A Method to Import PSpice Models into OrCAD

LT.Cdr.Krisada Sangpetchsong
Lecturer

1 Abstract

Computer simulation of electronic circuits is becoming an indispensable step in electronic system design, development, and validation. PSpice is a well-known software module used for electronic circuit simulation. This paper describes a method for importing electronic component PSpice models in a legacy text file format into OrCAD for PSpice simulation.

2 Introduction

Whenever possible, it is always a good practice to design and validate electronic circuits using computer programs and simulations prior to actually building the circuits. This practice is valuable for several reasons. Firstly, it can help ensure designers of proper circuit responses early in the design life cycle. Subsequently, it can help reduce time needed for hardware debugging, leading to shorter overall development time. Designers can also perform preliminary sensitivity analysis to select components values that provide optimal performance within design constraints.

With advances in electronics Computer Aided Design (CAD) software packages, electronics engineers today have several powerful tools for uses at their disposal. At present, one of the most popular electronics CAD packages is OrCAD. This software affords designers a complete set of tools for both analog and digital circuit design and simulation, sensitivity and smoke analysis, and lastly for PCB design and manufacturing. PSpice is a well-established software module used for electronic circuits design and simulation, included in OrCAD package. With today's trend, it is very easy for an engineer to download PSpice models of most electronic components from any well-known manufacturers' websites. The main problem is that most models are available in PSpice legacy text file format which is not readily applicable for simulation using PSpice. It is therefore beneficial to examine how to manipulate these text file models for simulation.

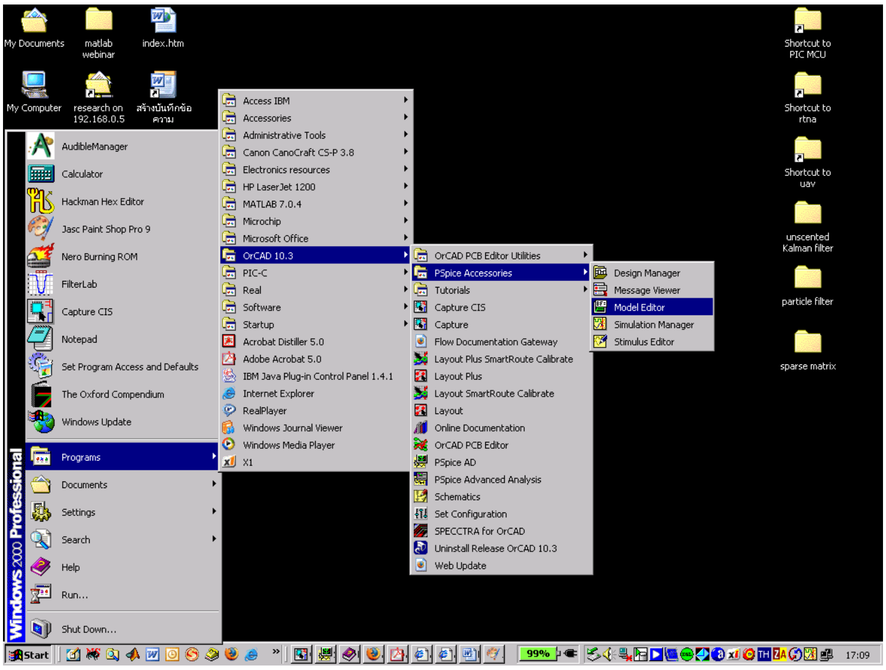
Operational Amplifiers (OpAmp) are the most basic components in analog circuits. It is as important for an analog circuit designers as a screwdriver is to any mechanics. OpAmp is used in such fundamental and important applications as signal conditioning (amplification, level shifting, inversion, summation, subtraction) and analog active filtering (lowpass, bandpass, highpass). This paper is the first part of a two-part story. It describes how to import downloaded text file format models into OrCAD. As an example, the model used is a Microchip OpAmp part number MCP616. The second portion will present an example of how to use the imported model in anti-aliasing filter design and validation.

