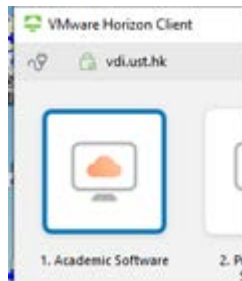




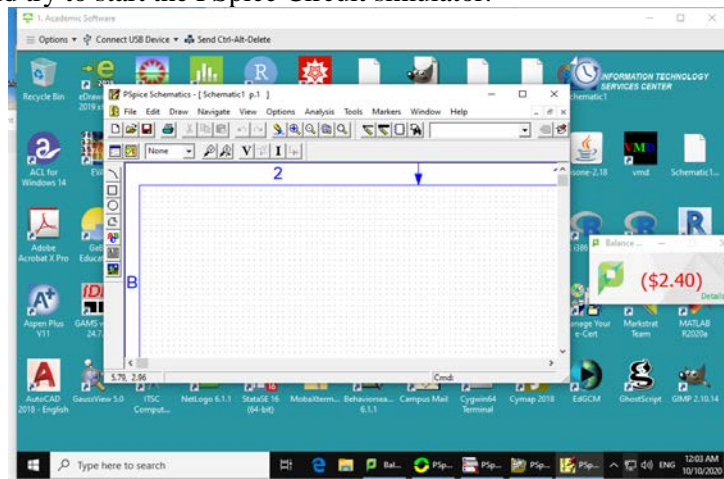
PSpice Circuit Simulator installation

1. Online Mode

- You have to check with the ITSC Website to get more information about the Virtual Barn.
<https://itsc.ust.hk/services/academic-teaching-support/facilities/virtual-barn/user-guide>
- You should try to login into the Virtual Barn with Academic Software pool from your own computer at your place.



- You should try to start the PSpice Circuit simulator.



2. Run PSpice on your own computer

- For the Mac user – The PSpice software could not be run in Mac. You have to use the online mode.
- For Window user – You could download the PSpice circuit simulator and install into your Window PC.

Canvas (Elec2400) -> Lab Material -> 5) PSpice software