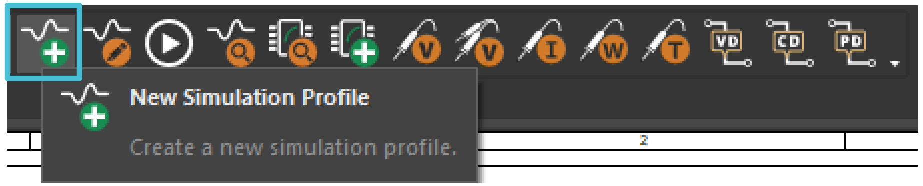
Simulation profiles

TI Precision Labs – PSpice® for TI

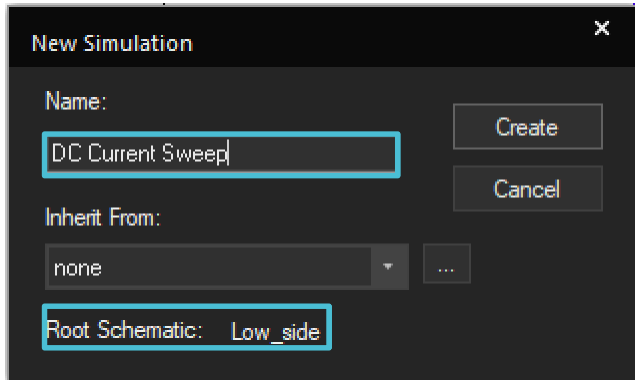
Creating a new simulation profile

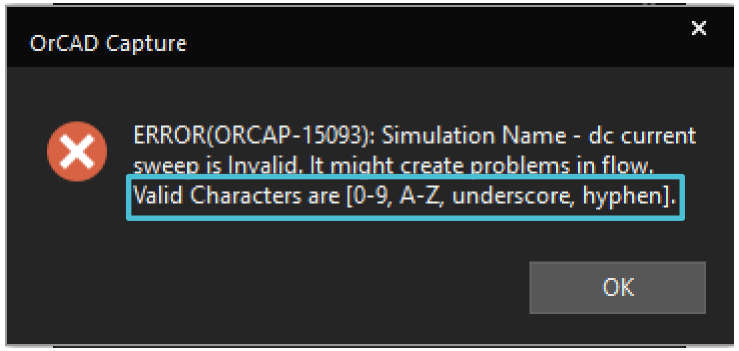
- Navigate to the “New Simulation Profile” with the root schematic open

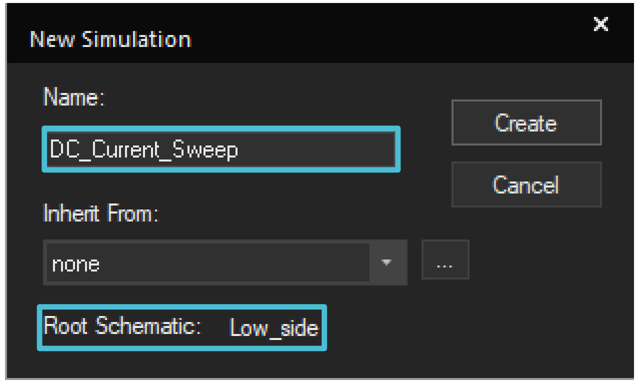


Simulation profiles naming conventions

- If the name chosen creates problems within a file path (invalid characters) the screen will prompt to chose a different name

