Introduction to Cadence's Capture CIS for users of Schematics

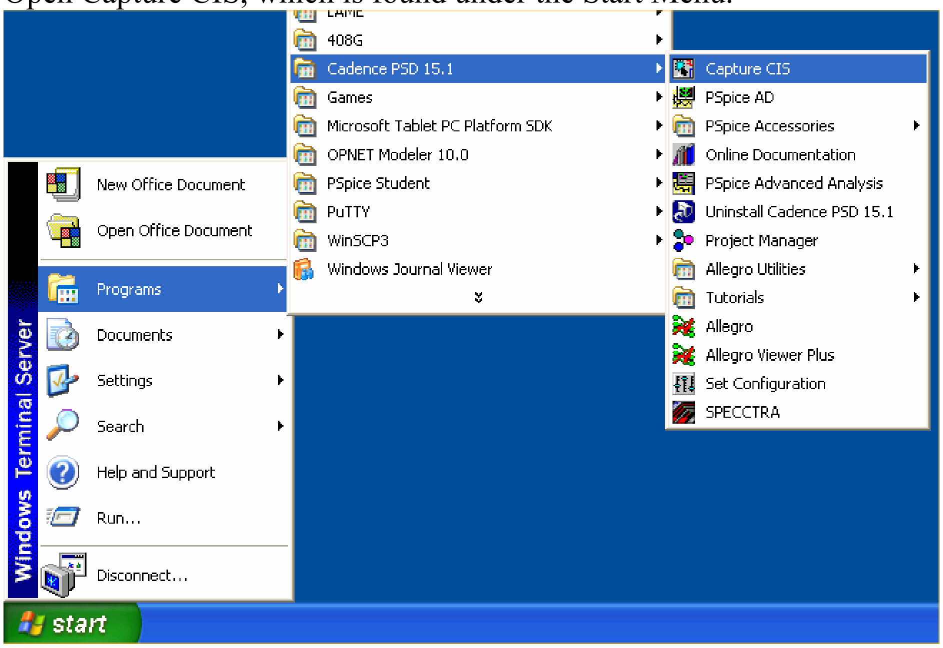
Nicholas Ganig (nganig@eng.umd.edu) 05 February 2004

Introduction

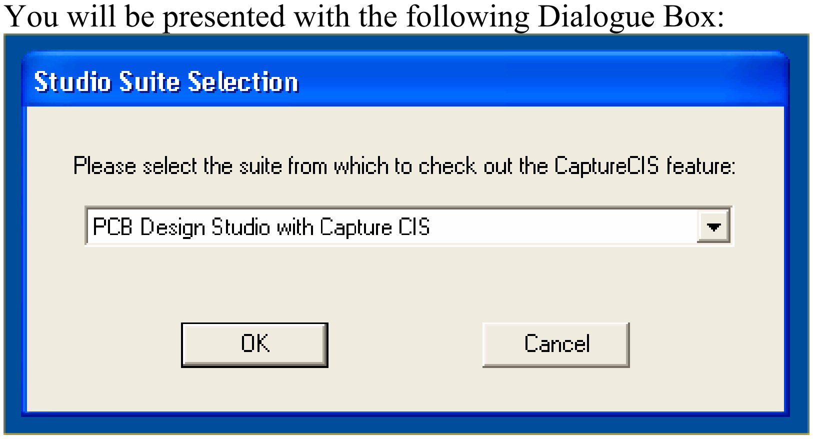
With the release of Cadence PSD 15.1, the Schematics CAD program is no longer present, requiring a retraining of our students and faculty. This guide is an attempt to ease the transition from PSD 14.2's Schematics program to PSD 15.1's Capture CIS Program. While Capture CIS has many more features than Schematics, I will only cover those which were relevant to Schematics users. Similarly, there were features used in Schematics that I have not yet encountered, and so this guide is not intended to be complete, instead coving only the essentials. I trust that you can learn the remainder on your own.

Creating a New Schematic

Open Capture CIS, which is found under the Start Menu.