3 Importing the model

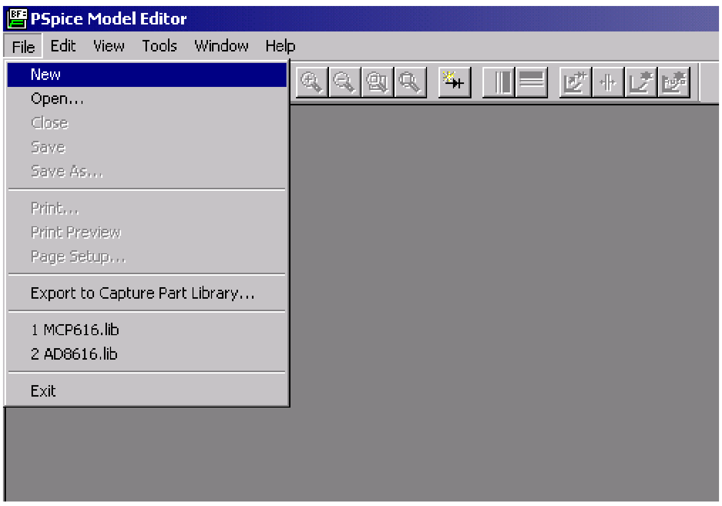
Download required PSpice models from a supplier's website, e.g. Texas Instrument, Analog Device, National Semiconductor. Model files are ASCII text files, usually end

with *.cir* or *.txt*. An example model for Microchip OpAmp is available for download from http://ww1.microchip.c
om/downloads/en/DeviceDoc/MCP616_MM_B.txt.

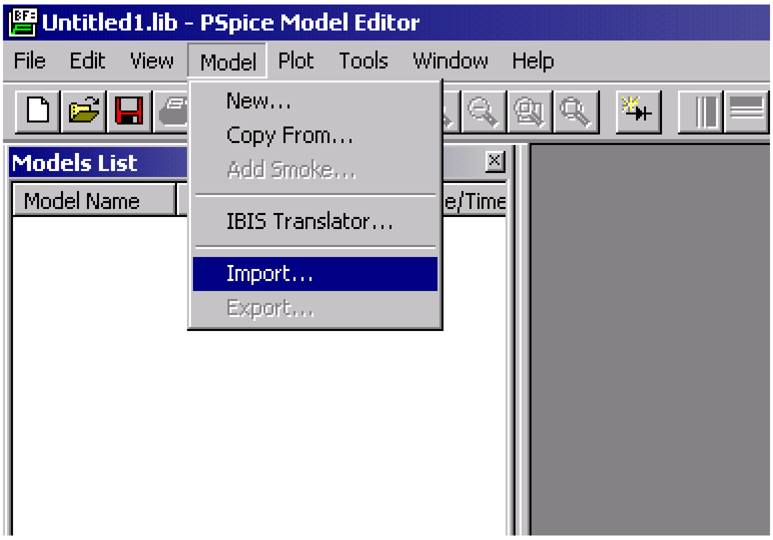
1. Open PSpice Model Editor.



