Simulation Software

PSpice

Pspice 9.1 (Windows Circuit Simulator)

<https://drive.google.com/file/d/1VOtUYsaFzPAUfJTuCCqaetKoTI-eOOYB/view?usp=sharing>

Pspice 9.2 (Windows Circuit Simulator)

<https://drive.google.com/file/d/1oXTMyhgSKoiPOlfoFOoZflTsFXX97GA1/view?usp=sharing>

Proteus

Proteus 8.11

<https://docs.google.com/document/d/e/2PACX-1vTGGxmgSCuE4Y8WBFubmxQdTxoHkA7XqVa3VmWdiKYH8X896FA3O8ldBqs1VhMMyPLsC9-w7dEnYOqX/pub>

Google Docs

PSpice\_student\_version\_9.1.rar

Google Docs

ORCAD PSpice 9.2 family release.rar