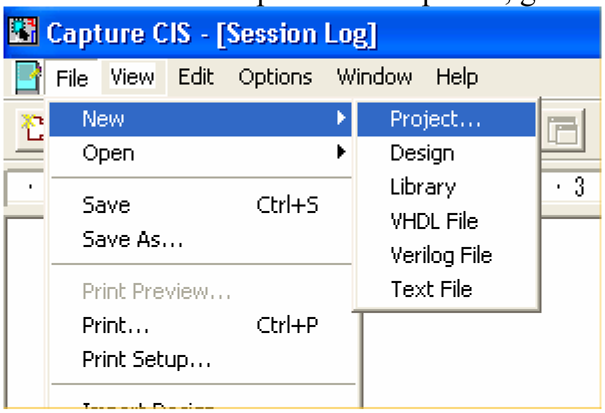


You will be presented with the following Dialogue Box:

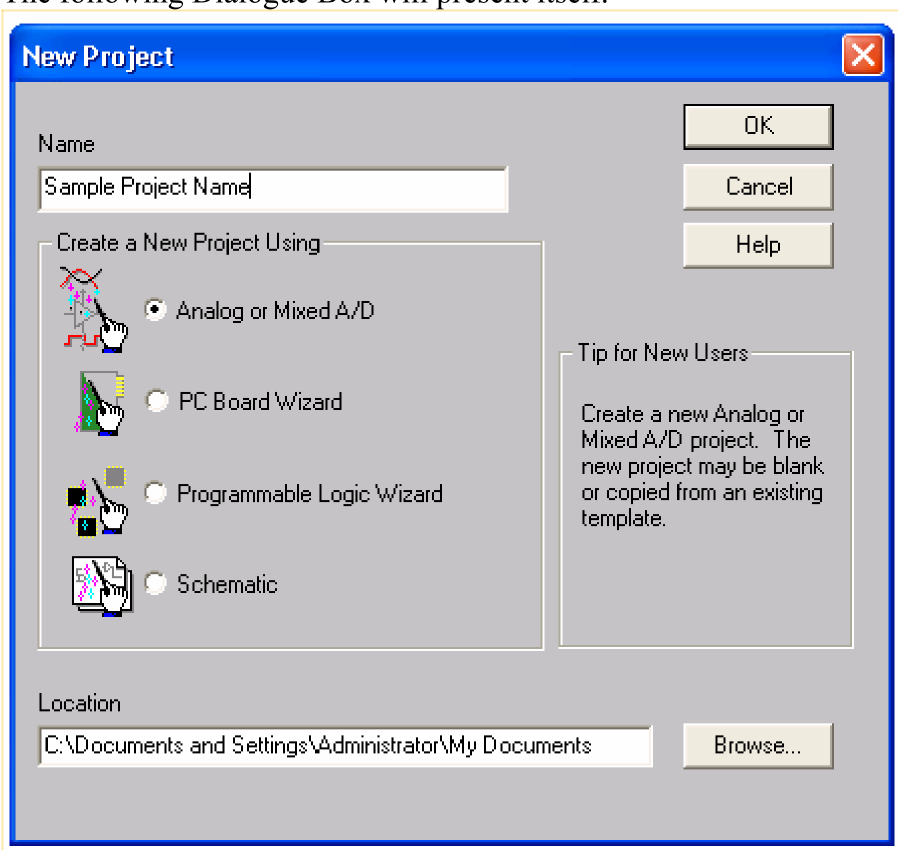


Choose 'PCB Design Studio with Capture CIS' and click OK.

Once the Cadence splash screen passes, go to the File Menu, and select New, Project...



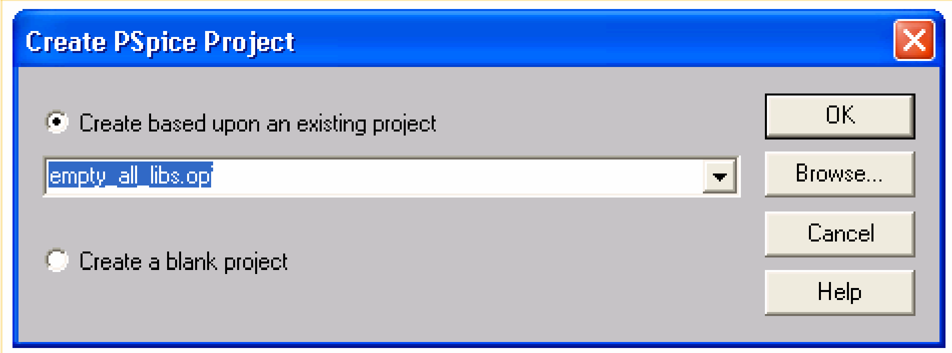
The following Dialogue Box will present itself.