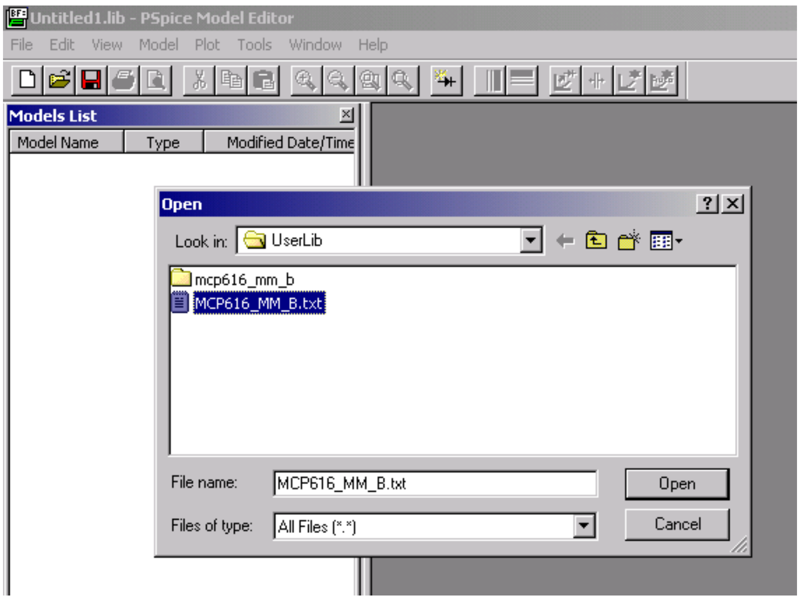
2. In Model Editor Select *File -> New*



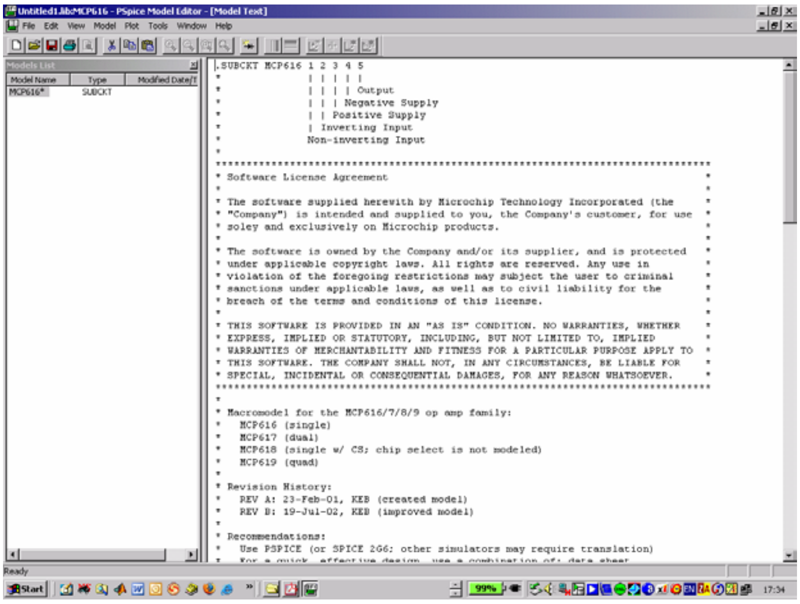
3. Select *Model -> Import*



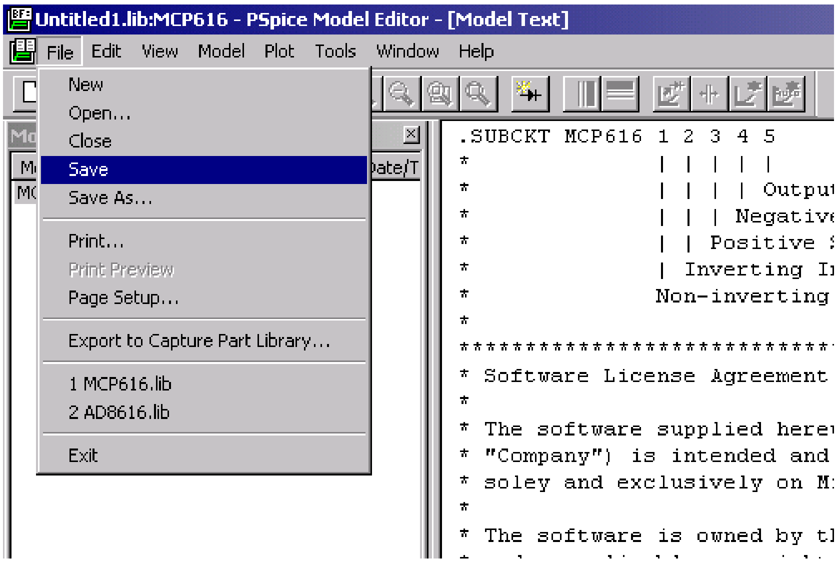
4. Select the Model (.cir or .txt) file which was saved earlier.



