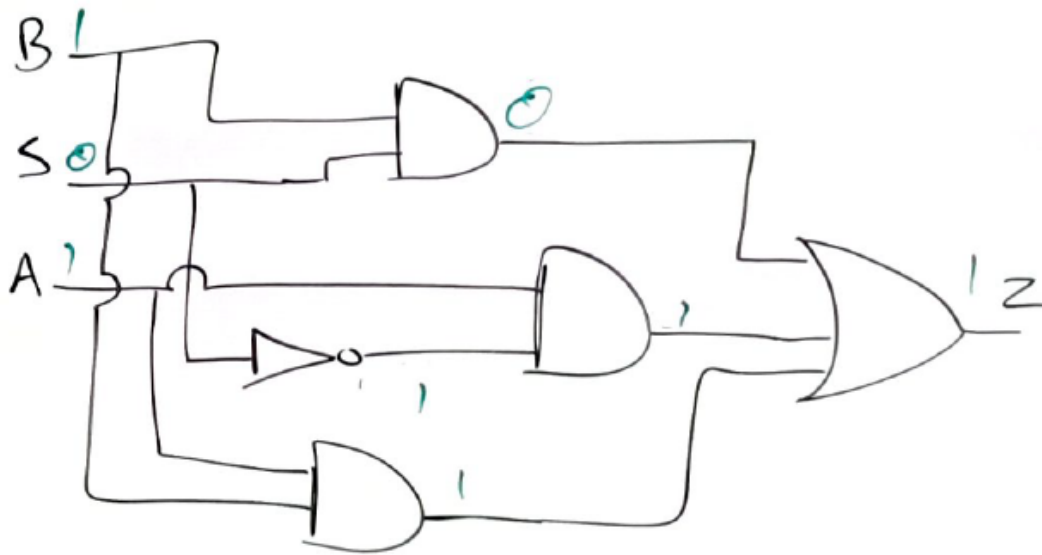**CET246 Electronic Design Automation**
David J. Broderick, Ph.D
Laboratory Exercise #7: Custom Schematic Symbols

What to do:

1) Open KiCad and start a new project
2) Create a new schematic in Eeschema (within KiCad)
3) Create a project library in the symbol editor
4) Create all of the symbols needed for the schematic drawn below

What to turn in:

1) A single zip file with all schematic and library files. Name the file according to this format:

    Course_Semester_YourLastName_Lab##.zip

    So I would name my file:

    CET246_Fall2018_Broderick_Lab06.zip