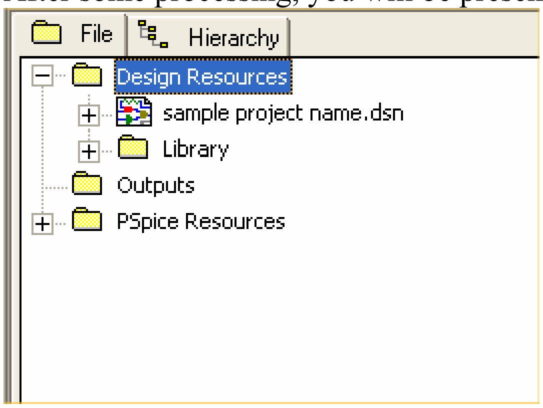
Choose what folder you'd like to save the project in. The Project consists of several files, so a dedicated folder is ideal. Furthermore, I recommend saving this on the 'U:' drive, but if you'd like, the path to the 'My Documents' folder is above. Simply substitute your username for 'Administrator.'

Make sure you have selected 'Analog or Mixed A/D.' under 'Create a New Project Using' You must also name the project, any name is acceptable. Click OK to continue.

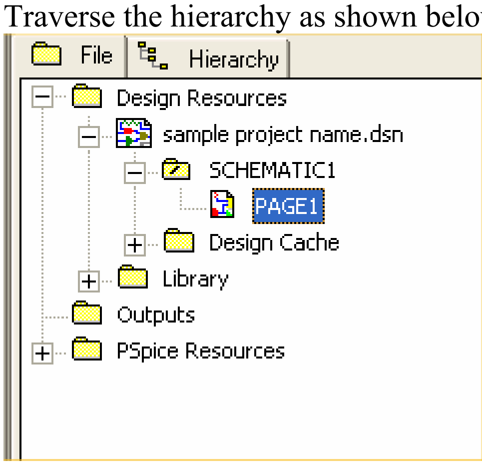
The next dialogue you will be presented with asks which template you'd like to use. Choose 'empty_all_libs.opj' and hit ok. Do not create a blank project, as you will have no parts to work with.



After some processing, you will be presented with your project's hierarchy.