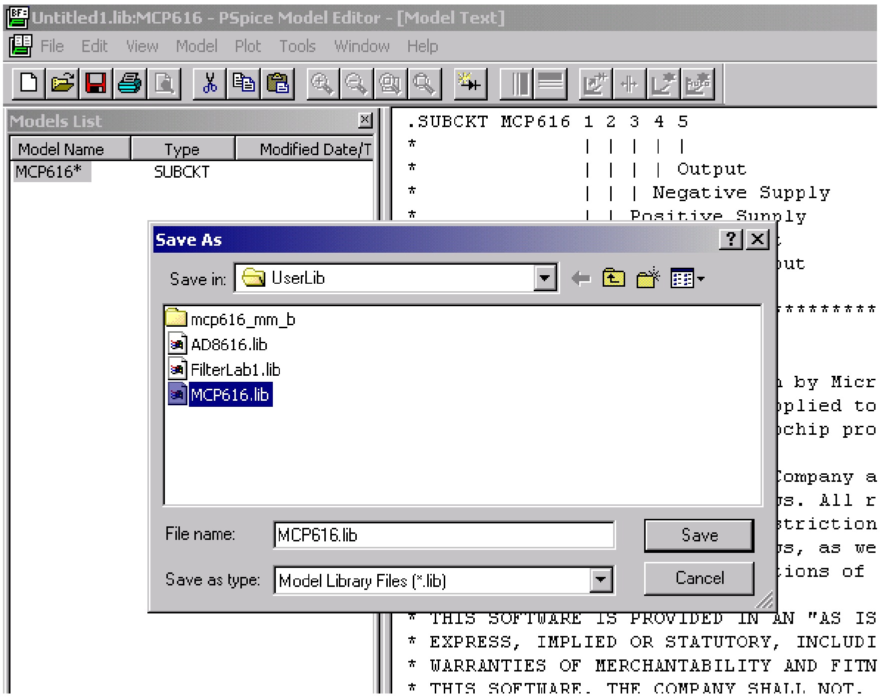
The Model file will be displayed as follows.



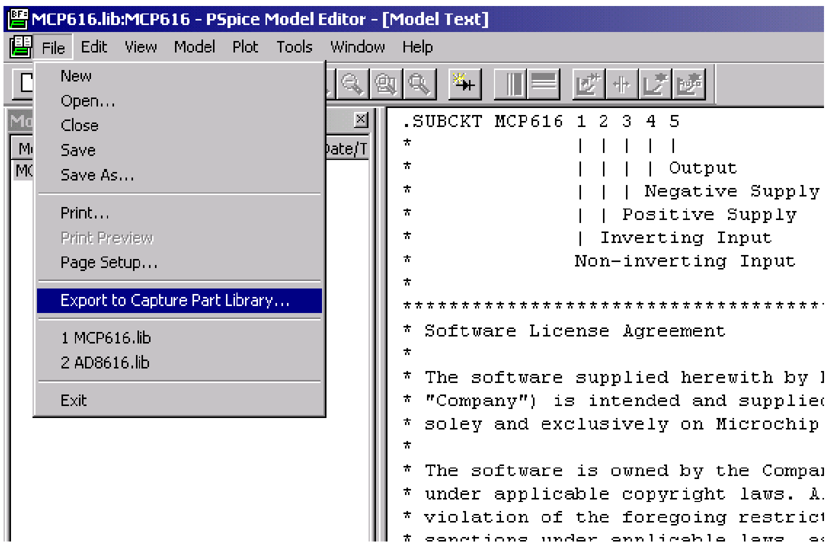
5. Save the imported file by selecting *File -> Save*. The file should have *.lib* extension.



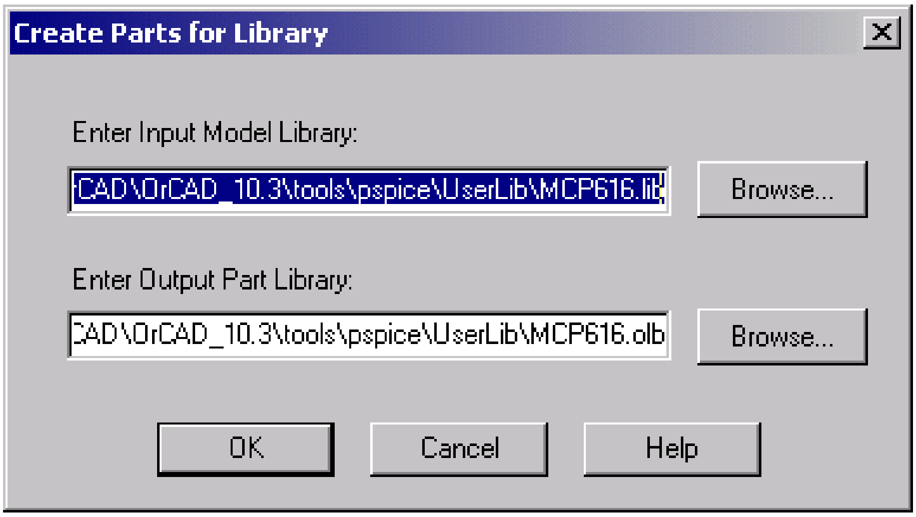
It is a good practice to use the name of a particular part as the file name. For this example use *MCP616.lib*.



