EXPLICIT RESISTIVE PRESSURE OUTLET BOUNDARY CONDITION

HELLMUTH, R.1, MURPHY, E.2, HOUSTON, G.3
1Vascular Flow Technologies, rudolf.hellmuth@vascular-flow.com
2University of Dundee, e.t.murphy@dundee.ac.uk
3University of Dundee, j.g.houston@dundee.ac.uk

Keywords: boundary condition, lumped model, haemodynamics, ODE

Many industrial and biological systems consist of a network of pipes and chambers of different sizes. Because of its configuration, a complete CFD analysis of such systems is normally impractical, and only regions of interest are simulated. Simulations of regions of interest can be performed with accuracy only if the boundary conditions (BCs) are experimentally measured. However, this is not always possible, specially if the simulation is trying to predict the hydrodynamic performance of a device in its design stage. To that end, the hydrodynamics inside the region of interest (i.e., in the CFD domain) is coupled with the hydrodynamics outside the region of interest, which would alter the BCs. Velocity and pressure waves travel into the region of interest, where they are transformed by the CFD simulation, and they continue being damped, dispersed and reflected downstream from the CFD domain. As a consequence, solutions to the governing equations in the CFD domain are highly dependent on the outflow BCs imposed to represent the rest of the system downstream of the region of interest. The importance of modelling the dynamics of the downstream region has been demonstrated in publications of haemodynamics [1], but it is equally important in industrial cases.

Methods

The approach presented here for connecting the dynamics of an entire system of internal flow with the CFD domain consists of time-varying flow rate inlets, and pressure outlets that are softly coupled with the outlet velocity field. The soft coupling consists in obtaining the outlet pressure value by applying the outlet flow rate, which is the surface integral of the outlet velocity, to a lumped model representing the downstream hydraulic network. The simplest outlet lumped model is a flow resistance outlet, which is analogous to Ohm’s Law of electricity. Therefore, the outlet pressure is calculated as

$$p = p_0 + RQ,$$

where $p$ is the patch uniform pressure, $p_0$ is a reference pressure, $R$ is the downstream resistance, and $Q$ is the volumetric flow rate. A linear relation as in Eq. 1 is valid for laminar flow, where the resistance is calculated from the Hagen–Poiseuille equation:

$$R = \frac{128 \mu L}{\pi D^4},$$

where $\mu$ is the dynamic viscosity of the fluid, and $L$ and $D$ are the length and diameter of the pipe, respectively. The resistance is not linear in neither compressive nor turbulent flows, but non-linear equations similar to Eq. 1 can also be derived.

The compliance of the downstream flow, either due to some elasticity of the pipe system, or to the compressibility of the system can also be introduced. A simple case is the two-element windkessel model, illustrated in Figure 2. The windkessel effect is a term used in medicine to account for the shape of the arterial blood pressure waveform in terms of the interaction between the stroke volume and the compliance of large elastic arteries [2]. The lumped model for this model is analogous to an RC electronic circuit, whose pressure has a time-varying dynamics given by

$$\frac{p - p_0}{R} + C \frac{d(p - p_0)}{dt} = Q,$$

where the compliance is defined as $C := \frac{dV}{dP}$, for $V$ being the volume of fluid stored in the lumped model. The ordinary differential equation (ODE) represented in Eq. 3 can be integrated by finite difference method. With implicit backward difference the pressure becomes

$$p_i = p_{0,i} + \frac{RQ_i \Delta t + RC (p_{i-1} - p_{0,i-1})}{\Delta t + RC},$$

where $\Delta t$ is the time step size, and the subscript $i$ is the time step index. The transient time constant of this RC system is given by

$$\tau = RC.$$

The lumped models for the outlets shown in Eqs. 1 and 4 were implemented explicitly as Dirichlet pressure BCs. The patch internal field is used to compute the flow rate, instead of the flux phi on the patch, in order to take advantage of relaxation factors of the SIMPLE algorithm. For positive flux, the velocity BCs were set as zero-gradient Neumann, and for negative flux, they were set as Dirichlet, with the values obtained from the patch-face normal component of the internal-cell value. A blood vessel bifurcation was used to test the dynamics of the BCs.

Results

The geometry of a blood vessel bifurcation was chosen to test the new outlet BCs. The exits of the vessel bifurcation were named as inlet, outlet0, and outlet1. The solution of a case where the resistance of outlet1 is 100-fold larger than of outlet0 ($R_1 = 100 R_0$) can be seen in Figure 2. In this case, almost 99% of the flow exits the CFD domain through outlet0.

In order to test the stability of the system, a flow rate program of fast step shifts was set at the inlet. The result of that for a 10-fold larger resistance at outlet1 (i.e., $R_1 = 10 R_0$) can be seen in Figure 3. The damped flow rate and pressure transitions occur due to transient effects inside the fluid domain. The transient inertial effects are also clearly observed by the pressure overshoot at the inlet. Negative fluxes were also possible, which is shown on the graphs after 10 ms.

An example of the dynamical response of the same bifurcation geometry with the two-element windkessel model at the outlets is shown in Figure 4. In that case, the compliance of both outlets are the same, but the resistance at outlet1 is 10-fold larger (i.e., $R_1 = 10 R_0$). This result in the time constant of outlet1 also being 10-fold larger that of outlet1 (Eq. 5). In the shown case, the time constants were $\tau_0 = 2$ ms and $\tau_1 = 20$ ms. Hance, the flow rate response at each outlet vary in time in order to accomodate the time-varying resistance of all the other outlets, as well as the transient effects inside the CFD domain. As demonstrated in Figure 4, shortly after a fast inlet flow change at $t = 10$ ms, the flow rate is higher at outlet1 than at outlet0, becuse resistance of former is lower than of the later. The flow rate becomes higher at outlet0 than at outlet1, once the resistance of outlet1 becomes higher than the resistance of outlet0.
Figure 3: Dynamical response of flow rate (left) and pressure (right) at the exits of a vessel bifurcation with resistive outlets. A flow rate program of fast step shifts was set at the inlet, and flow resistance of outlet1 was 10-fold larger than of outlet0.

outlet0.

Figure 4: Dynamical response of flow rate (left) and pressure (right) at the exits of a vessel bifurcation with windkessel outlets. A flow rate program of fast step shifts was set at the inlet, and RC time constant of outlet1 was 10-fold larger than of outlet0.

Discussion

The possibility of applying explicit lumped models at outlet boundary conditions has been demonstrated in OpenFOAM. However, this formulation is not inherently stable, due to its explicit nature. The stability of the CFD simulation depends that the time step size can capture the changes of the imposed flow rate. A too quick flow rate change can cause an abrupt change of pressure at the outlets, that may cause oscillations of both pressure and flux in all outlets. Since the pressure change is dependent of the resistance of the outlet, there is a magnitude limitation for the resistance parameter to be realizable. Fortunatelly, with not too large resistance levels, convergence can be attained by the use of relaxation factors in the SIMPLE algorithm. This limitation is less severe when the lumped model has a soft transient response, such as the two-element windkessel model.

Acknowledgments

The authors thank Dr. Eugene de Villiers from Engys (http://engys.com/) for helping with the code development. We gratefully acknowledge the support of Vascular Flow Technologies (http://vascular-flow.com/), specially of Craig Dunlop and Mike O’Sullivan. This project has received funding from the European Unions Seventh Framework Programme for research, technological development and demonstration under grant agreement no 324487.

References
