid}} \cup \Gamma_{\text{fb}} \) its boundary consisting of parts where fluid flows into and out of the domain and parts where the fluid suffices a no-slip condition or has a free boundary. \( \Gamma^D \) and \( \Gamma^N \) denote the parts where \( u \) is constrained by Dirichlet and Neumann boundary conditions, respectively. \( f_{\text{st}} \) represents the surface tension in case of a free boundary and \( f_{\text{out}} \) is the force acting upon the fluid at the outlet. Details can be found in [6].

Solutions of (1) do not only depend on initial data, boundary values, and the viscosity, but also on the Reynolds number. For viscous fluids governed by the Navier-Stokes equations, the solution may show singularities when the domain contains vertices, the boundary values are not continuous, or the type of boundary condition changes between Dirichlet and Neumann. Adaptivity in space and time to control these phenomena is not covered in this work.

Discretizing (1) in space with the Galerkin finite element method yields the discrete Navier-Stokes equations, which in full generality read

\[
M \frac{d u_h}{d t} = -A(u_h, u_h) + D(u_h) - G p_h + f \quad (2a)
\]
\[
C^T u_h = g. \quad (2b)
\]

\( u_h \) and \( p_h \) denote discrete approximations to the velocity and pressure fields. For Newtonian fluids, the discrete diffusion term \( D(u_h) \) is linear, i.e. \( D(u_h) := Du_h \). The spatial discretization process and the efficient incorporation of boundary conditions is described in detail in [6].

### 3 Spatial discretization

The incompressible Navier-Stokes equations are part of a fairly general general class of partial differential equations, namely general advection-diffusion equations, which can be solved using Femlab. However, since we would like to couple external algorithm to Femlab, we only use Femlab to discretize the following time-dependent partial differential equation in space.

Let \( \Omega \subset \mathbb{R}^d \) be a connected set and

\[
y = y(t, x_1, \ldots, x_d) = (y_1(t, x_1, \ldots, x_d), \ldots, y_N(t, x_1, \ldots, x_d))^T
\]

be the vector of dependent variables, i.e. an unknown function \( y : [0, T_{\text{final}}] \times \Omega \to \mathbb{R}^N \) that is to be determined from the system of partial differential equations

\[
\sum_{k=1}^{N} d_{ak} \frac{\partial y_k}{\partial t} + \nabla \cdot \Gamma_I = F_I \quad \text{in } \Omega \quad (3a)
\]
\[
- n^T \Gamma_I = G_I + \sum_{m=1}^{N_R} \frac{\partial R_m}{\partial y_I} u_m \quad \text{on } \partial \Omega \quad (3b)
\]
\[
0 = R_m \quad \text{on } \partial \Omega \quad (3c)
\]
where \( d_{ak} \in \mathbb{R} \) are the mass coefficients, \( \Gamma_i(t, x_1, \ldots, x_d, y_1, \ldots, y_N) \in \mathbb{R}^d \) the flux vector, and 
\( F_i(t, x_1, \ldots, x_d, y_1, \ldots, y_N) \in \mathbb{R} \) the source term. \( G_i \in \mathbb{R} \) is a boundary source term, \( \mu_m \in \mathbb{R} \) a Lagrange multiplier, and \( R_m \in \mathbb{R} \) restricts the solution component \( y_i \) on the boundary. The equation index \( l \) ranges from 1, \ldots, \( N \), while the constraint index \( m \) ranges from 1, \ldots, \( N_R \).

Femlab’s general form is based upon the partial differential equation (3) which can be used to discretize initial boundary value problem (1) in space. There are several ways to do this and a comparison of different implementations can be found in [6]. Let

\[
y = y(t, x) = \begin{pmatrix} u(t, x) \\ p(t, x) \end{pmatrix} \in \mathbb{R}^{d+1}
\]

and \( x \in \Omega \subset \mathbb{R}^d \) and \( N = d + 1 \). The so-called total stress form excels all other forms for our purposes since true physical forces are modeled best. It is given by

\[
d_a = \begin{pmatrix} \rho I_d & 0_d \\ 0_d & 0 \end{pmatrix}, \quad \Gamma = \begin{pmatrix} -\eta(\nabla u + (\nabla u)^T) + pl_d & 0_d \\ 0_d & 0 \end{pmatrix}, \quad F = \begin{pmatrix} b - \rho(u \cdot \nabla)u \\ \nabla \cdot u \end{pmatrix}.
\]

Advantages of the total stress form are that physical outflow boundary conditions as well as boundary conditions on a free boundary or a rigid obstacle are natural boundary conditions of the underlying Galerkin finite element method.

