# A Tutorial on Using Hspice and Spice Viewer as a Standalone Tool

**Dept. of Electrical and Computer Engineering**
**Concordia University**
**Montreal, Quebec**
**Canada**

**By**
**Ted Obuchowicz**
**VLSI/CAD Specialist**
**Concordia University**

**December 10, 1997**
**Revised: October 7, 2009**

Revision History

January 4, 2001: updated paths to electronic documentation.

Sept. 13, 2005: updated paths to electronic documentation for HSPICE 2005.03 version
found in /CMC/tools/meta/docs

October 7, 2009: updated portion on viewing simulation results using Synopsys Spice Viewer
as Awaves is no longer supported by Synopsys.