

## Help in Setting Up Eagle CAD

The first time you run Eagle CAD from your account you will want to set the following options:

### Menu Options

#### Directories

Libraries	V:\eagle\lbr
Design Rules	V:\eagle\dru
User Language Programs	V:\eagle\ulp
Scripts	V:\eagle\scr
CAM Jobs	V:\eagle\cam
Projects	V:\eagle\shared_projects;w:\eagle

(you will need to create the “eagle” directory on your w:\ drive

### Menu Options

#### User Interface

Leave all the defaults, but also check “command texts”

Expand the “Libraries” category. This takes about 20 seconds to load all the libraries. After the libraries have been expanded right click on the “Libraries” item and select “Use All”. This should put a green dot to the far left of each listed library. You may have to change the size of the window to see the green dot. Finally select File and Save all.

To create a new project:

In Control Panel under Projects and your w:\eagle directory, right click and create a new project. Give it a name.

Then right click on the project name and “add a schematic”.

Then use Add, Move, Delete to organize your components in the correct places.

Wire up your circuit with wires and busses.

You will only need the following component libraries

74xx-us.lbr Use any logic in this library. Some of the items are listed below

74\*00

74LS00N NAND Gate

74\*02

74LS02N NOR Gate

74\*04

74LS04N Inverter Gate

74\*520

74LS520 Digital Comparator us for the 74LS521

custom.lbr

DS17887

Realtime clock chip

DSK\_6713\_MECHLAB

DSP Pins

