TPS7H1101-SP Pspice model is located on TI website under product toolssoftware tab this is standard for all TI designs. See link below.

<http://www.ti.com/product/TPS7H1101-SP/toolssoftware>

Once you open the model it will contain three schematic that showcase different behaviors. Note information highlighted below is from the Pspice model schematic sheet.

TPS7H1101-SP simulation contains three Schematics.

**Schematic 1 – transient as shown below**.



Running this simulation will show the transient behavior Vin-turn on, Vout, powergood (PG)



**Schematic 2 – is similar to schematic one except it addresses when two LDO’s are in parallel.**.



**Schematic 3 - TPS7H1101-SP Output Impedance addresses - stability.**

**Here one measures output impedance as shown below.**

Group delay is converter to phase margin as shown by the equation highlighted.