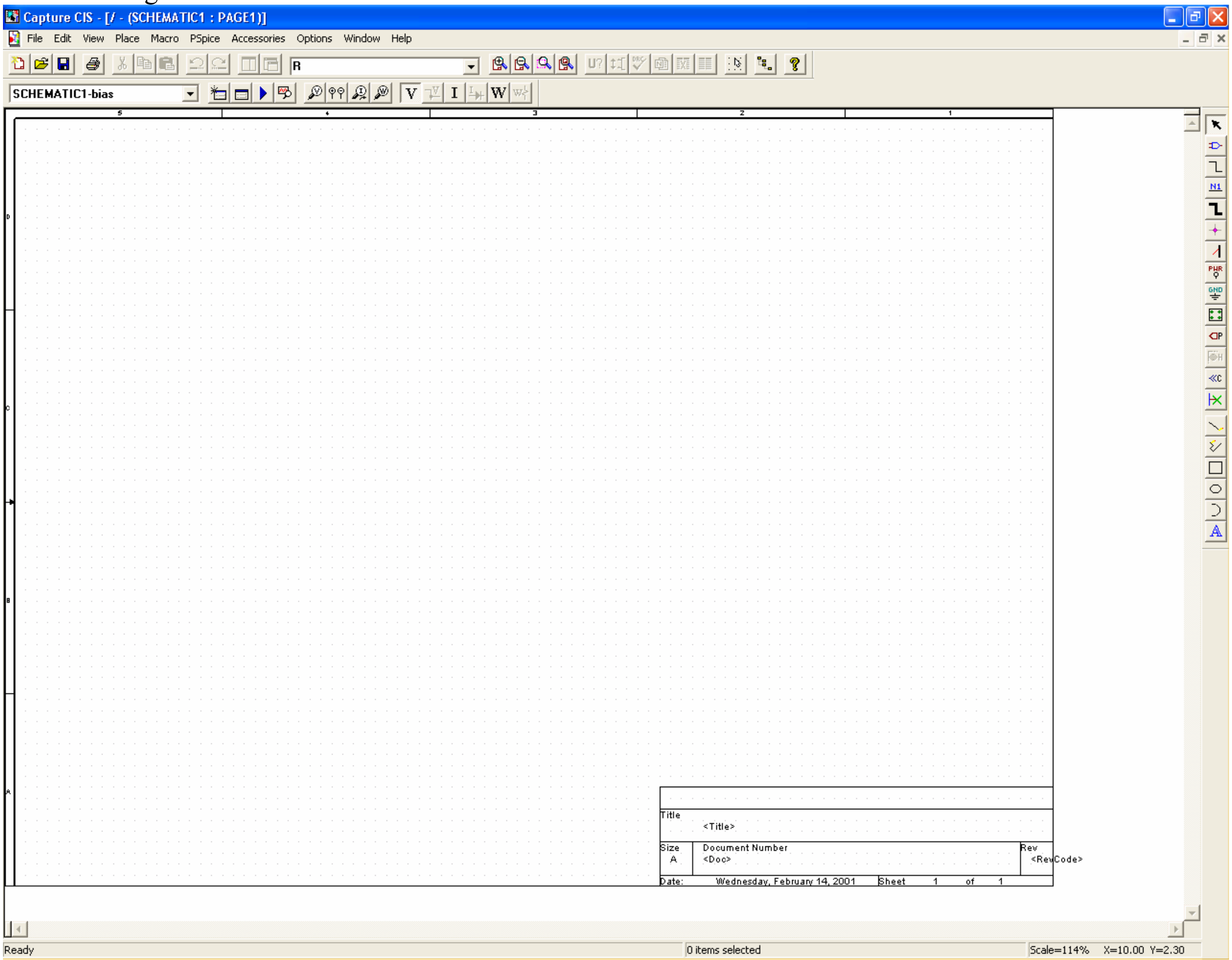


Traverse the hierarchy as shown below, to access the schematic.



Double-click on 'PAGE1' to access the schematic.