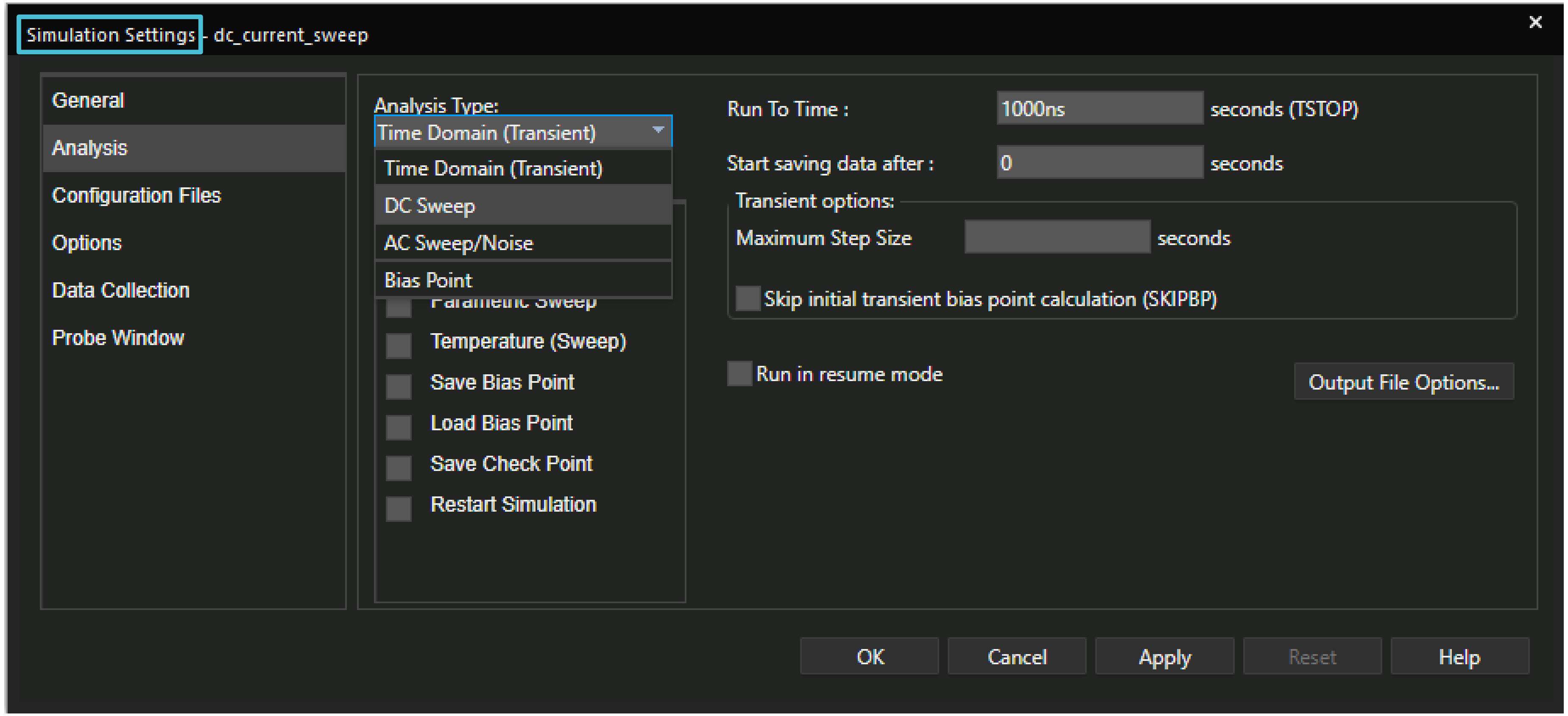






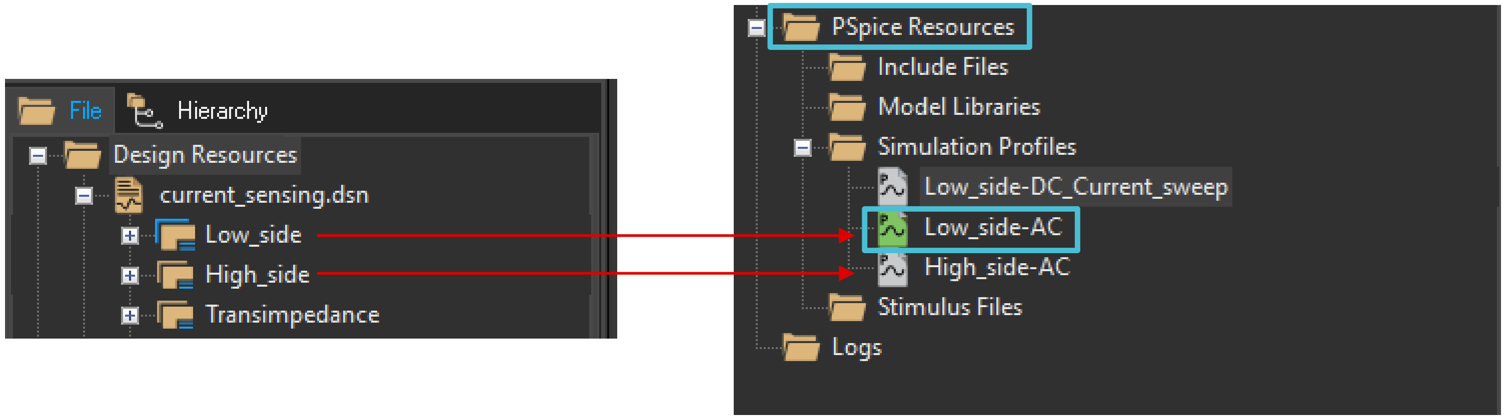
Simulation settings

- Simulation settings window allows for the setup of the simulation profile



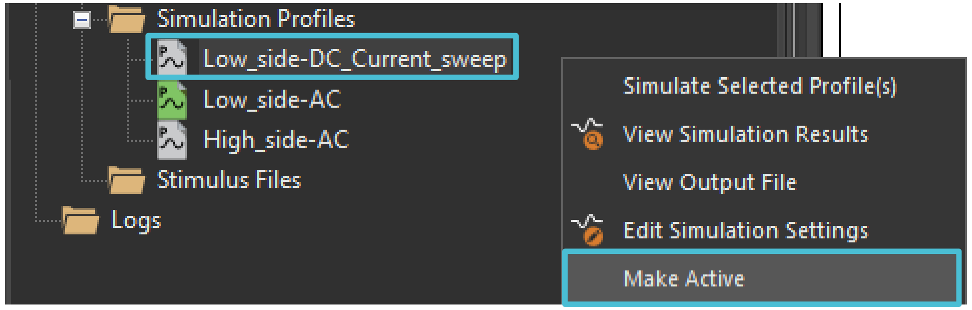
Simulation profiles location

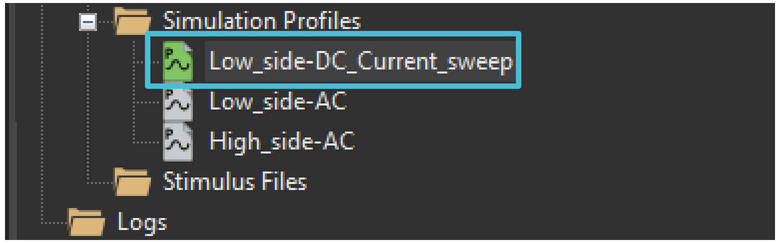
- Simulation profiles are located under PSpice Resources in the docked project explorer

