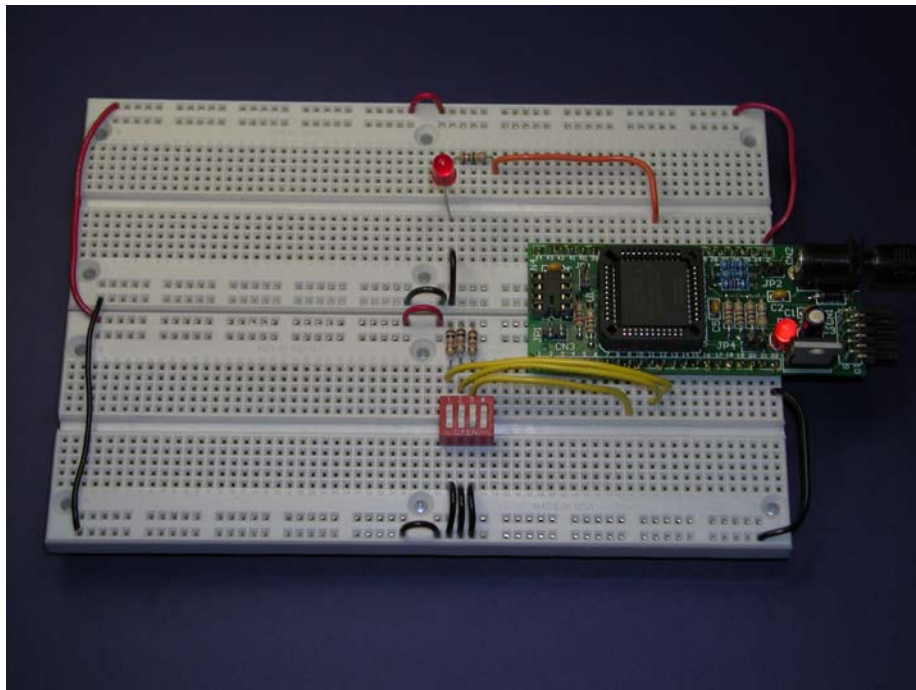


NM201: MAX44 - Altera FPGA/CPLD Development Board Kit

[MAX+PLUS II Baseline](#)

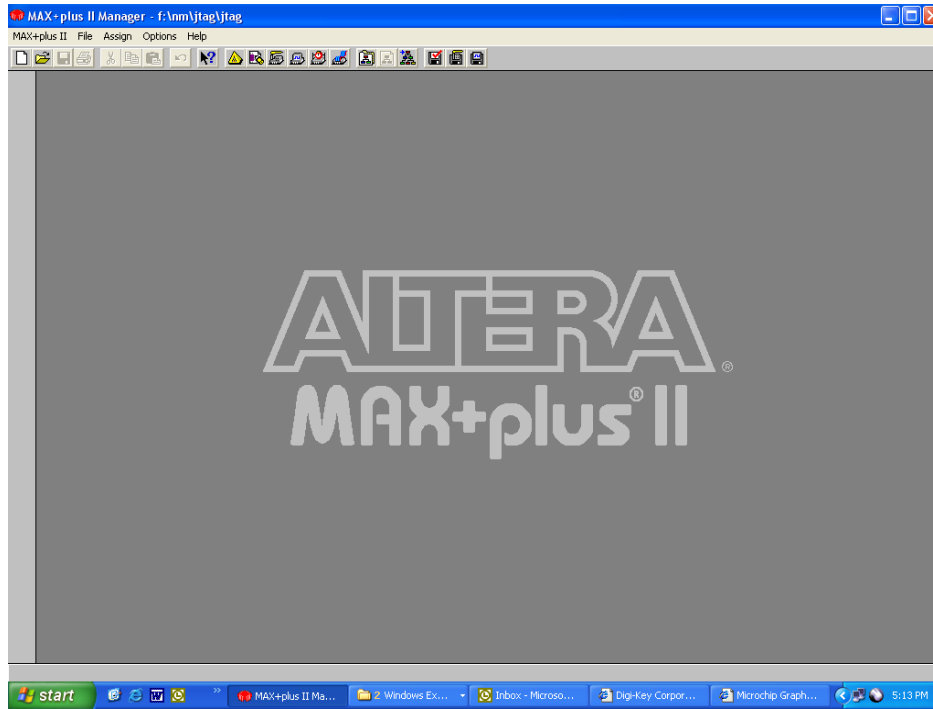
Tutorial



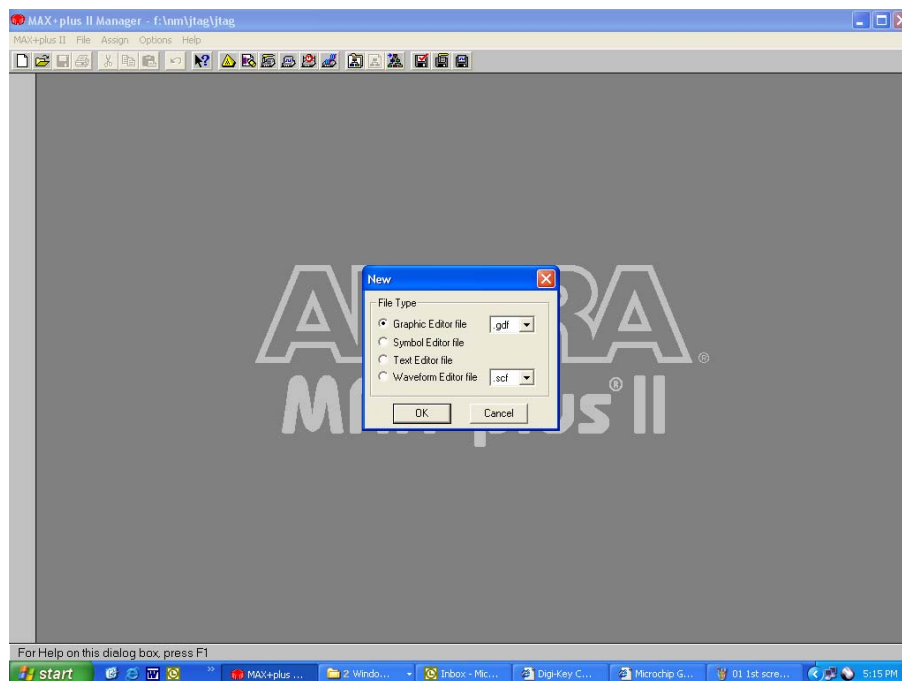
© 2002 NorthMicro, All Rights Reserved.

