WHIOCE PUBLISHING PTE. LTD.
International Journal of Bioprinting

RESEARCH ARTICLE

Wall shear stress during impingement at the building platform can exceed nozzle wall shear stress in microvalve-based bioprinting

Supplementary File

1. Supplementary figure

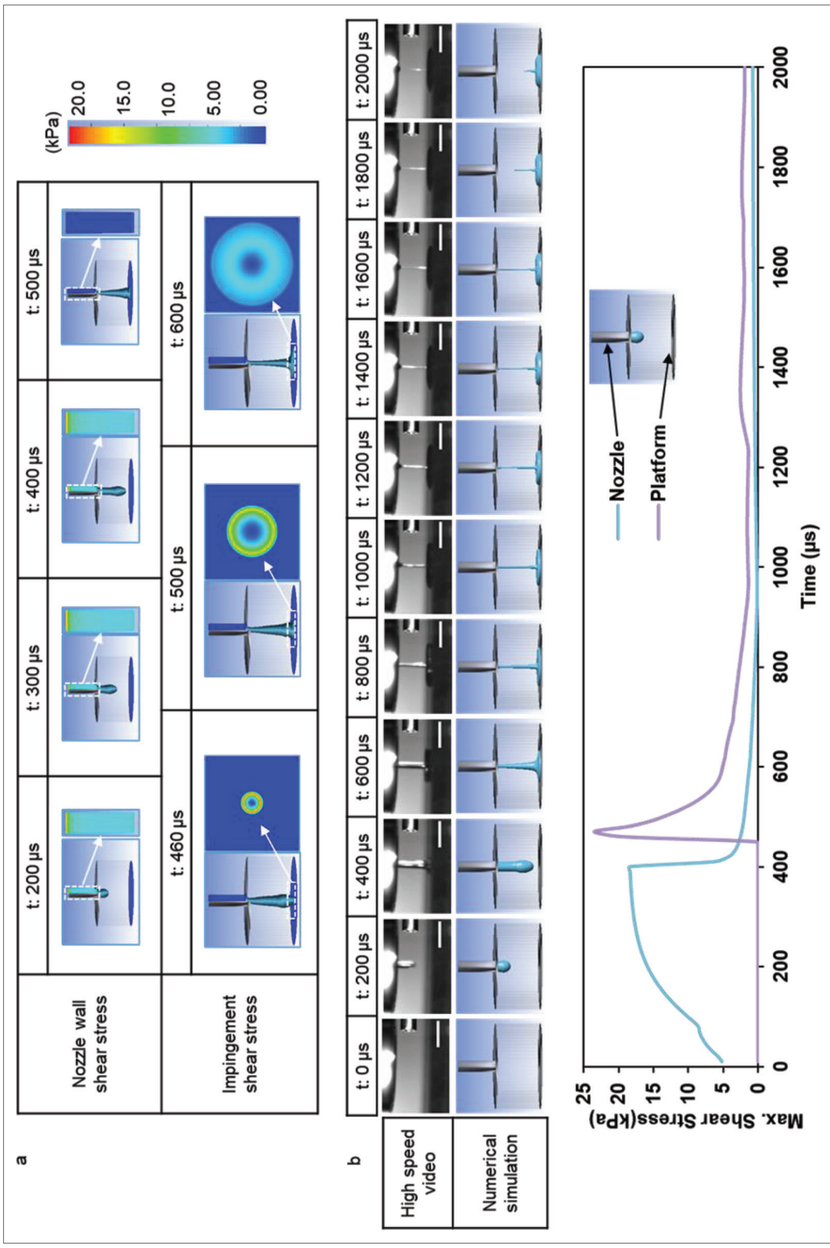

Figure S1. The nozzle wall shear stress and impingement shear stress for nozzle size of 300 µm and upstream pressure of 1.0 bar. (a) The contour plot of nozzle wall shear stress and impingement shear stress at selected time points. (b) Dispensing dynamics captured by high-speed camera together with maximum shear stress as a function of time.

2. Supporting information

2.1. Droplet impingement model

When solenoid microvalves are used for bioprinting, the bioink is under hydrostatic pressure inside the microvalve chamber and a solenoid mechanism is used to displace a movable piston to open or close the area at the inlet of the nozzle. When a droplet is dispensed from the microvalve, it travels through the nozzle-to-platform distance and impinges on the platform. A numerical model was used here to predict the mechanical stimuli during the impingement process. For this model, Ansys Fluent was utilized. In the following sections, three basic steps in simulating droplet impingement are described. First, the geometry of the simulation model and mesh generation are described. Then, the mathematical modeling and boundary conditions applied for numerical simulation of droplet impingement are presented. In the last section, the numerical setup used for solving the governing equations is introduced.

