A SPICE-based Multi-physics Seamless Simulation Platform for CMOS-MEMS

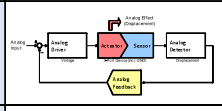
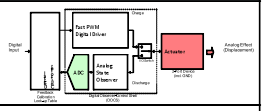
Satoshi Maruyama

Department of Electrical and Electronic Engineering Toshiyoshi-lab

Multi-physics simulation for MEMS (microelectromechanical systems) is becoming an indispensable tool to comprehend the behavior of the system under test, where electrical and mechanical coupled response should be analyzed simultaneously . In this late news, we report a SPICE version of such multi-physics solver that is capable of microelectromechanical transient analysis, AC harmonic analysis, and electro-mechanical mixed-signal simulation that can be performed seamlessly with the LSI simulation. Also, we are developing a new type of digital control chip that could drive a MEMS actuator as well as pick-up the displacement at the same time.

Introduction

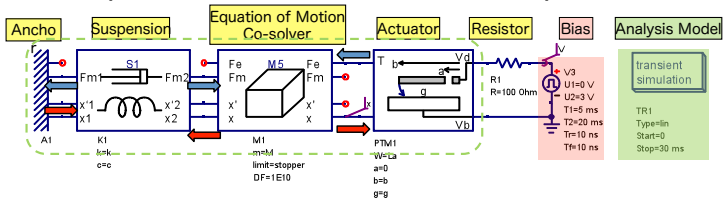
This work has been extended to explore a new scheme of MEMS driver circuit based on the frequency division drive and control; an electrostatic MEMS micro actuator is regarded as a capacitor in the switched –capacitor circuit, and time sequentially connected to either the driver or capacitance-detection circuit at a speed over the mechanical response such that the actuator's displacement is monitored in a very short time without causing degradation. This new scheme works as a digital shell between the analog-controlled MEMS and digital controlled.

	Conventional	This Work
		
Feedback	Sensor (Built-in or Extra)	Observer Control (Displacement speculated from charge induction)
Application	For most MEMS Actuators	Electrostatic Micro Actuator Only (at this moment)
Control	Analog	Digital
Benefit		Digital IP Library can be used for control circuit diagram.

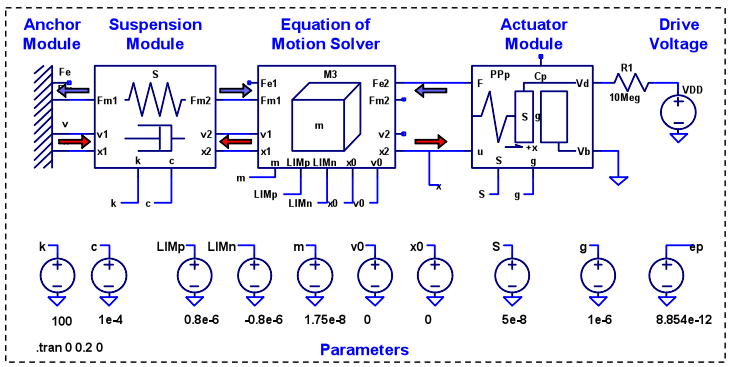
Experimental setup

Force on Current, Displacement & Velocity on Voltage

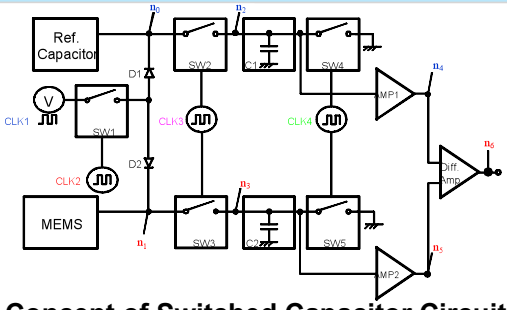
Ouc s (Quite Universal Circuit Simulator)



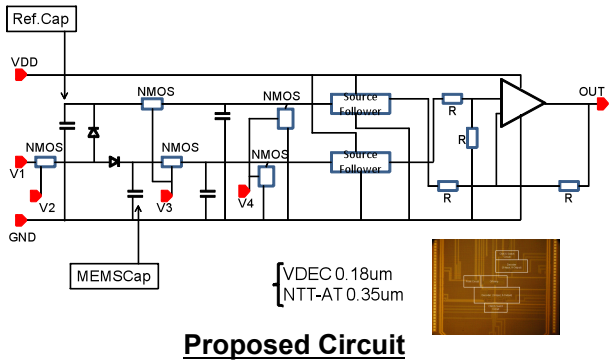
LT-Spice



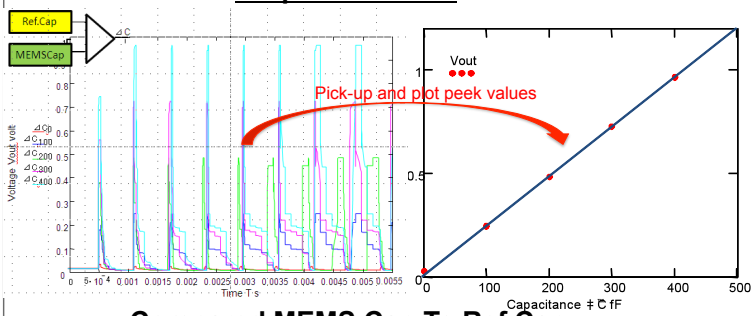
Result



Concept of Switched Capacitor Circuit



Proposed Circuit



Compared MEMS Cap To Ref.Cap

Conclusion

We have developed a SPICE-based multi-physics simulator for MEMS that could handle both electromechanical and electrical simulation on a single platform. Equivalent circuit model for micromechanical device was directly synthesized from its analytical mathematic model by using an equation-defined nonlinear current source. Unlike most MEMS simulators, our methodology has higher degrees of freedom in adapting to various MEMS devices, as compared in Table. Seamless extension to mask design layout is under development. And, we have developed a new type of digital control chip for All-Digital Control Integrated MEMS.

Contact: satomaru@iis.u-tokyo.ac.jp