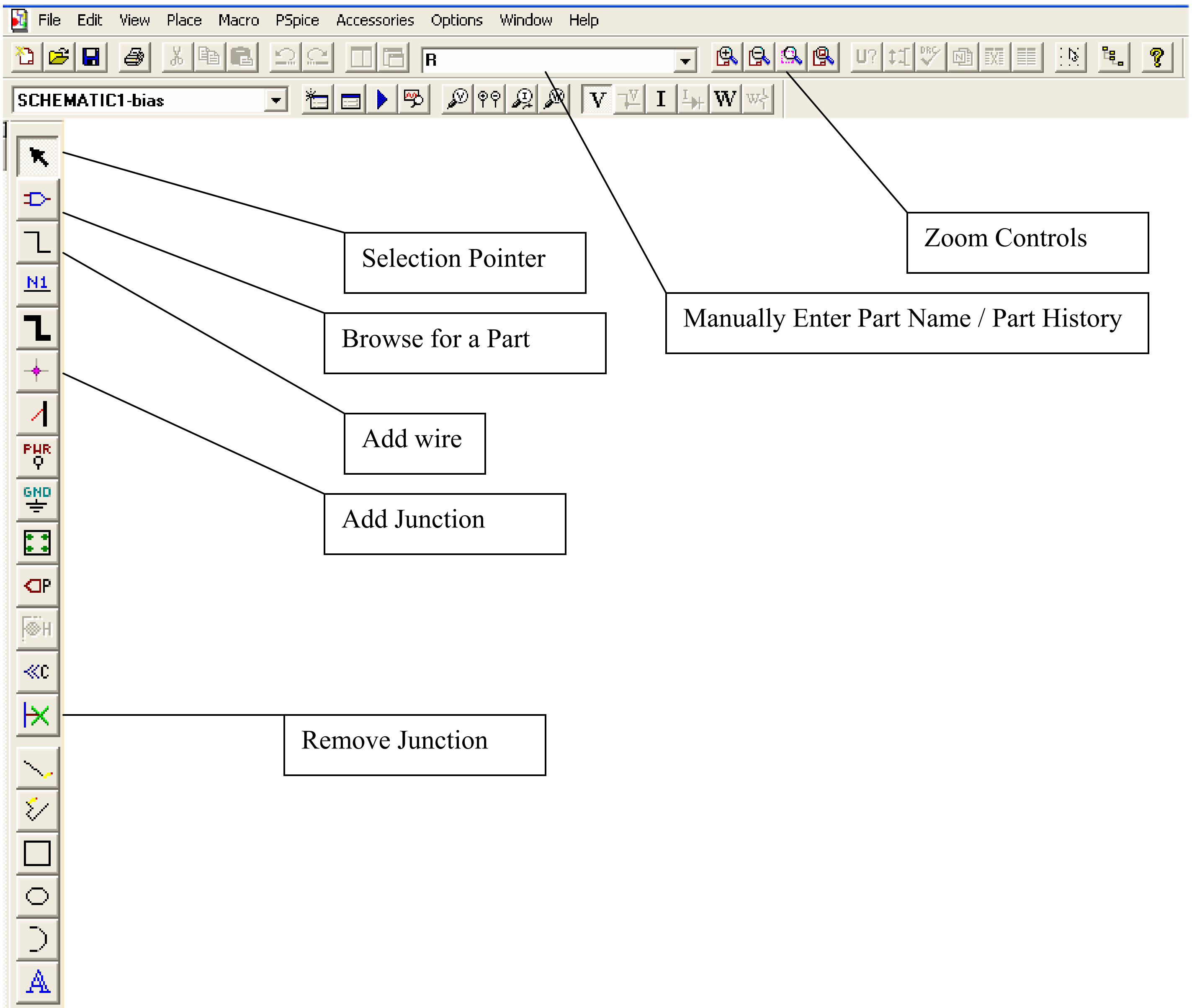
The following window should look somewhat familiar.



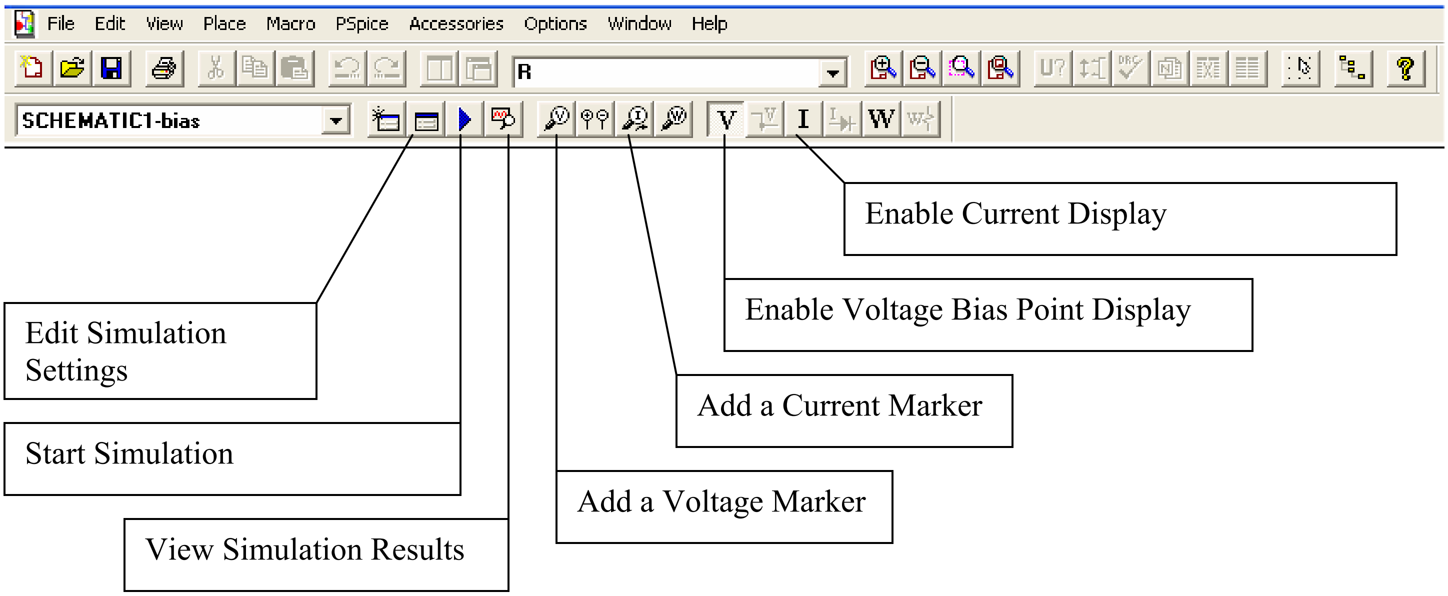
Now you have a new schematic.

