Supporting Information

FEM Simulation

In FEM Simulation a two-dimensional model with linear elastic response is established and explicit procedure is used by commercial software ABAQUS. The ring and channel are put on the same plane and the ring is initially in contact with the outer boundary. The kinematic contact is used in the model and friction has negligible effect. The ring can move freely in the channel and uniform growth strain ε along the axial direction of ring is applied. Thermal expansion along axial direction is used to simulate the growth strain ε. In this model we assume that the channel is rigid and fixed and friction between ring and channel is ignored. The ring is discretized by CPE4R elements with reduced integration. The mesh convergence study has been conducted to guarantee mesh density. In simulation for higher modes small rigid smooth objects are used to help to apply the lateral displacement. These objects are small enough compared to the size of ring so that they have no influence on deformation induced by growth.

Qualitative Experiment

In qualitative experiment the two ends of a PVC (polyvinyl chloride) sheet was adhered to make a closed elastic ring and the annular channel (plastic) was 3D-printed by Ultimaker 2+. The channel model for 3D-printing is prepared by commercial software Solidworks.