Supplementary information

Remodeling of an in vitro microvessel exposed to cyclic mechanical stretch

Finite element simulation

COMSOL Multiphysics®5.4 was used to perform finite element simulation of the 3D vasculature under mechanical stretch. 3D structure of the multilayer composed of PDMS membrane and fibrin gel layer incorporating the vasculature was drawn using geometry toolbar of the software. Multilayer was adopted as linear elastic material within solid mechanics interface to obtain deformation field throughout the tube once exposed to negative pressure from the bottom of the membrane. Fig. S1 shows the simulation results for longitudinal and circumferential displacement fields of the vasculature under stretch. As results of simulation, the longitudinal and circumferential strains under CS^{high} are 8% and 63%, respectively which are in a good agreement with the experimental results. Simulation of the vascular deformation under mechanical stretch helps to better understand the nature of the longitudinal and circumferential displacement field within the vasculature.

A

B

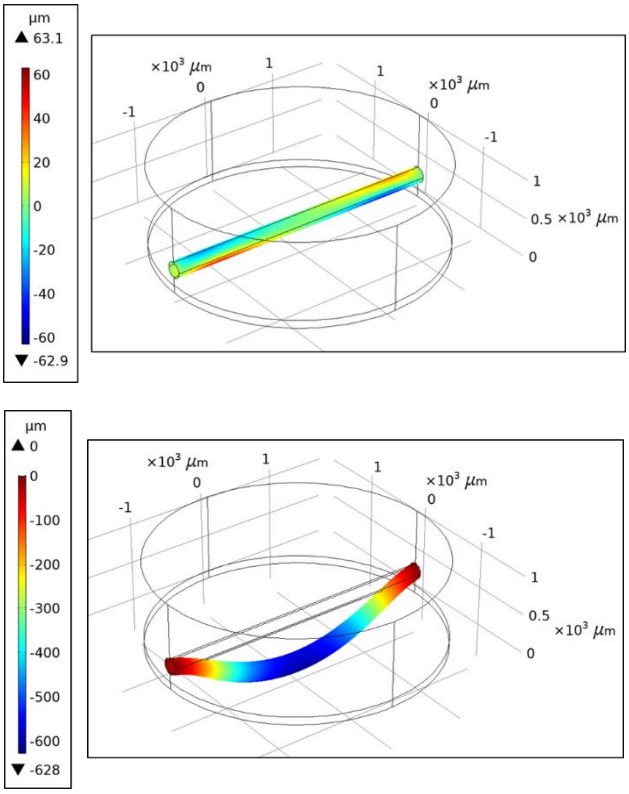

Fig. S1: Fibrin gel layer incorporating a vasculature under deflection. A) Circumferential displacement field of the vasculature under CS^{high} . B) Longitudinal displacement field of the vasculature under CS^{high}.