Protected by U.S. and International copyright laws as applicable.

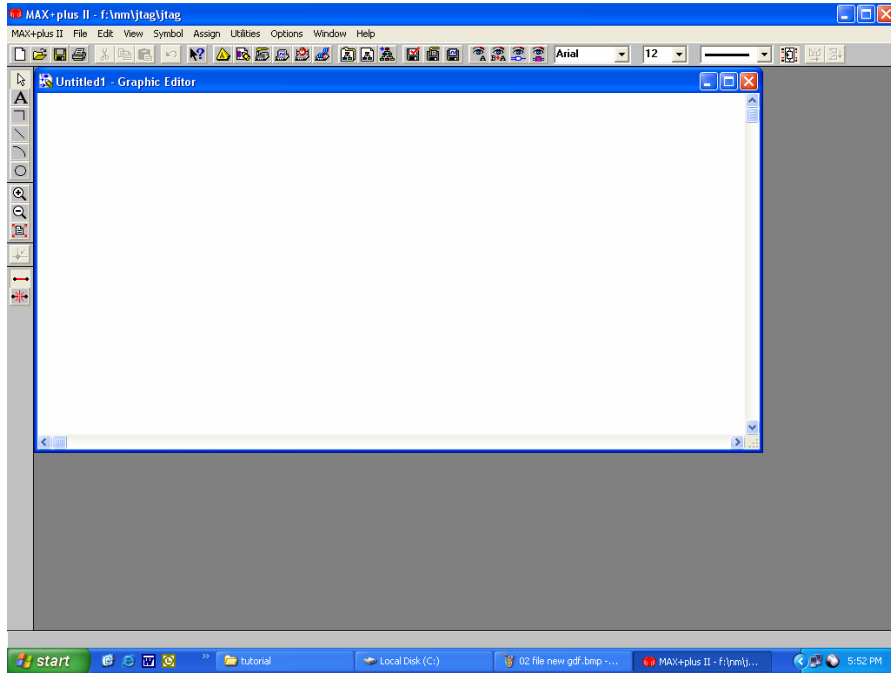
Install the Max+PlusII Baseline software from Altera. Follow the installation and licensing instructions on their site.
Double click on the Max+PlusII Baseline software icon. You should see the following screen.



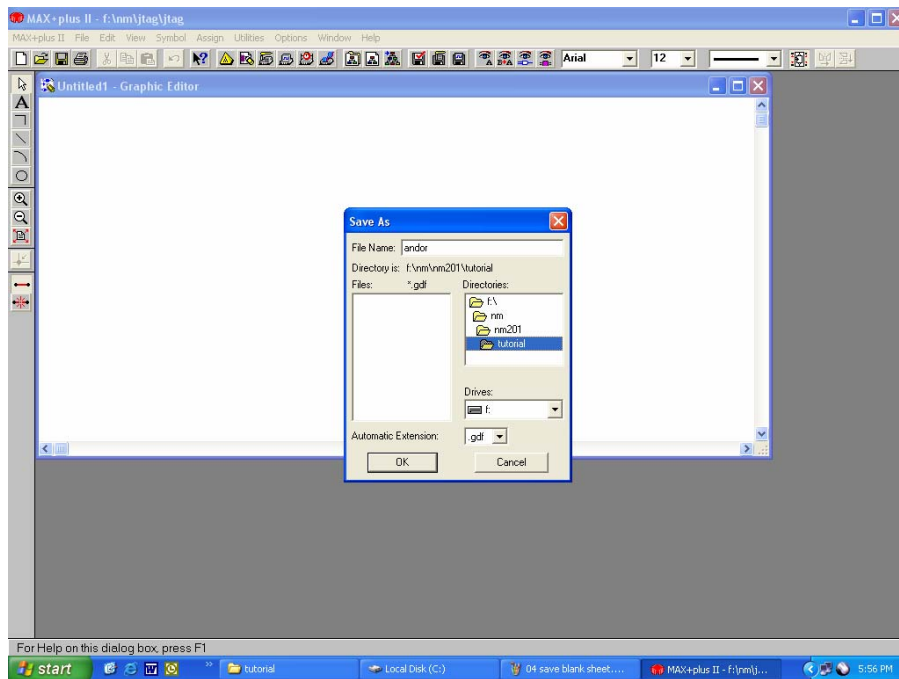
Click on File – New
Make sure the radio button for the Graphic Editor file is selected and then click OK.