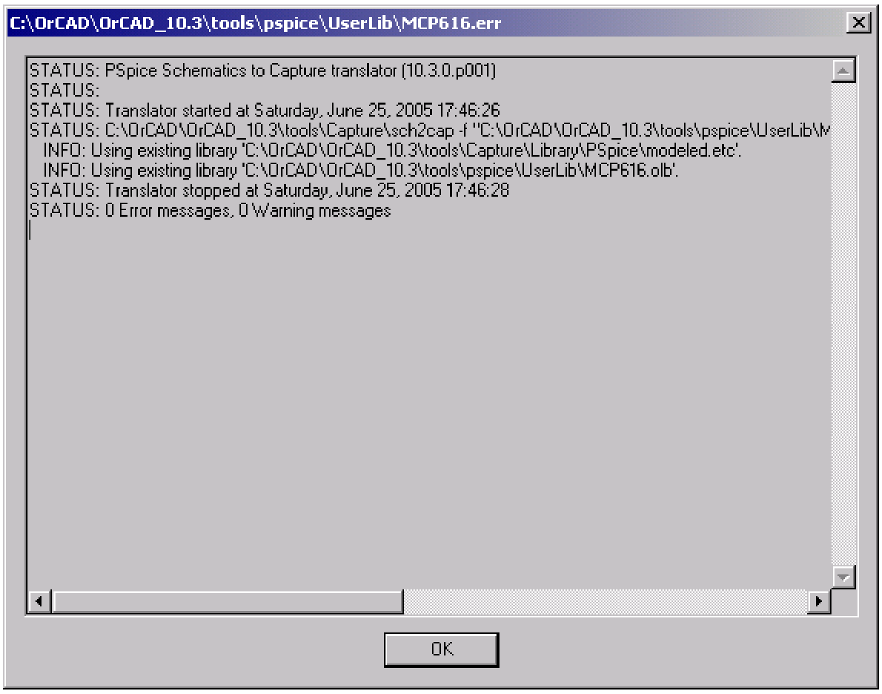
6. Select *File -> Export to Capture Part Library….* This step will create the PSpice model that can be edited and used for simulation in OrCAD PSpice.



The following dialog box will appear. Enter appropriate name and directory. Then click OK.

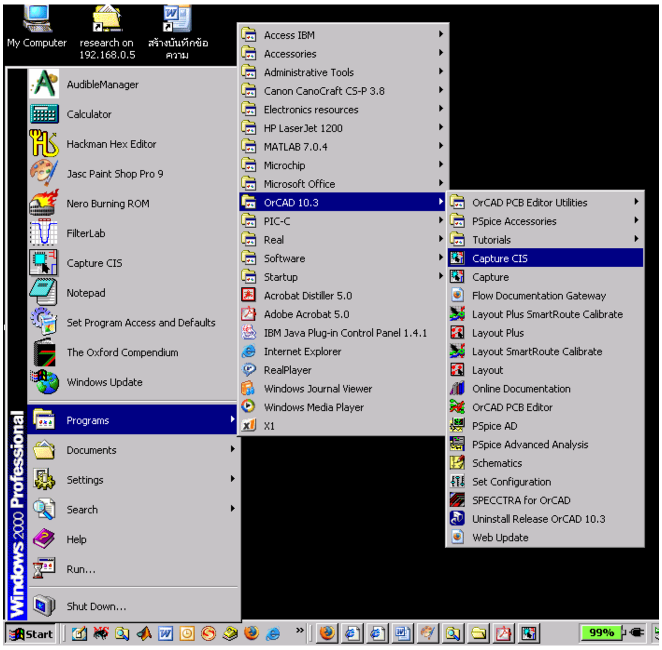


If the model is successfully created, the following dialog box appears. Select OK. The Model Editor can be closed at this point.

