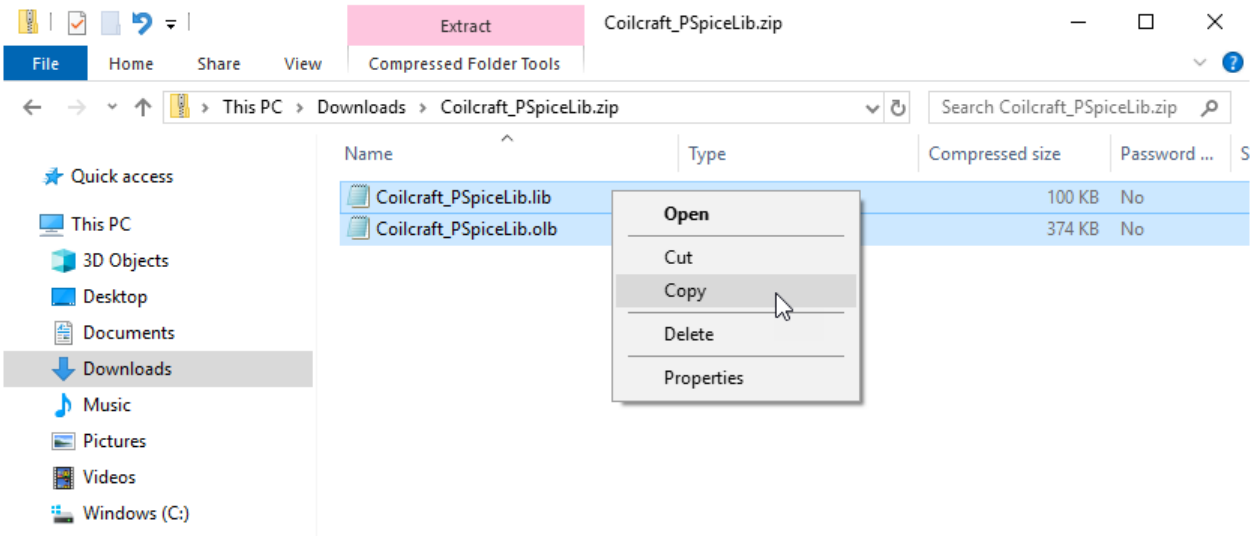
Model libraries for PSpice®

Installing Coilcraft Advanced model libraries

1) Download the Coilcraft_PSpiceLib.zip file

2) Extract the .zip file and navigate into the extracted folder

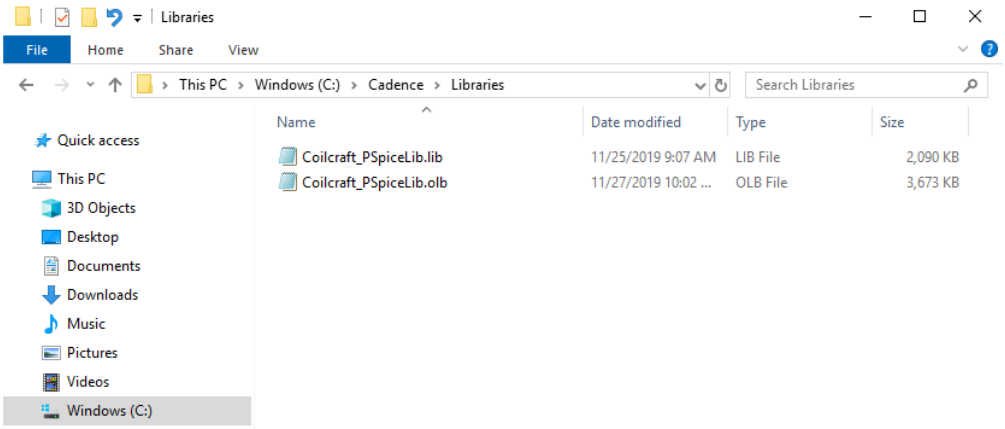


3) Copy the contents from the extracted folder

4) Navigate to “C:\Cadence”
   a. This should be the default install location for OrCAD/PSpice
   b. Create a new folder and name it Libraries.