Having defined the geometry and having generated a mesh in Femlab, the total stress form is simply defined by

\[
fem.equ.ga = \Gamma, \quad fem.equ.da = d_a, \quad fem.equ.f = F.
\]

Femlab’s \texttt{assemble} function can be used to perform the Galerkin finite element spatial discretization based on Taylor-Hood elements which yields, after elimination of constraints,

\[
D_e(y_e(t), t) \frac{dy_e(t)}{dt} = L_e(y_e(t), t)
\]

where \( D_e, L_e, \) and \( y_e \) denote the eliminated mass matrix, eliminated load vector and eliminated solution vector, respectively. Therefrom and from the Jacobian of the eliminated load vector, all components of the discrete Navier-Stokes equations can be extracted, i.e.

\[
M \frac{du_h}{dt} = B(u_h, u_h) - Cp_h \quad (6a)
\]

\[
C^T u_h = g \quad (6b)
\]

where \( B(u_h, u_h) = -A(u_h, u_h) + D(u_h) + f \) denotes a discrete Burgers operator containing advection and diffusion term as well as body forces and contributions from Dirichlet boundary conditions. Note that in contrast to (2), the discrete gradient operator and the discrete divergence operator are adjoint due to the total stress form.

4 Time integration

In computational fluid dynamics, discrete projection methods have been introduced as a useful and efficient way to significantly reduce the computational cost of time-dependent incompressible viscous flow simulations in the velocity-pressure formulation. This means that the system (6) is split into a series of simpler equations, such as discrete advection-diffusion equations and discrete Poisson equations, for example, and explicit or implicit updates. A fully implicit discrete projection method is presented which may be regarded as a discrete counterpart (cf. [9])
and fully implicit version of Chorin’s classical projection method (cf. [2, 3, 4]). More details on
discrete projection methods can be found in [6].

In order to reduce the computational cost, consider the following splitting scheme and solve

\[
\frac{1}{\Delta t} \left( M\tilde{u}_h^{n+1} + Mu_h^n \right) = -A(\tilde{u}_h^{n+1}, \tilde{u}_h^{n+1}) + D(\tilde{u}_h^{n+1}) + f^{n+1}
\]

for \( \tilde{u}_h^{n+1} \) and

\[
C^T M^{-1} C\lambda_h = C^T \tilde{u}_h - g^{n+1}
\]

for \( \lambda_h \) and then perform two updates

\[
u_h^{n+1} = \tilde{u}_h - M^{-1} C\lambda_h \quad \text{and} \quad p_h^{n+1} = \frac{\beta_0}{\Delta t}\lambda_h.
\]

Using this approach, the nonlinear iteration in order to solve (7) is only in the velocity unknowns
due to the pressure decoupling, i.e. only “smaller” linear systems compared to a fully coupled
approach need to be solved. Therefore, the nonlinear iteration is significantly cheaper com-
pared to a fully coupled approach. The linear systems which need to be solved within the
nonlinear iteration always have full rank, no matter whether there are pressure modes or not.
In addition, only one pressure Poisson equation must be solved per time step in the projection
step (8), where possible pressure modes have to be taken into account.

This time integration scheme requires (within a simplified Newton method and a preconditioned
conjugate gradient method) the solution of linear systems

\[Mu = b\]

and

\[(M + c_1 \Delta t J_A - c_2 \Delta t J_D)u = b, \quad c_1, c_2 \in \mathbb{R},\]

in steps (7), (8), and (9) where \( J_A \) and \( J_D \) denote the Jacobians of the advection and diffusion
term, respectively. These linear subproblems are solved with an algebraic multigrid (AMG)
solver (cf. [1, 7]) based on the Ruge-Stüben algorithm (cf. [5]). In contrast to geometric multigrid
methods, AMG requires no a priori given hierarchy of coarse grids. In fact, the construction of
a problem-dependent hierarchy – including the coarsening process itself, the transfer operators
as well as the coarse-grid operators – is part of the AMG algorithm, based solely on algebraic
information contained in the given system of equations (cf. [8]).

5 A fluid-structure interaction simulation

As an object moves through a fluid, the viscosity of the fluid acts on the moving object with
a force that resists the motion of the object. At the same time, the fluid flow depends on the
motion of the obstacle. The fluid velocity on each structure boundary must be constrained to
match the rigid-body motion of the structure. The approach presented here is based on body
fitted unstructured moving grids for optimal spatial resolution and the implicit discrete projection
method described in the previous section.