2.2. Geometry and mesh generation

For the simulation of the droplet impingement, an axisymmetric geometry was considered (**Figure S2a**). This two-dimensional geometry consists of a nozzle and the area between the nozzle and platform. The length (1.0 mm) and diameter (*D*) of the nozzle of solenoid microvalves (SMLD 300G, Fritz Gyger AG, Gwatt, Switzerland; nozzle diameters 150 µm or 300 µm) were used for this geometry. The origin of the Cartesian coordinate system was located on the symmetry axis at the outlet of the nozzle.

Two types of SMLD solenoid microvalve were used to determine the nozzle diameter: D = 150µm and D = 300µm. Different nozzle-to-platform distances (H) within the range of 0.3 mm to 3 mm were used. Note that in this geometry, the direction of gravitational acceleration is in the opposite direction of the x-coordinate.

Ansys Meshing was used for automatic mesh generation within the geometry of the droplet impingement model. The "Quadrilateral Dominant" method was used to generate mostly quadratic linear elements inside the axisymmetric domain. Smaller elements close to the nozzle walls and platform were generated to capture the high gradient of variables at the proximity of solid walls (boundary layer mesh). This fine mesh was generated using the "Inflation" tool in the Ansys Meshing software. **Figure S2b** depicts a sample mesh inside the simulation domain. For the grid study, three different meshes with different element sizes were used. The details of the parameter setting used for generation those meshes are reported in **Table S1**.

2.3. Mathematical modeling and boundary conditions

In Ansys Fluent, different methods are available for modeling multiphase fluid flow problems, e.g., Volume of Fluid, mixture, Wet Steam, and Inhomogeneous Eulerian. Each of those methods is suitable for special problem cases and physics. For the simulation model of droplet impingement, the Volume of Fluid model was used. This method is suitable for modeling the multiphase flow of several immiscible fluids. In this method, a single set of momentum equations is solved throughout the fluids domain, and conservation of mass equation is utilized to track the volume fraction of each fluids. A two-dimensional axisymmetric problem (**Equation S1a**), the volume fraction equation (**Equation S1b**), and conservation of momentum (**Equation S1c**) are:

$$\frac{\partial}{\partial t}\left(f_\alpha \rho_\alpha\right)+\frac{\partial}{\partial x}\left(f_\alpha \rho_\alpha u_x\right)+\frac{\partial}{\partial r}\left(f_\alpha \rho_\alpha u_r\right)+\frac{f_\alpha \rho_\alpha u_r}{r}=0 \tag{S1a}$$

$$\begin{aligned}\frac{\partial}{\partial t}\left(f_\alpha \rho_\alpha u_x\right)+\frac{1}{r}\frac{\partial}{\partial x}\left(r f_\alpha \rho_\alpha u_x u_x\right)+\frac{1}{r}\frac{\partial}{\partial r}\left(r f_\alpha \rho_\alpha u_r u_x\right)\\ =-f_\alpha\frac{\partial p}{\partial x}+\frac{1}{r}\frac{\partial}{\partial x}\left[r f_\alpha \mu_\alpha\left(2\frac{\partial u_x}{\partial x}-\frac{2}{3}\left(\nabla.\vec{u}\right)\right)\right]\\ +\frac{1}{r}\frac{\partial}{\partial r}\left[r f_\alpha \mu_\alpha\left(\frac{\partial u_x}{\partial r}+\frac{\partial u_r}{\partial x}\right)\right]+M_\alpha\end{aligned} \tag{S1b}$$