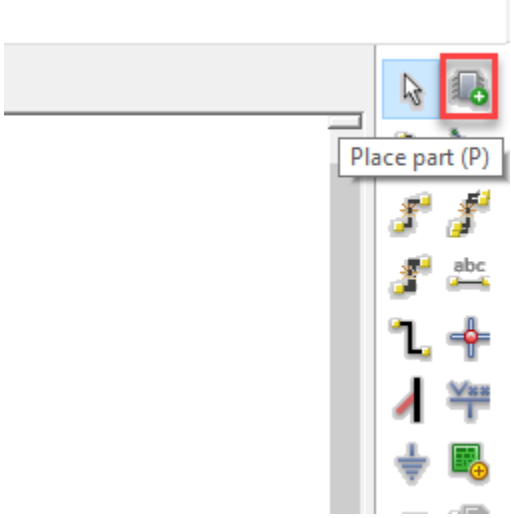
*Alternatively these files can be placed anywhere that makes sense to the user.

5) Paste the files from the Coilcraft_PSpiceLib.zip here.
   a. If you already have the Coilcraft Advanced model library and are updating you will need to choose to replace the destination files.

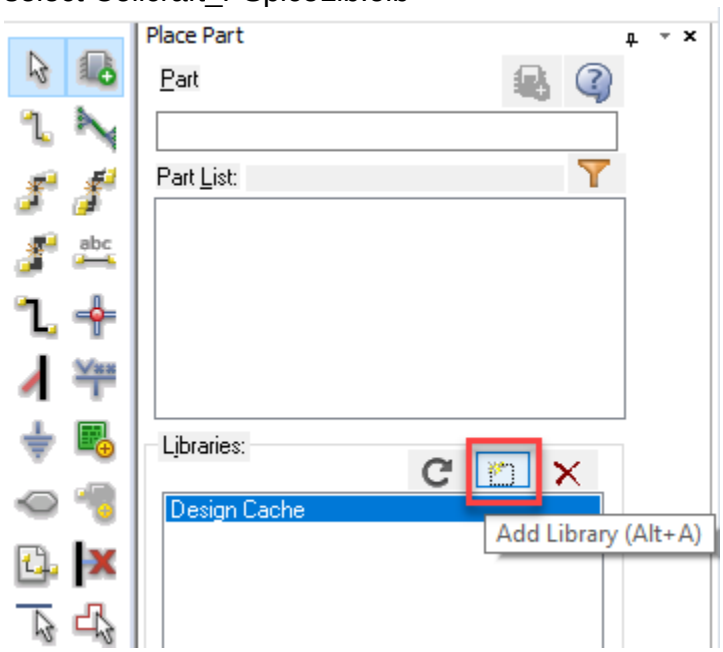


Using Coilcraft Advanced models

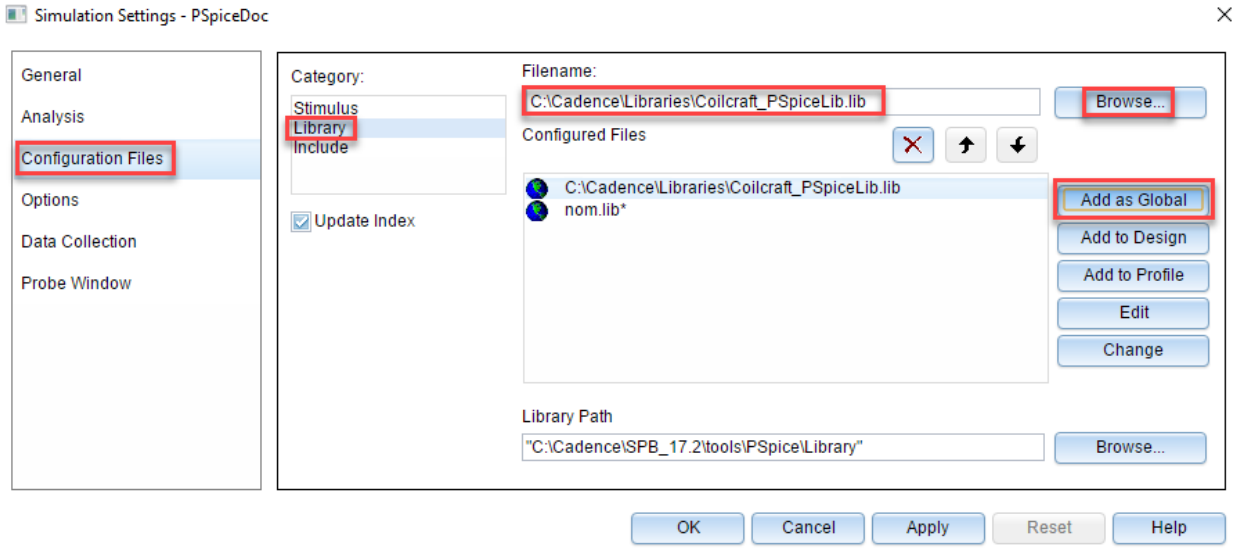
1) To use a Coilcraft Advanced inductor model simply select the place part icon



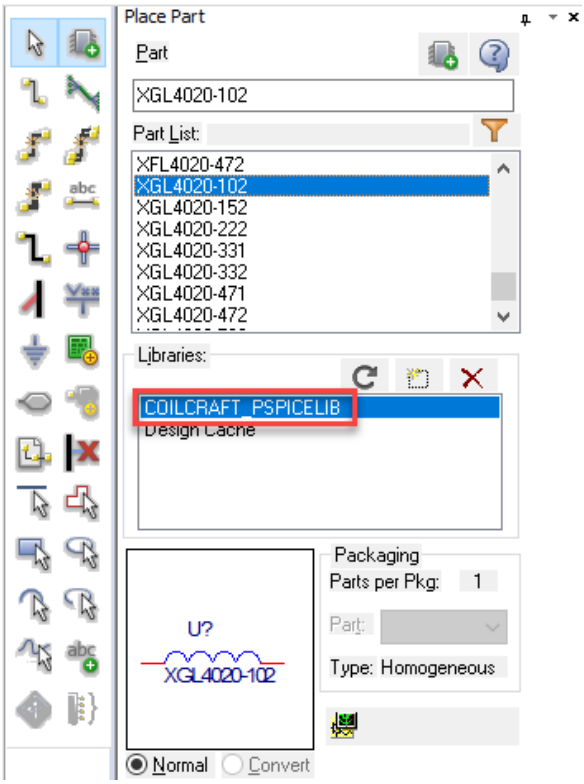
2) Click on Add Library and once you have navigated to the location you saved the library files to select Coilcraft_PSpiceLib.olb



3) Ensure the library is included in the simulation settings
    a. Settings can be accessed when creating a new simulation profile or by editing the current one
    b. Select the configuration files tab and with the library category highlighted browse to the save location selecting Coilcraft_PSpiceLib.lib
    c. Select Add as Global to make the library available for future use or Add to Design to add the library for only the current design.



4) With the Coilcraft_PSpiceLib library selected search the parts list for the inductor you want to include in your design. Double clicking the part number will select it and allow the placing of the inductor onto the schematic.



Plotting Waveforms

To plot Inductance vs frequency:
IMG(V(Vin)/I(R2))/(2*pi*frequency)

To plot Impedance vs frequency:
V(Vin)/I(R2)

To plot ESR vs frequency:
R(V(Vin)/I(R2))

To plot Q factor vs frequency:
ABS(IMG(V(Vin)/I(R2))/R(V(Vin)/I(R2)))

*note voltage node names may be different it is up to user to ensure voltage is across inductor