In order to simulate the motion of \( N_{fsi} \in \mathbb{N} \) moving obstacles within a fluid, the Navier-Stokes
equations are coupled with the equations for rigid body motions. The initial boundary value
problem for a fluid-structure interaction problem is therefore given by the complete initial boundary value problem (1) augmented by the following equations for \( i = 1, \ldots, N_{\text{fsi}} \) moving structures \( \text{fsi}(i) \),

\[
\begin{align*}
    m_{\text{fsi}(i)} \frac{d\mathbf{u}_{\text{fsi}(i)}}{dt} &= m_{\text{fsi}(i)} \mathbf{g} + F_{\text{fsi}(i)}(\mathbf{u}) \\
    \frac{d\mathbf{x}_{\text{fsi}(i)}}{dt} &= \mathbf{u}_{\text{fsi}(i)} \\
    I_{\text{fsi}(i)} \frac{d\omega_{\text{fsi}(i)}}{dt} &= T_{\text{fsi}(i)}(\mathbf{u}) \\
    \frac{d\theta_{\text{fsi}(i)}}{dt} &= \omega_{\text{fsi}(i)}(0) = (\mathbf{0}, \mathbf{0}), \quad \omega_{\text{fsi}(i)}(0) = (\mathbf{0}, \mathbf{0}).
\end{align*}
\]  

(12a) \hspace{1cm} (12b) \hspace{1cm} (12c) \hspace{1cm} (12d)

\[
F_{\text{fsi}(i)}(\mathbf{u}) = \int_{\Gamma_{\text{struct}}} (-p I_d x d + \eta (\nabla \mathbf{u} + (\nabla \mathbf{u})^T) n)n
\]

(12e) \hspace{1cm} (12f)

\[
T_{\text{fsi}(i)}(\mathbf{u}) = \int_{\Gamma_{\text{struct}}} r_{\text{fsi}(i)} T (-p I_d x d + \eta (\nabla \mathbf{u} + (\nabla \mathbf{u})^T) n)n
\]

(12g)

\[
\mathbf{u} = \mathbf{u}_{\text{fsi}(i)} + \omega_{\text{fsi}(i)} \times r_{\text{fsi}(i)}
\]

(12h)

Mass, velocity of the center of mass, position of the center of mass, moment of inertia, angular velocity, and angle of rotation of the \( i \)th structure are denoted by \( m_{\text{fsi}(i)}, \mathbf{u}_{\text{fsi}(i)}, \mathbf{x}_{\text{fsi}(i)}, I_{\text{fsi}(i)}, \omega_{\text{fsi}(i)}, \) and \( \theta_{\text{fsi}(i)} \), respectively, and gravity (or any other type of body force) is denoted by \( \mathbf{g} \). Equations (12e) and (12f) represent the force and torque exerted upon the \( i \)th structure by the flow. Most important for the simulation is the fluid-structure coupling expressed by the transient no-slip boundary conditions (12g). The initial boundary value problem (12) describes fluid-structure interaction problems as long as there are no collisions between the moving structures.

As an example of fluid-structure interaction, consider the situation shown in figure 1. A reservoir is filled with fluid and more fluid is pumped into the reservoir through the inlet on the left side while fluid leaves the reservoir through the outlet on the right side. Within the reservoir, there is a solid structure (a circle) which moves under body forces (gravity) and the forces of the fluid acting upon the structure. The domain of the reservoir is represented by 18 boundary segments. Boundary segment \( \Gamma_1 \) represents the inlet where the velocity profile of the flow is given as

\[
\mathbf{u}_\text{in} = \mathbf{u}_\text{in}(s) = (s(1 - s), 0)^T
\]

where \( s \in [0, 1] \) linearly parameterizes \( \Gamma_1 \). Boundary segment \( \Gamma_{12} \) is the outlet where homogeneous traction boundary conditions are applied, i.e. the force acting upon the fluid at the outlet is \( F_{\text{out}} = 0 \in \mathbb{R}^2 \) on \( \Gamma_{12} \). Let the density of the fluid be \( \rho = 1 \), the viscosity \( \eta = 0.1 \), and the body force \( \mathbf{b} = (0, -2)^T \). The parameters of the structure are as follows: The circle’s radius is \( r = 0.15 \) and it has a homogeneous mass density. Its mass is \( m_{\text{fsi}} = 1 \) and its moment of ineratia \( I_{\text{fsi}} = 1 \).

At \( t = 0 \), the fluid is at rest and the initial position of the center of mass and velocity of the structure are \( \mathbf{x}_{\text{fsi}}(0) = (0.3, 1.8)^T \) and \( \mathbf{u}_{\text{fsi}}(0) = (0, 0)^T \), respectively. In addition, the initial angular velocity of the structure is \( \omega(0) = 2 \) and its initial angular orientation is \( \theta(0) = 0 \).

As the structure moves downward and comes closer to the inlet, it gets swept away by the stream going mainly from inlet to outlet. Figure 2 shows a sequence of plots of the velocity vector field.
Figure 1: Initial configuration

References


Figure 2: Velocity field