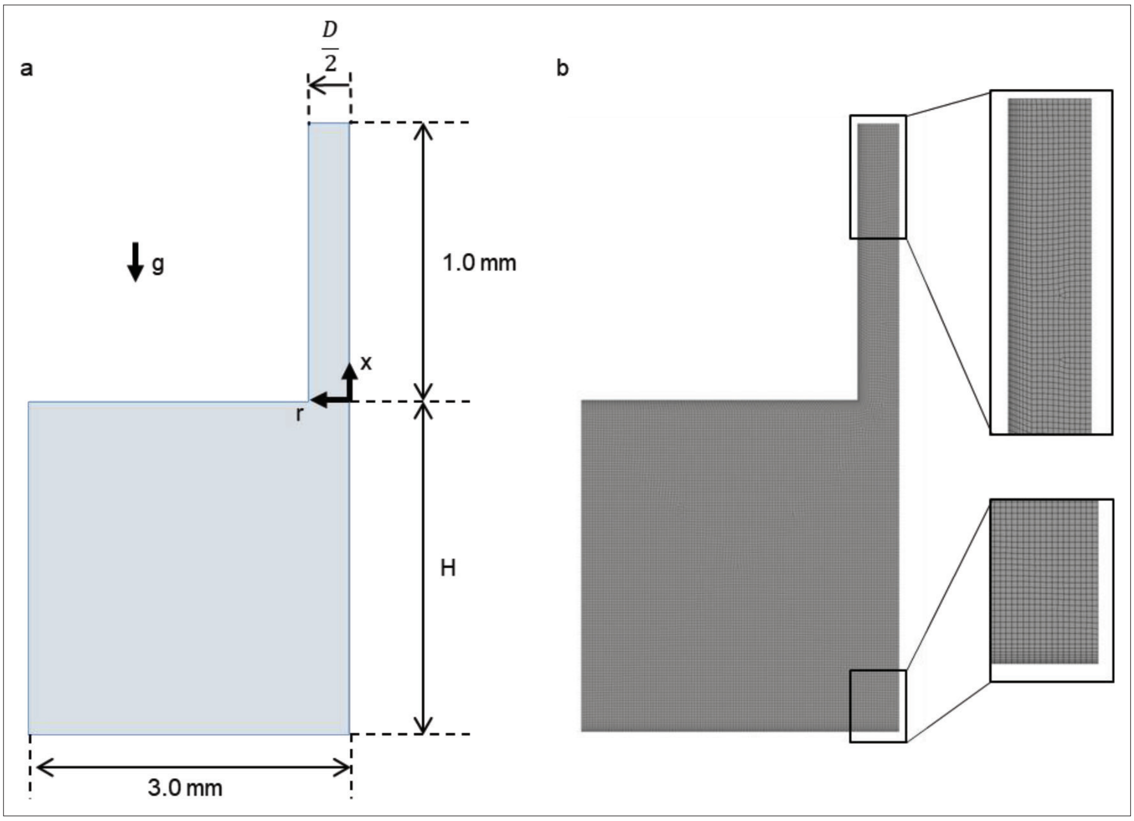

Figure S2. The geometry (a) and sample mesh (b) of simulation domain for droplet impingement model. *D* and H are nozzle diameter and nozzle-to-platform distance. The value of these two parameters varies based on the simulation case.

Table S1. Parameter set for mesh generation of different element sizes for grid study of droplet impingement model

	General element size	Inflation method			Total number of elements
		No. of layers	Growth rate	Thickness	
Course	12 μm	9	1.2	78 μm	11,271
Medium	10 μm	9	1.2	42 μm	16,711
Fine	8 μm	9	1.2	35 μm	25,616

$$\begin{aligned}\frac{\partial}{\partial t}\left(f_\alpha \rho_\alpha u_r\right)+\frac{1}{r}\frac{\partial}{\partial x}\left(r f_\alpha \rho_\alpha u_x u_r\right)+\frac{1}{r}\frac{\partial}{\partial r}\left(r f_\alpha \rho_\alpha u_r u_r\right)\\
=-f_\alpha \frac{\partial p}{\partial r}+\frac{1}{r}\frac{\partial}{\partial x}\left[r f_\alpha \mu_\alpha\left(\frac{\partial u_r}{\partial x}+\frac{\partial u_x}{\partial r}\right)\right]\\
+\frac{1}{r}\frac{\partial}{\partial r}\left[r f_\alpha \mu_\alpha\left(2\frac{\partial u_r}{\partial r}-\frac{2}{3}\left(\nabla.\vec{u}\right)\right)\right]-2 f_\alpha \mu_\alpha \frac{u_r}{r^2}+\frac{2}{3}\frac{f_\alpha \mu_\alpha}{r}\left(\nabla.\vec{u}\right)+M_\alpha\end{aligned} \tag{S1c}$$

where is $\nabla.\vec{u}$

$$\nabla.\vec{u}=\frac{\partial u_x}{\partial x}+\frac{\partial u_r}{\partial r}+\frac{u_r}{r}$$

