

Symbol Creation for MCP651 Opamp

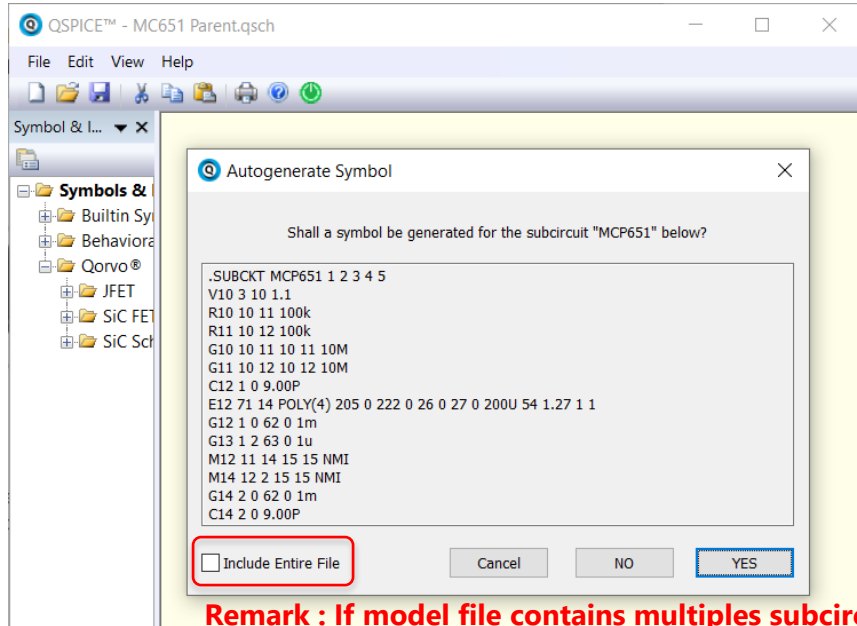
KS Kelvin Kelvin Leung
9-22-2023

Symbol Creation for MCP651 Opamp

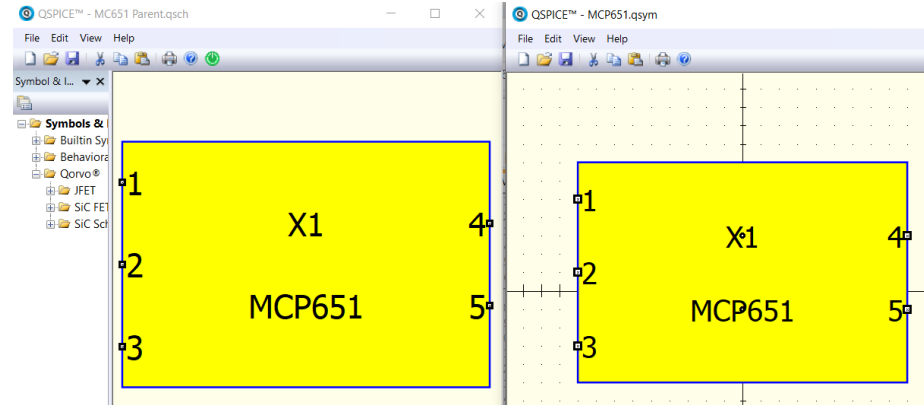
Qspice : MCP651.txt; MC651 Parent.qsch ; MCP651.qsym

- [1] Use text document to open MCP651.txt
- [2] Copy entire text in subckt file
- [3] In Qspice schematic, Ctrl-V paste text
- [4] Autogenerate Symbol windows pop up > Yes
- [5] Place the Autogenerated Symbol in schematic

- [6] In schematic window, mouse cursor over symbol, Ctrl-C to copy it
- [7] File > New > New Symbol
- [8] Click on Symbol window, Ctrl-V to paste it
- [9] Mouse cursor on edge of yellow box, press Delete to delete yellow box



Remark : If model file contains multiples subcircuit supporting each others, this option must be selected

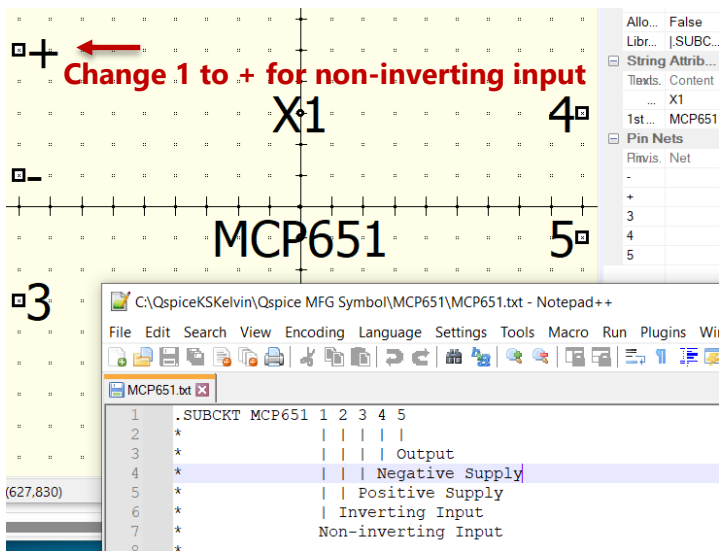


Symbol Creation for MCP651 Opamp

Qspice : MCP651.txt; MC651 Parent.qsch ; MCP651.qsym

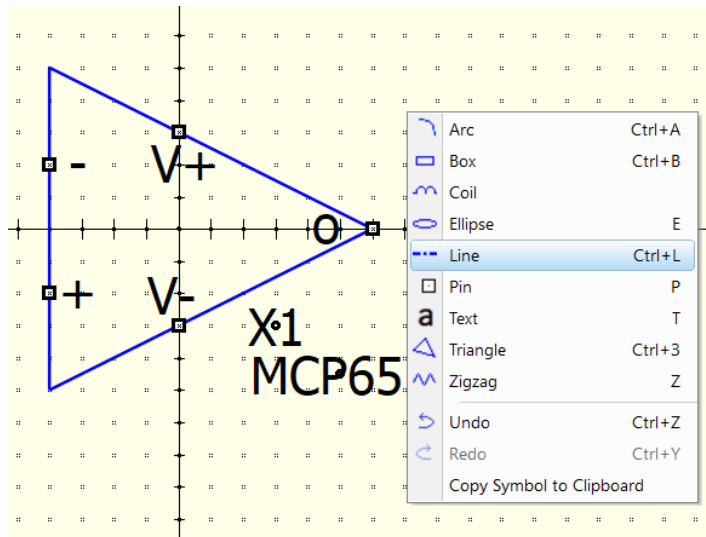
[10] Open subckt text, and find pin ID and node name

[11] Double click on Net, and change Pin name (any name you think it proper)

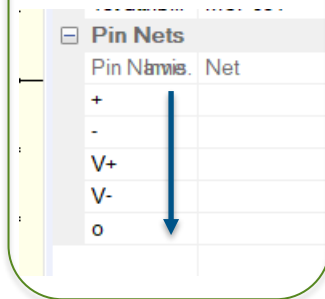


[12] Rearrange pin location and right click for various drawing tool to complete the symbol as you like

[13] Save the symbol



** close and re-open symbol (.qsym), in symbol properties, Pin Nets should follow order and the name you decided



Symbol Creation for MCP651 Opamp

Qspice : MCP651.txt; MC651 Parent.qsch ; MCP651.qsym

[14] From window explorer, drag the symbol (e.g. MCP651.qsym) directly to schematic window for placement

[15] Draw a complete schematic and run the simulation. Enjoy!!

