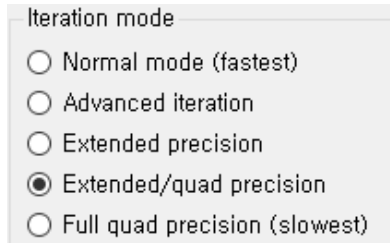


## Simulation Environment Before Simulator

1. Open windows explorer or click on My Computer or open other file manager of your choice.
2. Should be pick the file ([pspice\\_dig.lib](#) in attached) up and drop into the My Computer > Windows > Program files > SIMetrix900 > support > models folder before the simulation run

To replace the PSpice digital model file (pspice\_dig.lib) with SIMetrix subcircuit equivalents for onsemi isolated gate driver models.

3. Recommended use the simulator conversion option below.



## NCP51752 Simetrix model

1. The schematic and simulation is ready to use with SIMetrix 9.0.
2. The user should be able to simulate by pressing the run button.
3. If the NCP51752 is accidentally deleted the user must take the following steps or down load the schematic again
  1. Install the Library
    1. File > Model Library > add/ remove Libraries.
    2. Point the location that the NCP51752 Simulation files were saved during download.
    3. Ensure models are added to the current selected libraries by using the arrows on the select libraries tab.
4. Included models
  1. NCP51752BB for 8.7-V  $V_{CC}$  UVLO with respect to GND2 and 2-V bias between GND2 and VEE
  2. NCP51752BD for 8.7-V  $V_{CC}$  UVLO with respect to GND2 and 5-V bias between GND2 and VEE
  3. NCP51752CA for 12-V  $V_{CC}$  UVLO with respect to GND2 and 2-V bias between GND2 and VEE
  4. NCP51752CB for 12-V  $V_{CC}$  UVLO with respect to GND2 and 3-V bias between GND2 and VEE
  5. NCP51752CD for 12-V  $V_{CC}$  UVLO with respect to GND2 and 5-V bias between GND2 and VEE
  6. NCP51752DB for 17-V  $V_{CC}$  UVLO with respect to GND2 and 5-V bias between GND2 and VEE
  7. NCP51752DD for 17-V  $V_{CC}$  UVLO with respect to GND2 and 5-V bias between GND2 and VEE
  8. ONSEMI\_SiCMOSFET\_1200M3S

## Exclusions from the model

- Thermal behavior is not modeled.