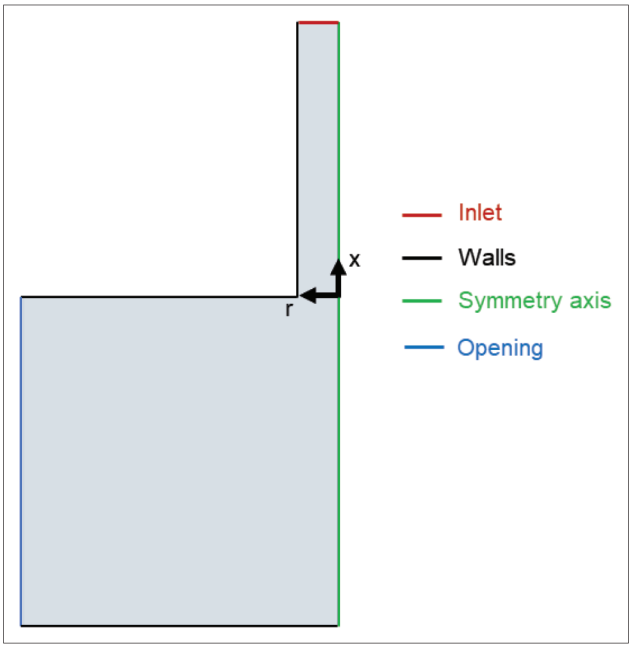

Figure S3. Boundary conditions for the simulation model of droplet impingement.

Table S2. The details of boundary conditions applied to the boundaries of simulation domain of droplet impingement model

Boundary	Boundary condition
Symmetry axis	Axis
Solid wall interfaces	No slip stationary walls with zero velocity and defined contact angle
Open surfaces	Pressure outlet, zero relative pressure, total pressure backflow
Inlet	Pressure inlet, total relative pressure, normal to boundary

Boundary and initial conditions of variables are required for solving the above equations. **Figure S3** depicts the axisymmetric domain and boundary conditions for modeling the droplet impingement during microvalve-based bioprinting. The details of each boundary condition are reported in **Table S2**. The volume fraction of liquid phase (bioink) at the inlet was set to one. The relative pressure at the inlet (upstream pressure) was variable depending on the simulation case and is reported for each case in the Results section. Moreover, when the microvalve is closed, the inlet boundary condition was changed from "Pressure inlet" to no-slip stationary wall.

2.4. Simulation model

A pressure-based transient simulation was set for the droplet impingement model. The gravitational acceleration was set to $g = 9.81$ m·s^{-2} in the opposite direction of the x-coordinate. Multiphase (Volume of Fluid) laminar flow model was activated. Implicit formulation with volume fraction cut-off of 1E-06 was considered. For interface modeling, the "Sharp" option was activated. Two fluids were introduced inside the domain: alginate 1.5% w/v as the primary phase and air as the secondary phase. Continuum surface force with constant surface tension was activated for phase interaction.

For transient multiphase flow, the initial state of the velocity field, static pressure, and volume fraction of fluids at time zero must be defined. All components of the velocity in the coordinate system were set to zero at time zero, shown mathematically as

$$u_x\left(t_0\right)=0 \ \ and \ \ u_r\left(t_0\right)=0$$

Table S3. General setting of numerical scheme used by Ansys Fluent for simulation of droplet impingement

Parameter	Value
Transient scheme	First-Order Implicit
Velocity pressure coupling	Pressure-Implicit with Splitting of Operators (PISO)
Volume fraction	Compressive
Gradient	Least Squares Cell-Based
Pressure	Body Force Weighted
Momentum	Second-Order Upwind
Convergence criteria	1.0E-03, 20 iteration

The relative static pressure within the simulation domain at time was set to zero. The following formula was patched to the simulation domain to define the initial air volume fraction at starting time *t*.

$$f_{air}\left(x,r,t_0\right)=1-step\left[50\times10^{-6}\left(1-\left(\frac{2r}{D}\right)^2\right)+x\right]$$

Considering the above equation as the air initial volume fraction, the nozzle is full of bioink at the beginning of simulation. Details of the numerical scheme used for this model are reported in **Table S3**.

For the sake of convergence, a six-step, user-specified variable time step was used as shown below:

a) 1000 steps with $\Delta t_1 = 1.0 \times 10^{-10}$ *s* for $0 \le t < 1.0 \times 10^{-7}$

b) 1000 steps with $\Delta t_2 = 5.0 \times 10^{-10}$ *s* for $1.0 \times 10^{-7} \le t < 6.0 \times 10^{-7}$

c) 1000 steps with $\Delta t_3 = 1.0 \times 10^{-9}$ *s* for $6.0 \times 10^{-7} \le t < 1.6 \times 10^{-6}$

d) 15,936 steps with $\Delta t = 2.5 \times 10^{-8}$ *s* for $1.6 \times 10^{-6} \le t < 400 \times 10^{-6}$

e) Changing the inlet boundary condition from pressure-inlet to wall at time $t = 400$ μs

f) 64,000 steps with $\Delta t = 2.5 \times 10^{-8}$ *s* for $400 \times 10^{-6} \le t < 2000 \times 10^{-6}$

3. Other files

Videoclips S1. Droplet formation captured by high-speed camera (HSC).
Videoclips S2. Simulation of droplet formation.