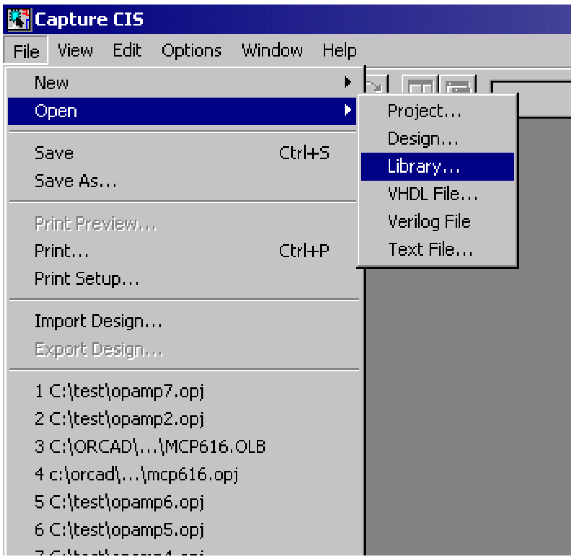


4 Editing the imported model

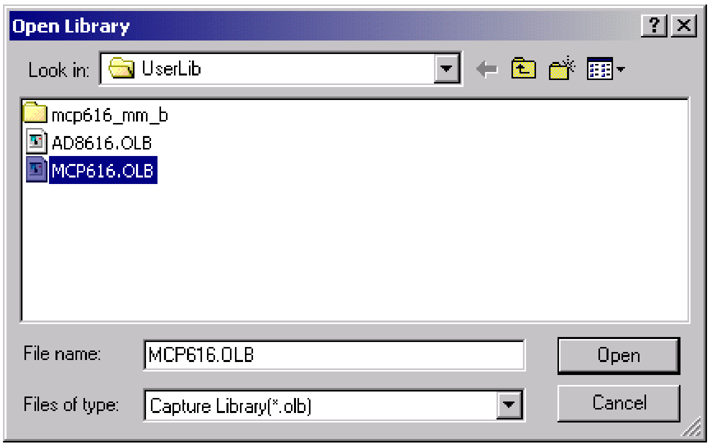
1. Open Capture program. Capture is one of OrCAD interface for designing schematics diagram.