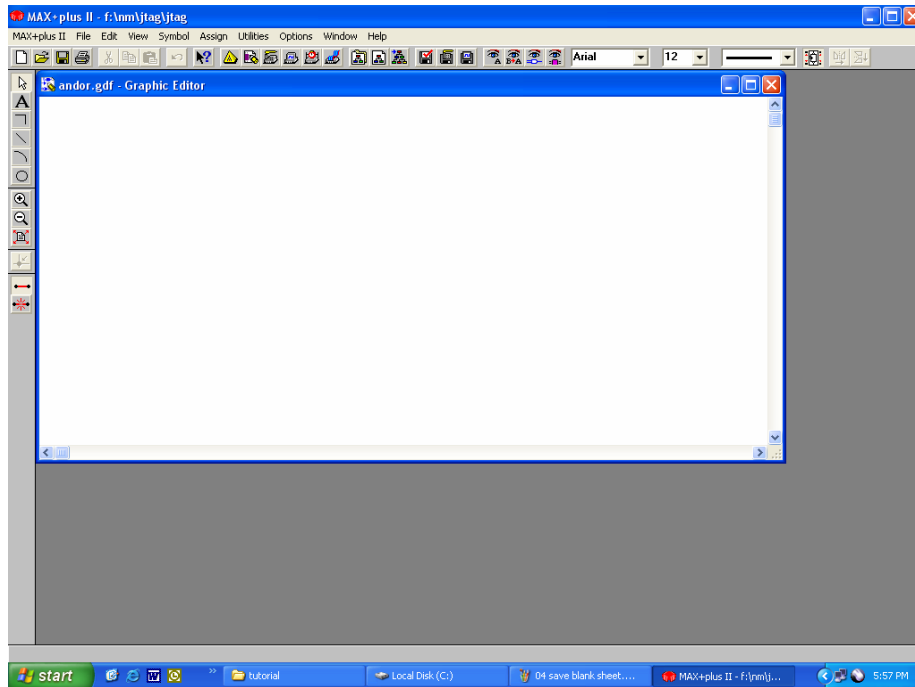
Click on File – Save As



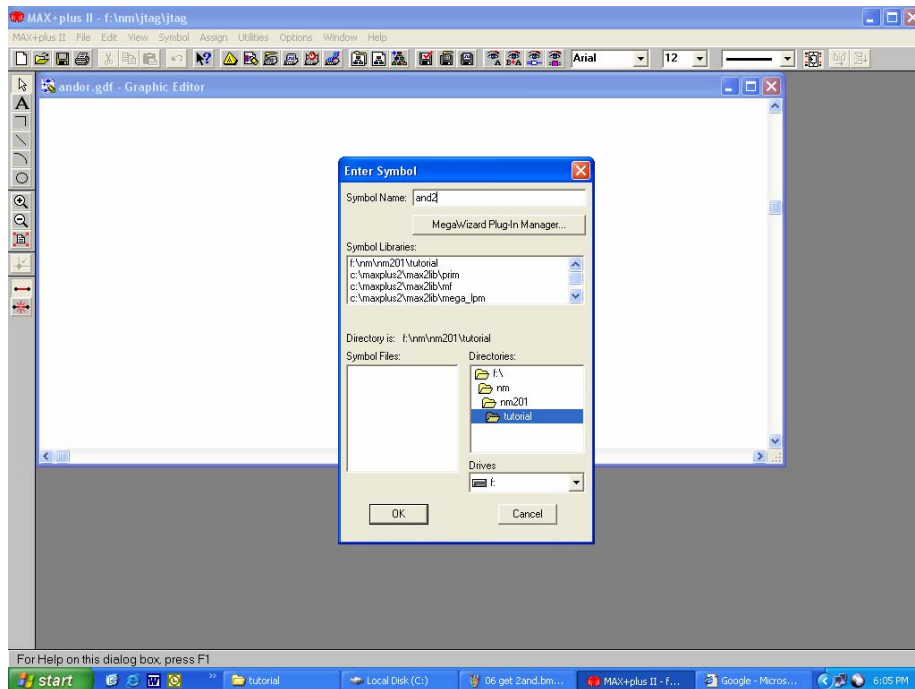
Select a directory to save this project in. Type in **andor** for the filename. Make sure the drop down box to the right of Automatic Extension has **gdf** selected. Click OK.



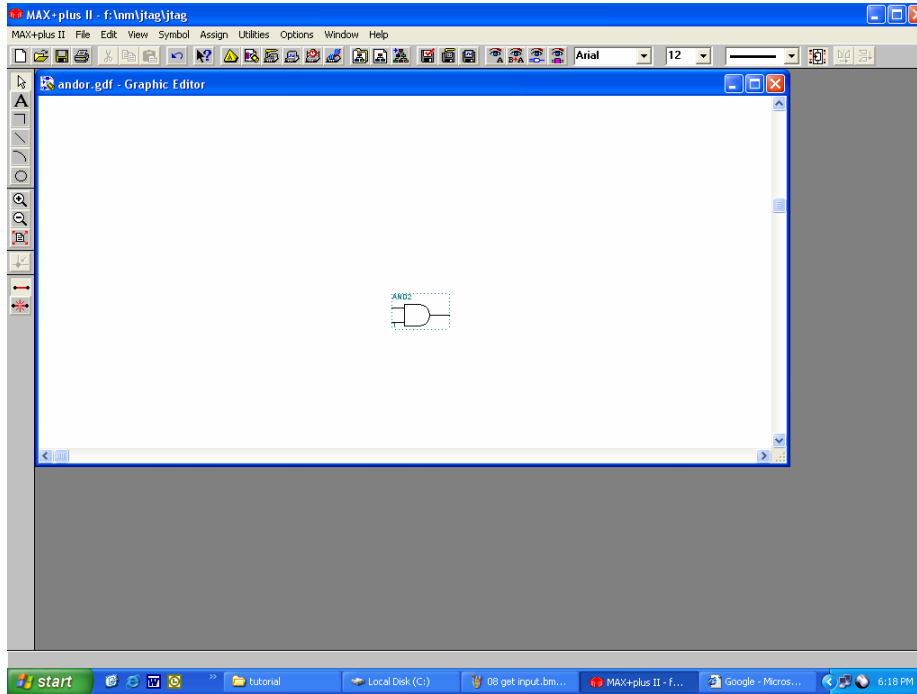
In the center of the blank Graphic Editor page double click the left mouse button. Alternatively you may Click on Symbol – Enter Symbol.



