Early Stage Researcher		Post-Doctoral Researcher/Established Researcher/PI	X
Please consider this for a poster presentation only		Please consider this for an oral presentation	X

Please place an X in any appropriate categories

DEVELOPMENTS TOWARD A RED-BLOOD-CELL-INCORPORATING COMPUTATIONAL FLUID DYNAMICS SOLVER FOR CARDIOVASCULAR-SYSTEM FLUID FLOW ANALYSIS

Gallagher, G, Boyle, F
Technological University Dublin
email: gerald.gallagher@dit.ie

INTRODUCTION

Computational fluid dynamics (CFD) is routinely used for numerically predicting cardiovascular-system fluid flows. Current CFD packages ignore the suspended cellular phases of blood which negatively affects simulation accuracy. The work undertaken is to allow the development of a physiologically realistic CFD solver capable of accurately modelling blood flow, focusing on the behaviour of plasma and red blood cells (RBCs) during flow.

A lattice Boltzmann fluid flow solver capable of modelling plasma is currently in development and will allow for the incorporation of a state-of-the-art RBC spring particle structural model using the immersed boundary method (Chen et al.).

The aim of this paper is to show the accuracy of the lattice Boltzmann method when modelling curved surfaces, before eventually coupling with the immersed boundary method and simulating an equivalent problem for comparative purposes. The test case simulated is 2D laminar flow around a stationary cylinder.

MATERIALS AND METHODS

The physical problem set up is described by Schäfer et al. The lattice Boltzmann D2Q9 single-relaxation-time (SRT) method was implemented in C++ and allows for execution on graphics processing units (GPUs) and utilises the NVIDIA CUDA framework. The current GPU in use is an NVIDIA GTX Titan Black, which has 2880 CUDA cores available. Half-way bounce-back boundary conditions are used to simulate no-slip conditions in the current simulations, which creates a "staircase" profile at the cylinder surface.

RESULTS

The results obtained from the simulations were compared with a compilation of simulation results from various research groups (Schäfer et al.). These results are shown in Figure 1 and Table 1; C_D is the drag coefficient, C_L is the lift coefficient, L_a is the recirculation zone length and ΔP_c is the pressure difference between the front and back of the cylinder. Figure 1 is based on the simulation with the most refined mesh which gave the most accurate results.

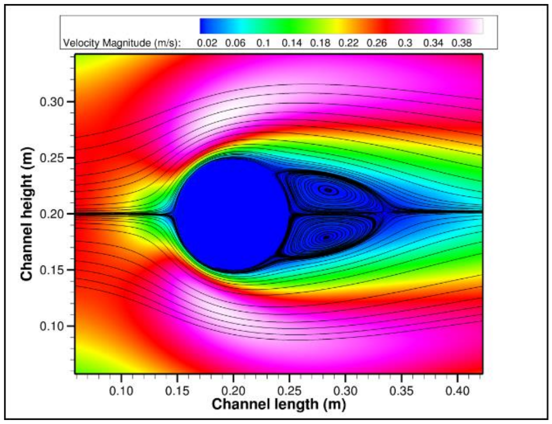

Figure 1 Close up of velocity magnitude in m/s and streamlines around cylinder for 1760 x 328 mesh.

Mesh (cells in x and y)	C_D (-)	C_L (-)	L_a (m)	ΔP_c (Pa)	Sim. time (s)
Varies[a]	5.7-5.9	0.0104-0.0110	0.0842-0.0852	0.1172-0.1176	Varies
440 x 82[b]	6.23	0.020	0.095	0.137	118.8
880 x 164[b]	5.74	0.014	0.088	0.116	400.4
1760 x 328[b]	5.63	0.012	0.085	0.116	1836.4

Table 1 Results from [a]Schäfer et al. (upper and lower bounds are taken for group results) and [b]current lattice Boltzmann simulations.

DISCUSSION

Results showing the excellent accuracy of the SRT lattice Boltzmann method for modelling curved boundaries are shown. A 2D lattice Boltzmann-immersed boundary code is currently in development with preliminary results already available for 2D laminar flow around a stationary cylinder. This will be immediately followed by 3D simulations with spherical bodies (both stationary and moving in the fluid), paving the way for coupling with the RBC structural model.

REFERENCES

Chen *(et al.)*, Materials Science and Engineering: C, 43:506-516, 2014.
Schäfer *(et al.)*, Flow Simulation with High-Performance Computers II: DFG Priority Research Programme Results 1993—1995, Vieweg+Teubner Verlag, 1996.