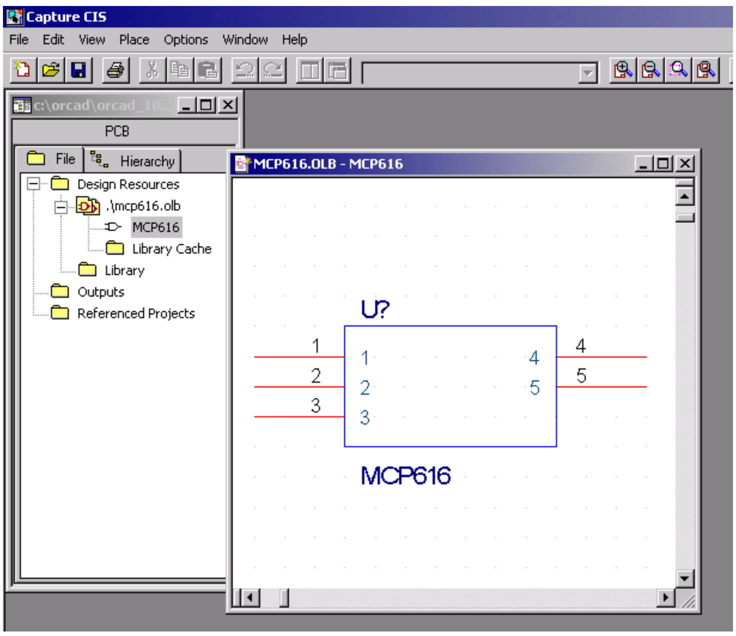
2. Select *File -> Open -> Library*



3. Open the library file MCP616.OLB saved earlier.

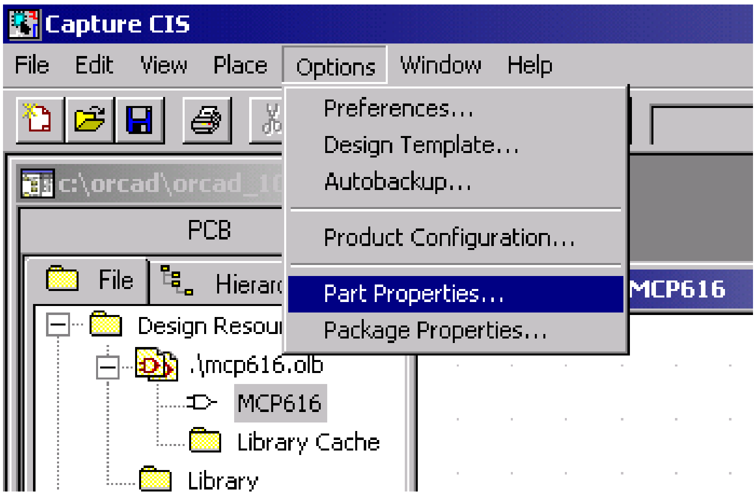


The following screen appears. Next we will change the current rectangular-shape of the model to the common triangular-shape used for OpAmp. If the graphical model does not appear as shown, it can be opened by double clicking the MCP616 icon to the left.
It should be noted from the following figure that there are two sets of numbers in black and in blue. The number in black will be denoted as physical pin assignment while the number in blue will be denoted as functional pin assignment. PSpice recognizes functional pin assignment which is used for simulation. The functional pin assignment should match the number listed in the model (text) file downloaded from the web. In our case they happen to be the same. For some downloaded models , it is possible for the physical and functional pin assignment to differ.

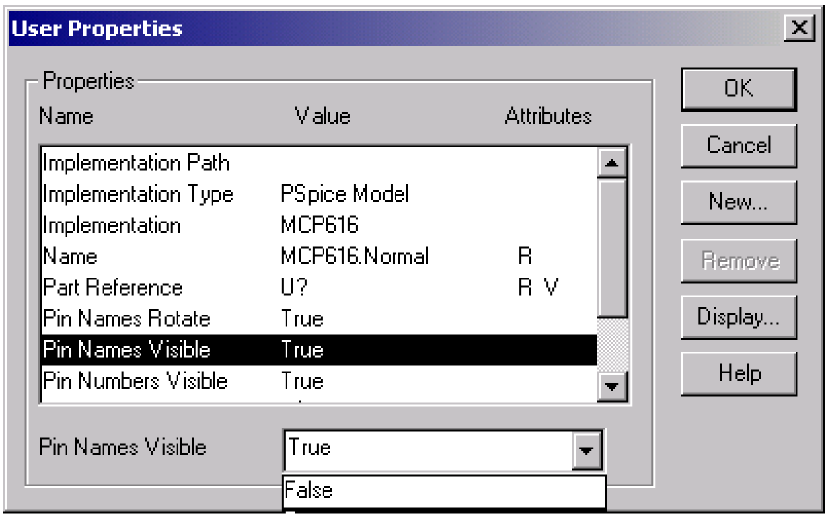


```
.SUBCKT MCP616 1 2 3 4 5
*                    | | | | |
*                    | | | | Output
*                    | | | Negative Supply
*                    | | Positive Supply
*                    | Inverting Input
*                    Non-inverting Input
*
****************************************
```

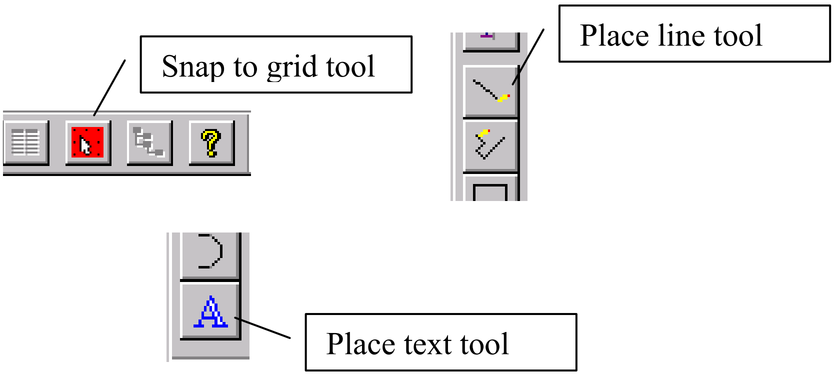
4. Turn off the functional pin assignment visibility to avoid confusion by selecting *Options -> Part Properties….*