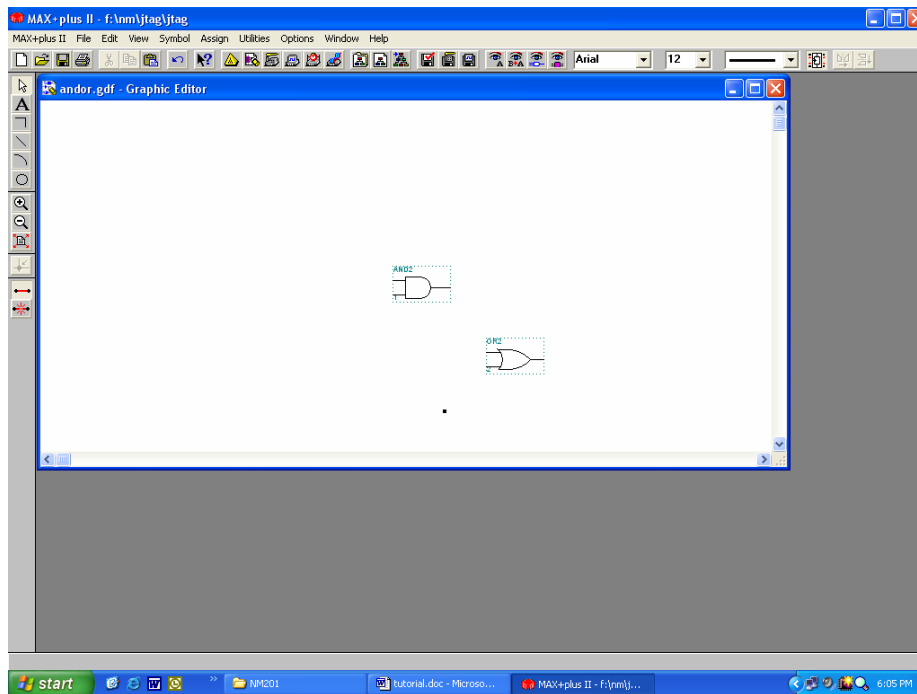
Type in **and2** for the Symbol Name and click OK. Alternatively you can browse for the part by clicking on the prim (primitive) directory, and selecting the part you would like. Other parts are also available under the mf and mega_lpm directories.



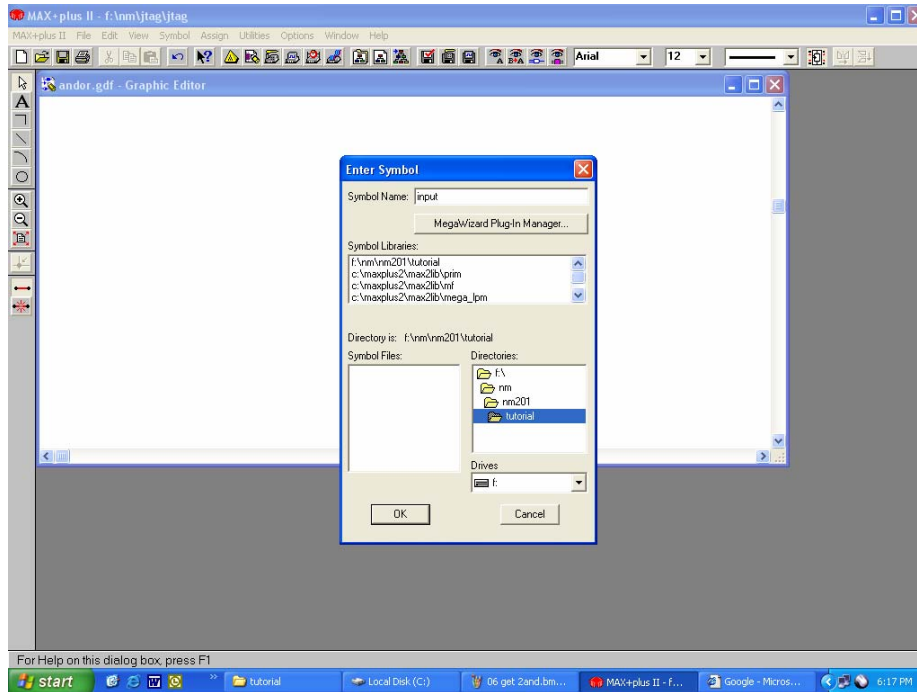
Double click a little below and to the right of the and gate and add a 2 input or gate (Type in or2 when it asks for the symbol name and click OK).



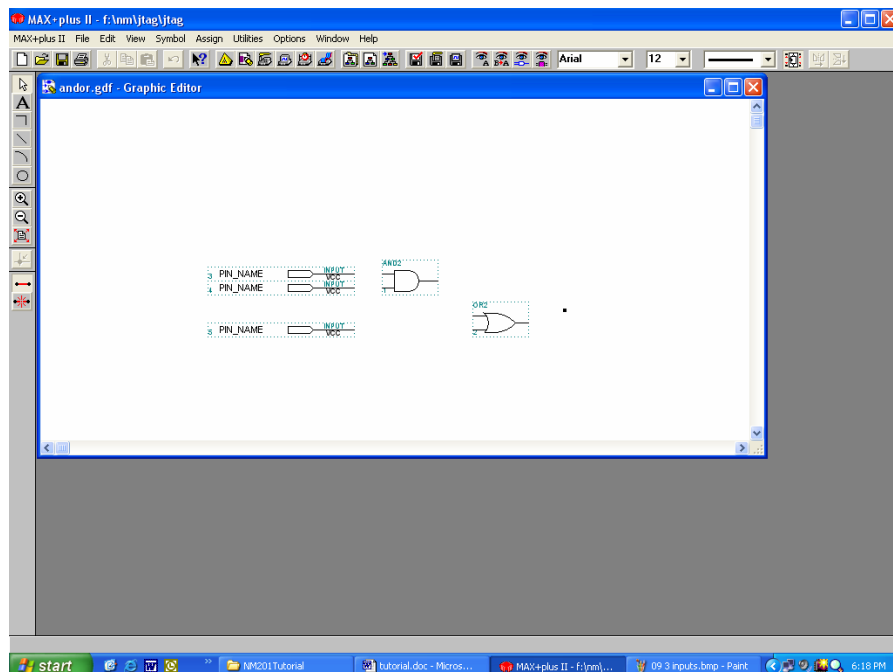
If the or gate is not positioned properly, you may left click on the symbol and while holding the button down, re-position it.