Composing a New Circuit

Thankfully, once you have created a new schematic the user interface is relatively similar. I will cover the layout of the two main tool palettes, considering their relevance to Schematics users.

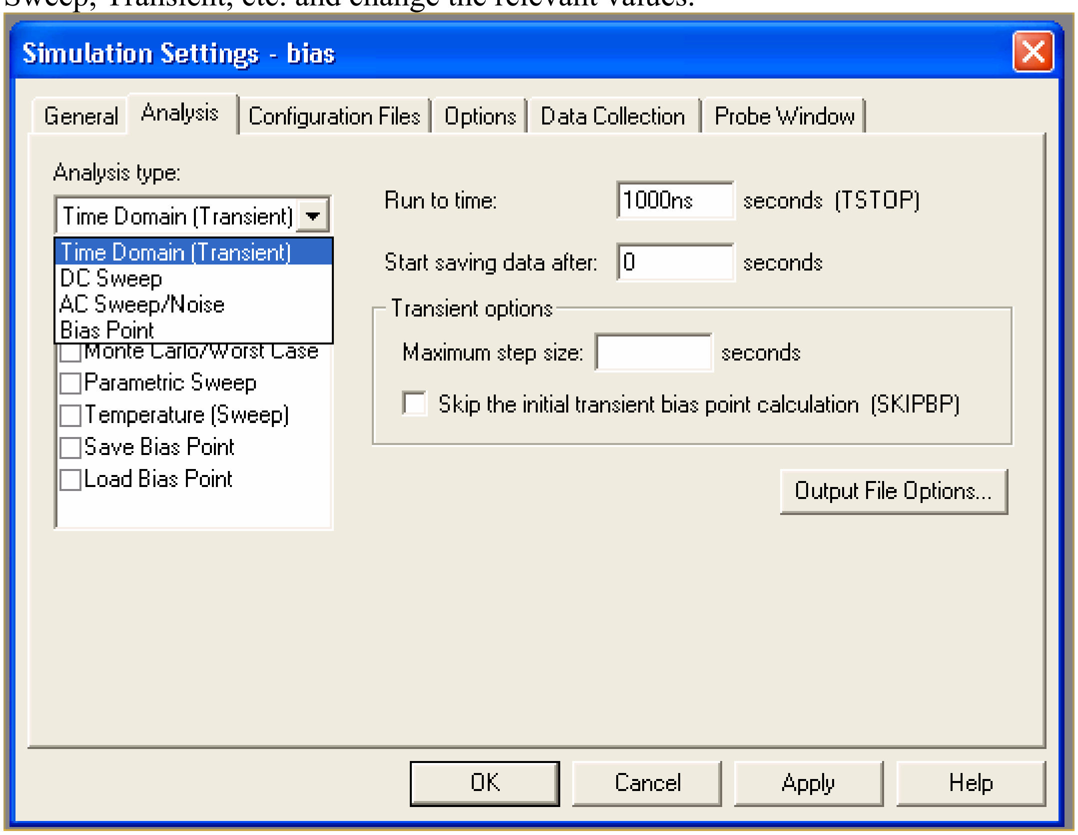


Simulation and Measurement Controls



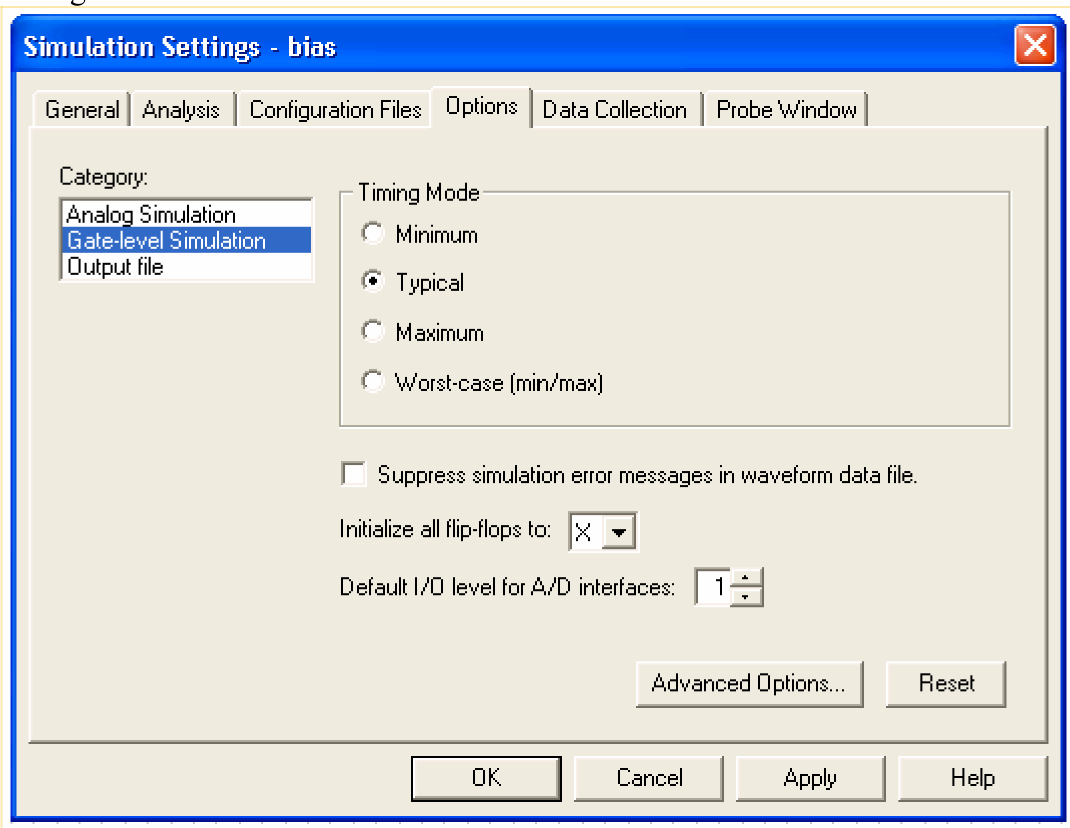
Detail of 'Edit Simulation Settings' Dialogue Box

This is of particular importance to our users, as this is where you can choose the type of simulation: i.e. DC Sweep, Transient, etc. and change the relevant values.



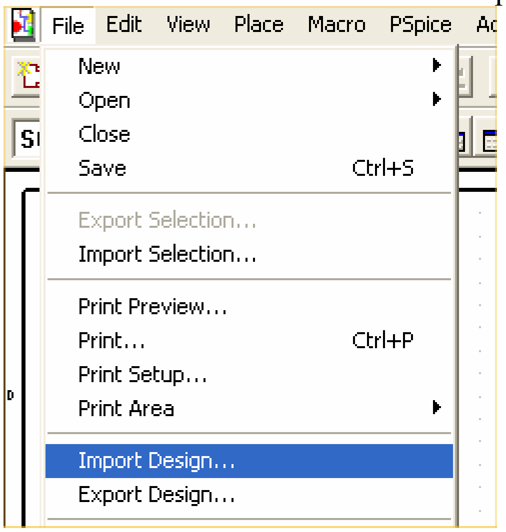
Detail of ‘Edit Simulation Settings’ Dialogue Box

This is also important, especially for digital simulations featuring flip-flops, as this tab contains the ‘initial state’ setting.