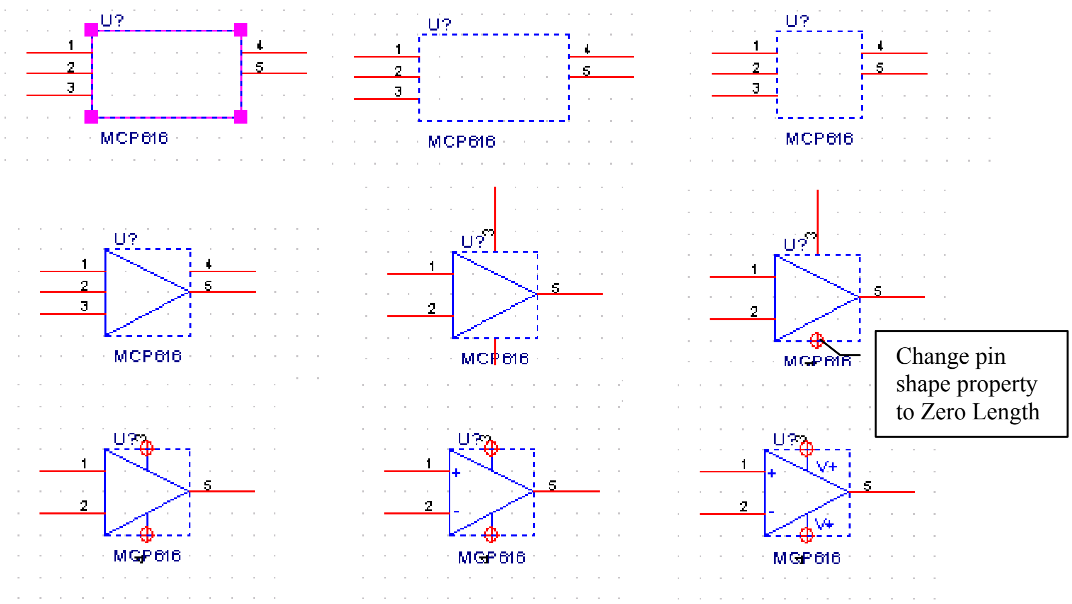
Change the "*Pin Names Visible*" property to *False*. However, before turning the visibility off, you should make note of the functional pin assignment position and number. These will be needed later when redrawing the part.



5. Next, we will redraw the model of the OpAmp. The process is outlined as follows. Delete the blue box with solid line (the dotted line will always remains), reshape the remaining dotted blue box to a square (normally 4 by 4 grid in size), draw a triangle using *Place line* tool, and finally, place text markings in appropriate places. The following tools are used when editing the drawing.



In order to move these pins to correct locations, the functional pin assignment is used not the physical pin assignment. The following drawings show steps when redrawing the OpAmp part.



6. After completing the redraw, the model can be saved by selecting *File -> Save*. It is now save to close Capture.

References

Bruce Carter, "Using Texas Instruments SPICE Models in PSPICE," Application Report SLOA070, September 2001, URL: http://www.ee.siue.edu/~gengel/pdf/modelCreator.pdf [cited 25 June 2005]

**** การที่ได้เขียนบทความนี้เป็นภาษาอังกฤษ เนื่องจากเอกสารความรู้สมัยใหม่ส่วนใหญ่ ไม่ว่าจะเป็นทางด้านวิชาการหรือเทคนิค เช่น Maintenance Manuals ต่าง ๆ หรือด้านการปฏิบัติการทางเรือ เช่น Maritime Maneuvering and Tactical Procedures – XTAC1000 ก็จะอยู่ในรูปของภาษาอังกฤษทั้งนั้น ดังนั้น ผู้เขียนจึงมีเจตนาที่จะกระตุ้นให้นักเรียนนายเรือเห็นความสำคัญของภาษาอังกฤษ โดยจะนำบทความนี้ไปใช้ประกอบการสอน เพื่อให้เกิดการพัฒนาขีดความสามารถทางภาษาอังกฤษของนักเรียนนายเรือ ****