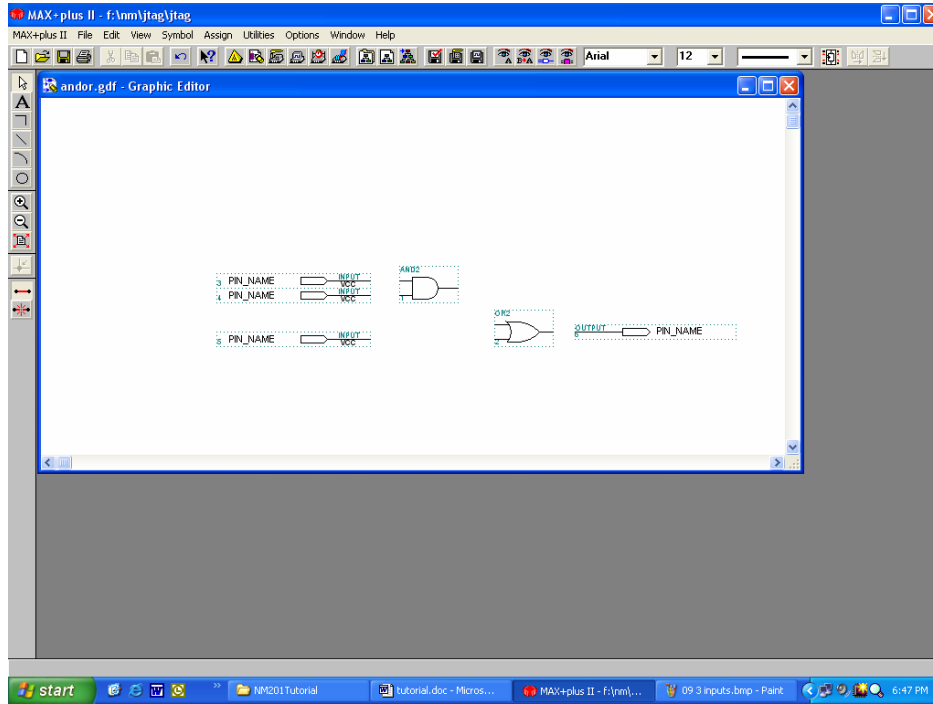
Double click somewhere to the left of the and gate. Type **input** for the symbol name. Repeat this two more times.



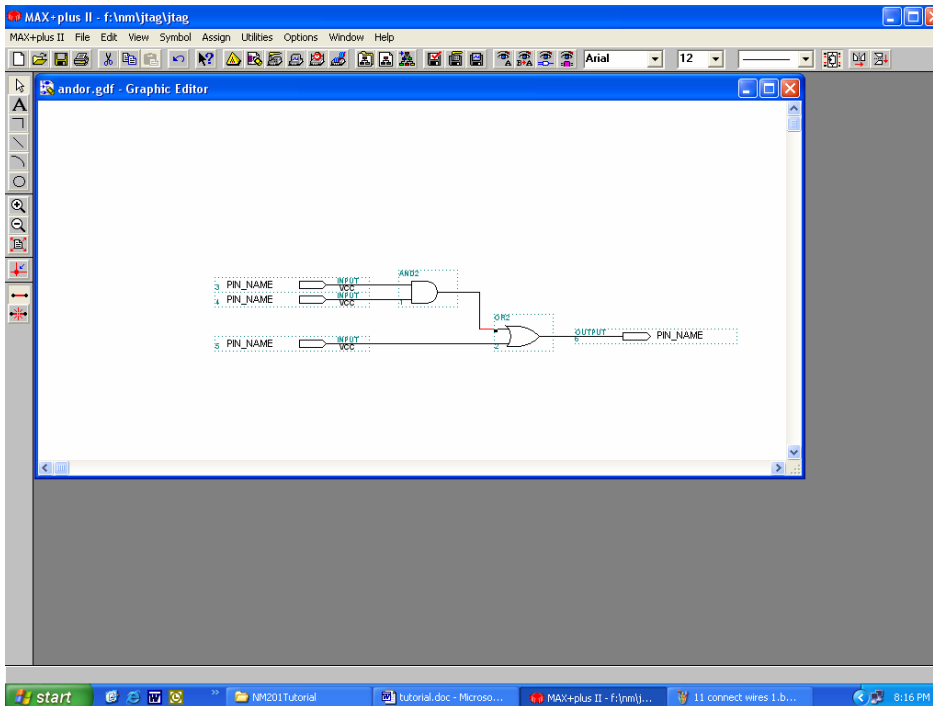
Double click to the right of the or gate. Type **output** for the symbol name.



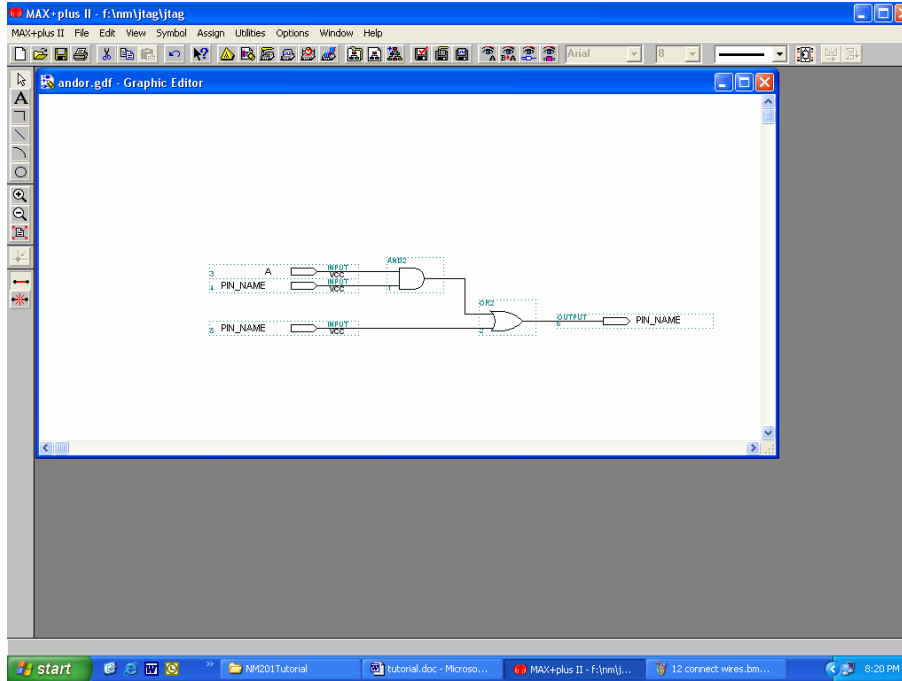
Connect the parts together by left clicking on the line attached to the top input symbol and while holding down the button, draw a line to the top input of the and gate.