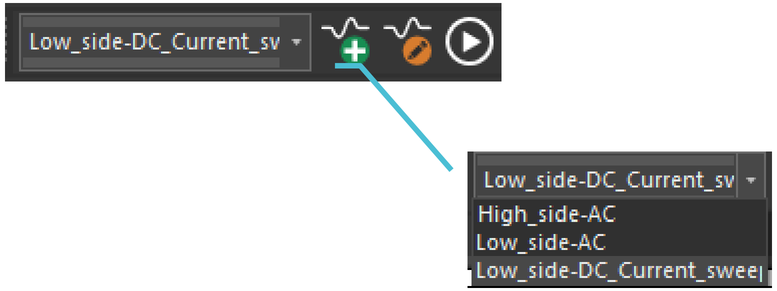


Switch between simulation profiles

- The active simulation profile for a schematic is highlighted in green
- Right click on the simulation profile name and select “Make Active” or select through the task bar drop down menu

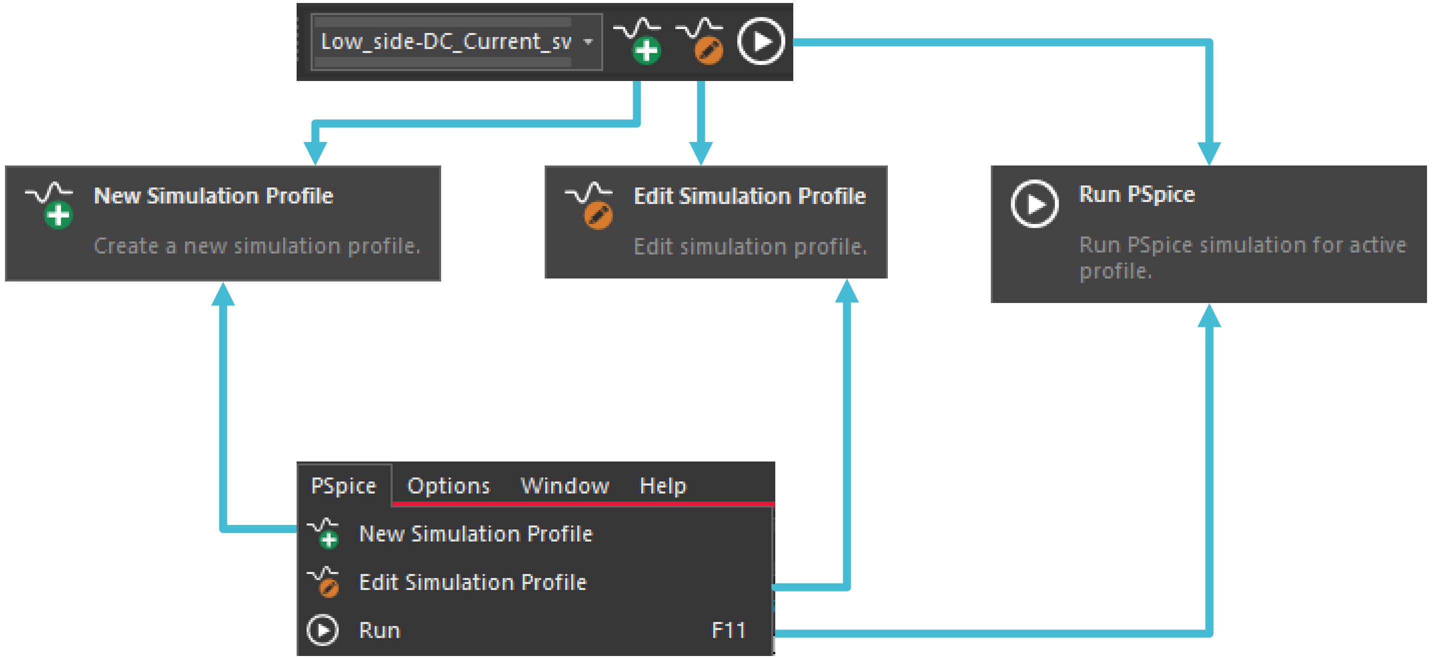






Simulation taskbar or the PSpice menu?

- The task bar and the PSpice menu both allow to create, edit, and run a simulation profile



Thanks for your time!

To find more **PSpice® for TI**
technical resources and search products,
visit **ti.com/tool/PSPICE-FOR-TI**.