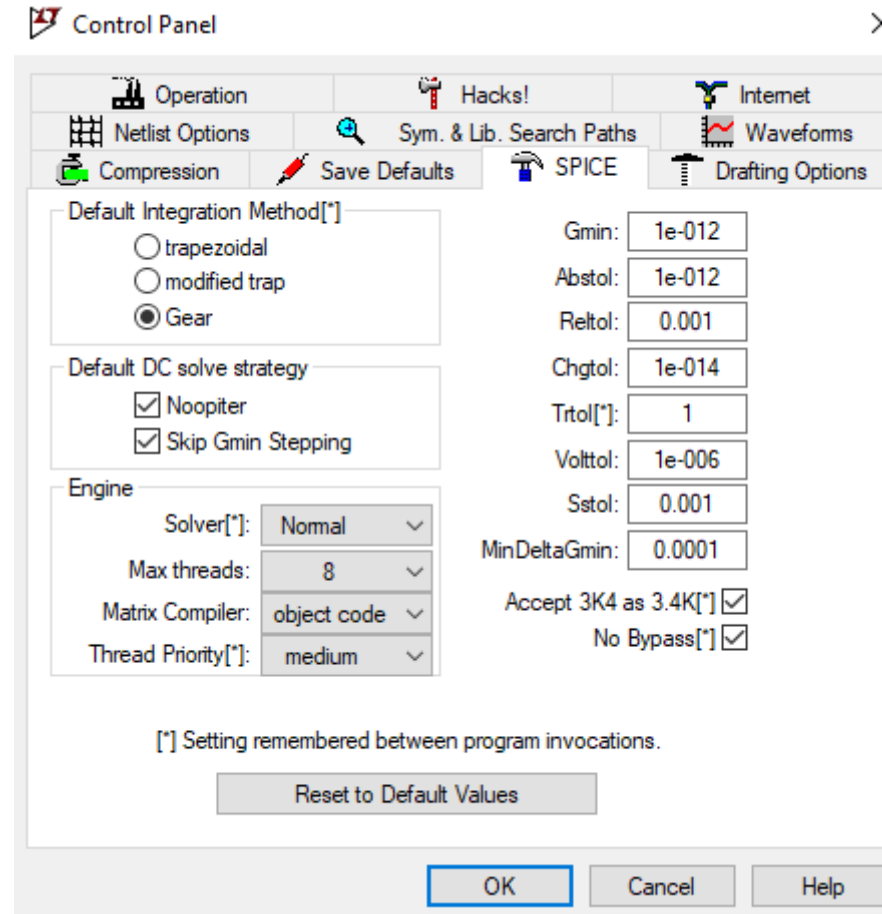


Tips for Achieving Simulation Convergence With EPC Space Modular Products

Tony Marini
Power Technologist
EPC Space
August 16, 2022

- Prior to running an LT Spice simulation of a circuit containing an EPC Space module model, it is suggested to do the following:
- Set the “Default Integration Method[*]” to “Gear” by choosing that radio selection option.
- Set parameters “Noopiter” and “Skip Gmin Stepping” by checking their selection boxes:
 - From the LT Spice menu select Simulate, then Control Panel, the SPICE. (Simulate > Control Panel > SPICE);
 - In the “Default DC solve strategy” box in the SPICE tab, tick both the “Noopiter” and “Skip Gmin Stepping” boxes.
- Click “OK”.

LT Spice Set-Up



- It is **STRONGLY** recommended that all bias sources (VBIAS and VDD) powering the EPC Space module models be provided with a finite ramp rate to the final value:
 - A linear or exponential ramp.
 - Suggest 1 us, minimum.
- This finite ramp rate serves to prevent convergence issues/problems at a sharp power supply application (i.e. $t=0$).
- All power supplies have some turn on “feature” with time:
 - They do not rise with infinite slew rate as may be (nearly) set with LT Spice model representations.
- Try to approximate and use the power supply slew rate(s) anticipated in the circuit.

- In some cases just providing a finite power supply ramp rate is not sufficient to achieve consistent simulation convergence.
- If this situation is encountered, it is recommended to add a small resistance in series with the VDD power supply to aid with convergence.
 - 0.0005 Ohms is a good starting point
 - This resistance can be raised and lowered by trial-and-error to achieve the desired convergence stability.
- The requisite series resistance may be added with a discrete resistor or by adding parasitic series resistance in the “Independent Voltage Source” parametric box for the VDD power supply.

EPC Space, LLC

A VPT and EPC Joint Venture

17 Parkridge Road

Haverhill, MA 01835

© 2022