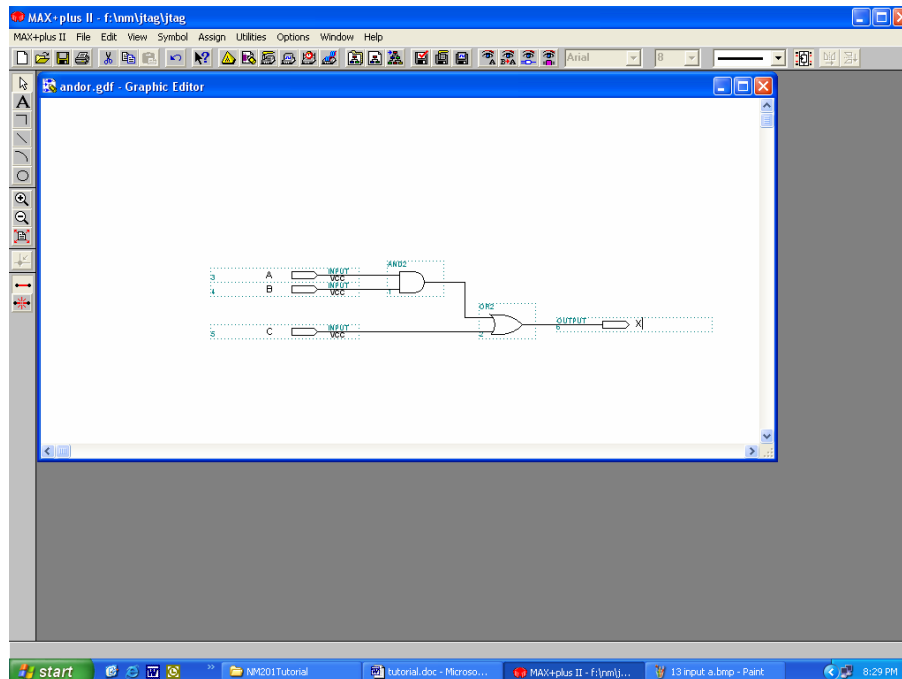
Connect the remaining lines as shown below. If you need to change the direction of a line, unclick it and click the line again, moving in the new direction.



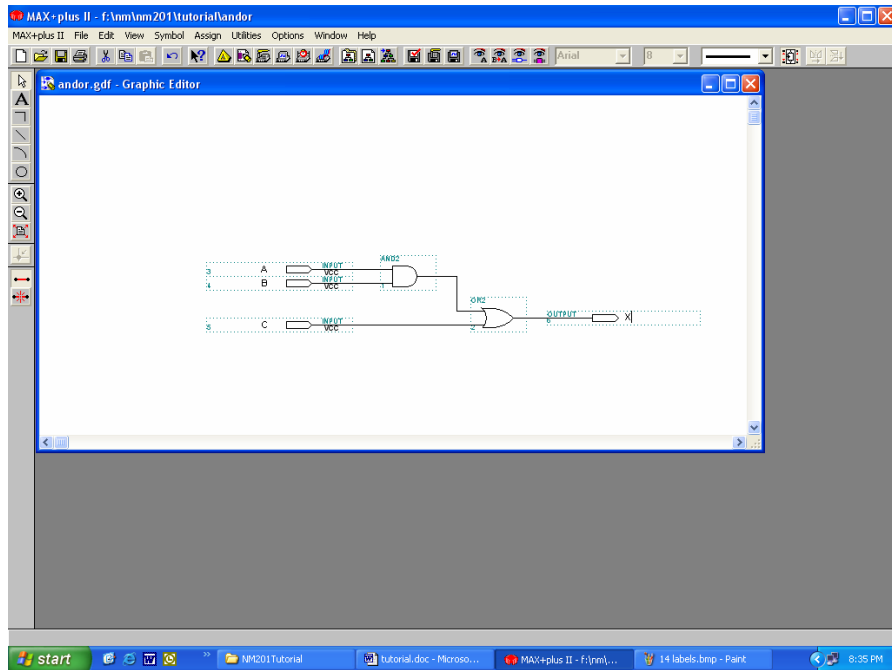
Double click on the top input PIN_NAME label and type in A.



Change the remaining PIN_NAME labels so they match the screen below.



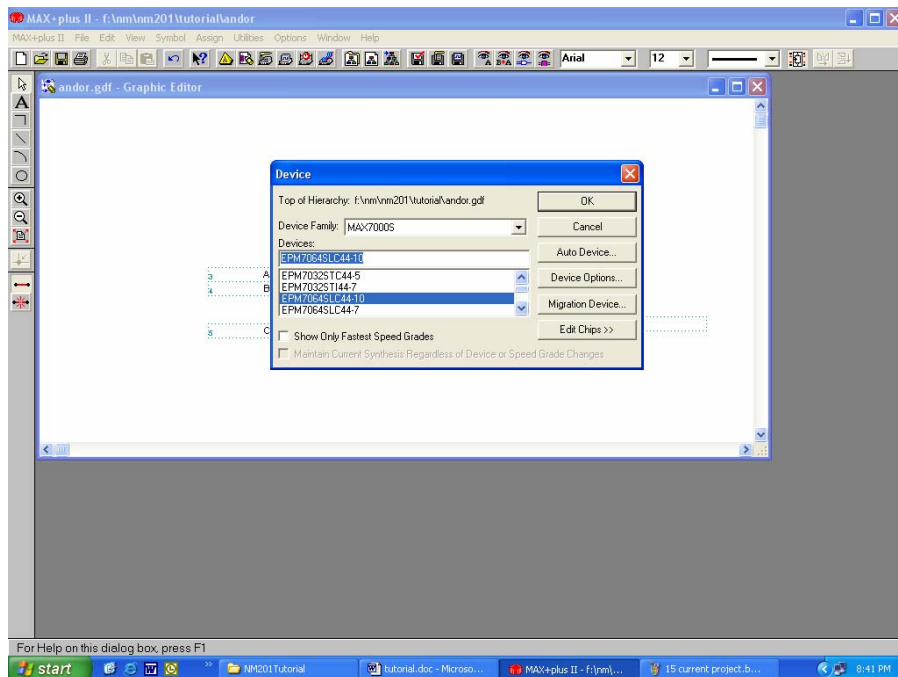
Click on File – Project – Set Project to Current File. Notice the top bar now has the name of the current file **andor**.