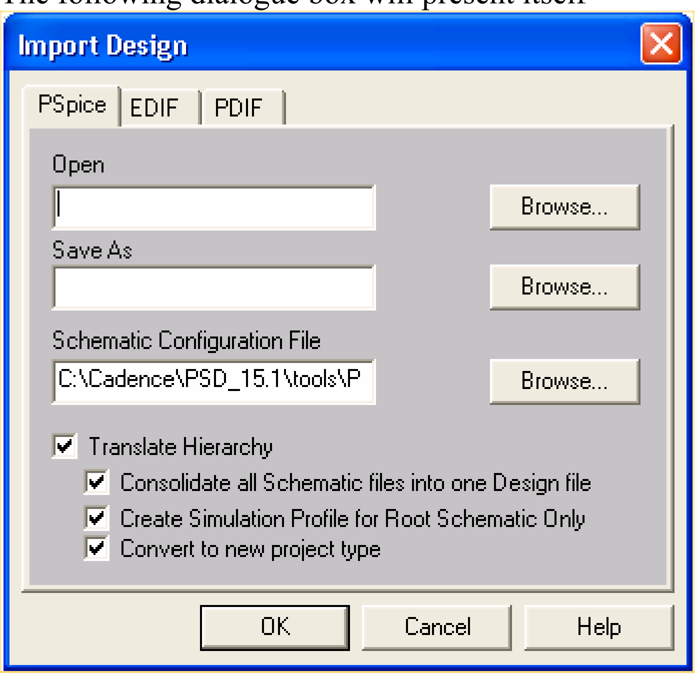


Importing an Old .sch Schematic File
This is a relatively straightforward procedure.

Click on the file menu and choose 'Import Design…'

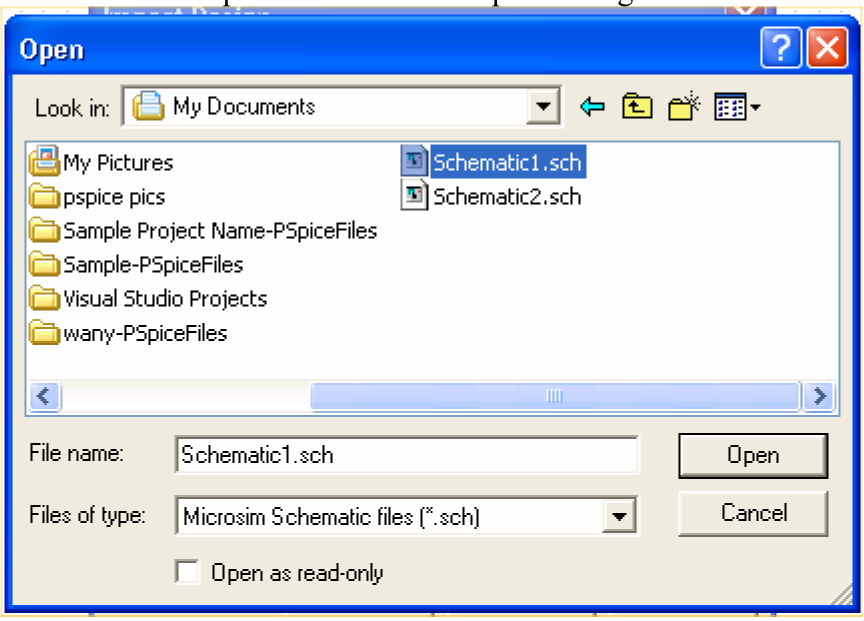


The following dialogue box will present itself



Click the 'Browse…' button next to 'Open'

You will then be presented with file Open Dialogue box like this



As long as the 'Files of Type:' menu reads 'Microsim Schematic files (*.sch) as it does above, simply find your schematic file and click open. You will be presented with the previous dialogue again, where you'll need to choose a save point. The rest should be easy to figure out.

Disclaimer

The contents of this document are correct to the best of my knowledge. I believe that I have covered all that is required to complete UMD courses such as ENEE 204 and 206, but if you feel that I've left something out, or included an error, please contact me. If you have other Pspice questions, please consult the online help or contact your instructor.