Click on Assign – Device.

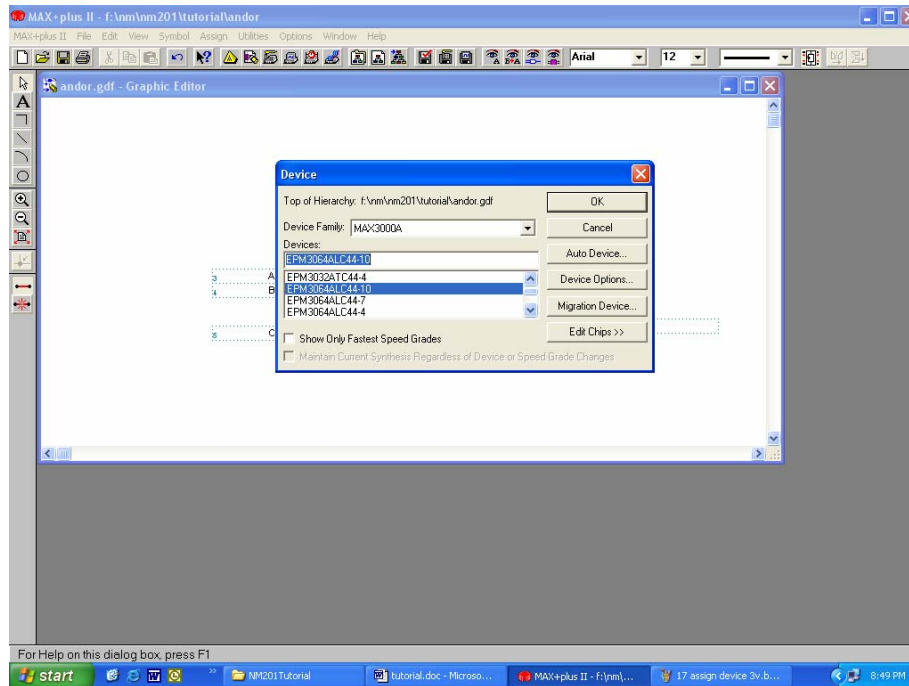
For the NM201K5

Under Device Family select MAX7000S. Uncheck ‘Show Only Fastest Speed Grades’ and select EPM7064SLC44-10. Click OK.

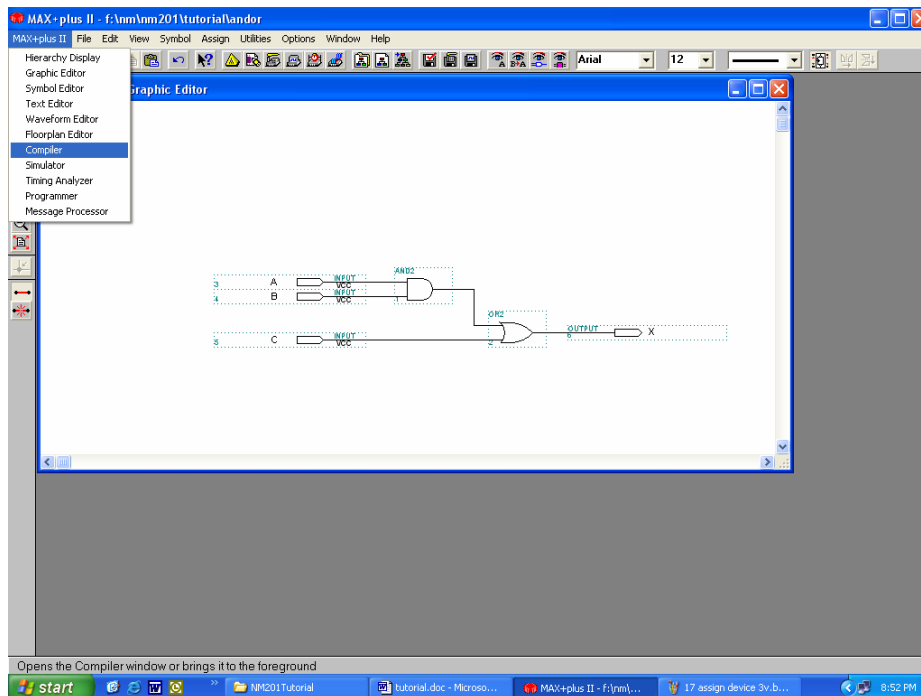


For the NM201K3

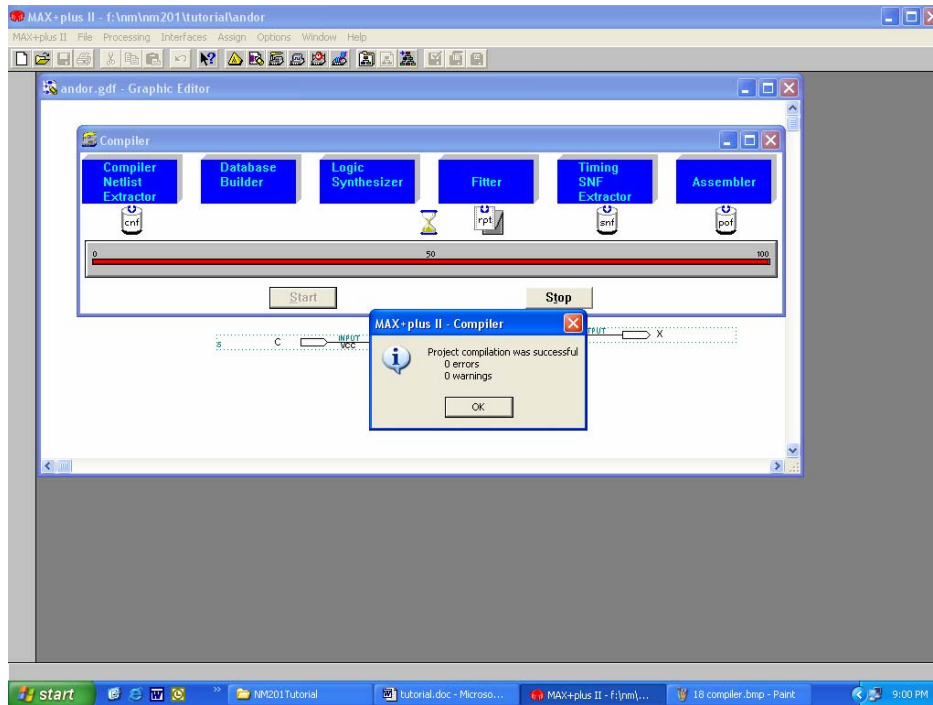
Under Device Family select MAX3000A. Uncheck 'Show Only Fastest Speed Grades' and select EPM3064ALC44-10. Click OK.



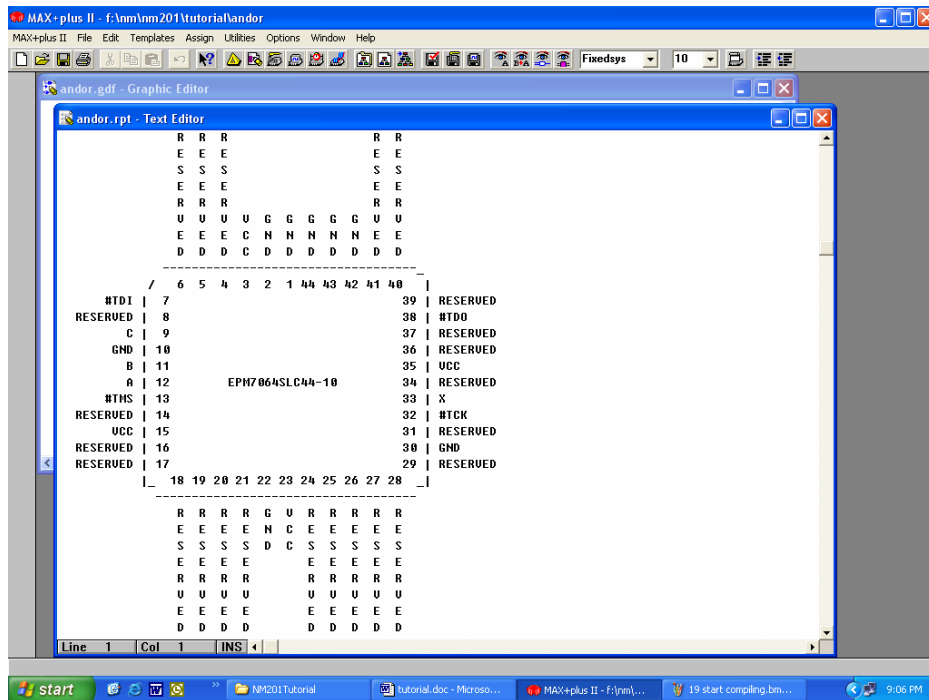
Click on the MAX+PLUSII menu to the left of the File menu. Click on compiler.



Click the start button. After the compiling has finished, a dialog box will pop up and tell you the results. Click OK. Under the Fitter box in the compiler window, there is an **rpt** icon. Double click this icon.



Scroll down through the andor.rpt file. The report has a picture of the CPLD chip with the new PIN_NAME labels A,B,C and X.



RESERVED pins are unused I/O pins in this design.

#TDI, #TDO, #TMS, #TCK pins are used by the NM201 development board when programming.

VCC pins are all tied together on the NM201 development board. Power is supplied via the regulator on board the NM201 development board.

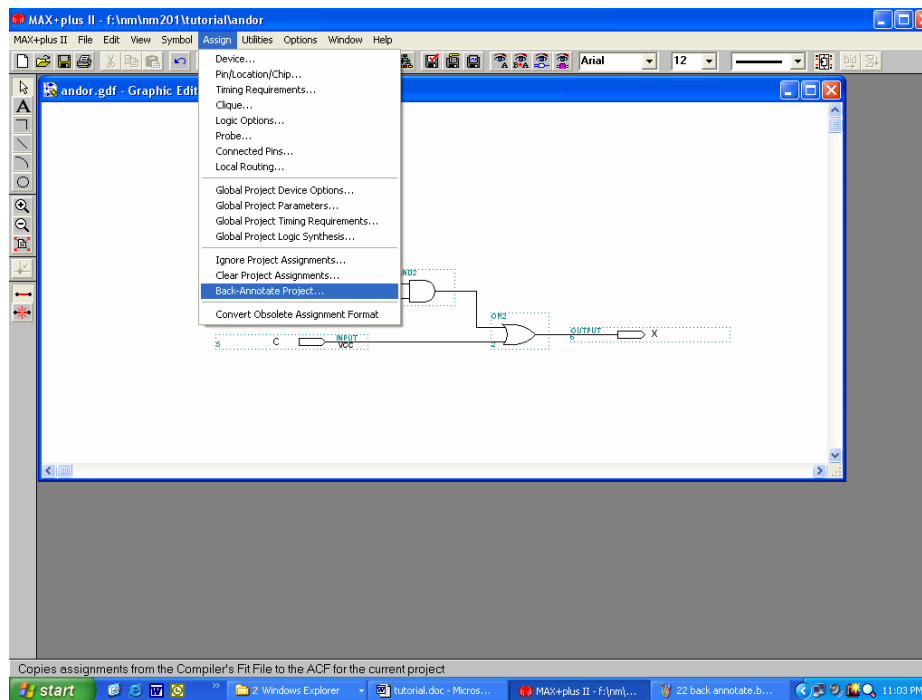
GND pins are all tied together on the NM201 development board, **except for the dedicated input pins.**

Dedicated Input pins 2, 1, 44, and 43 need to be tied to GND if unused.

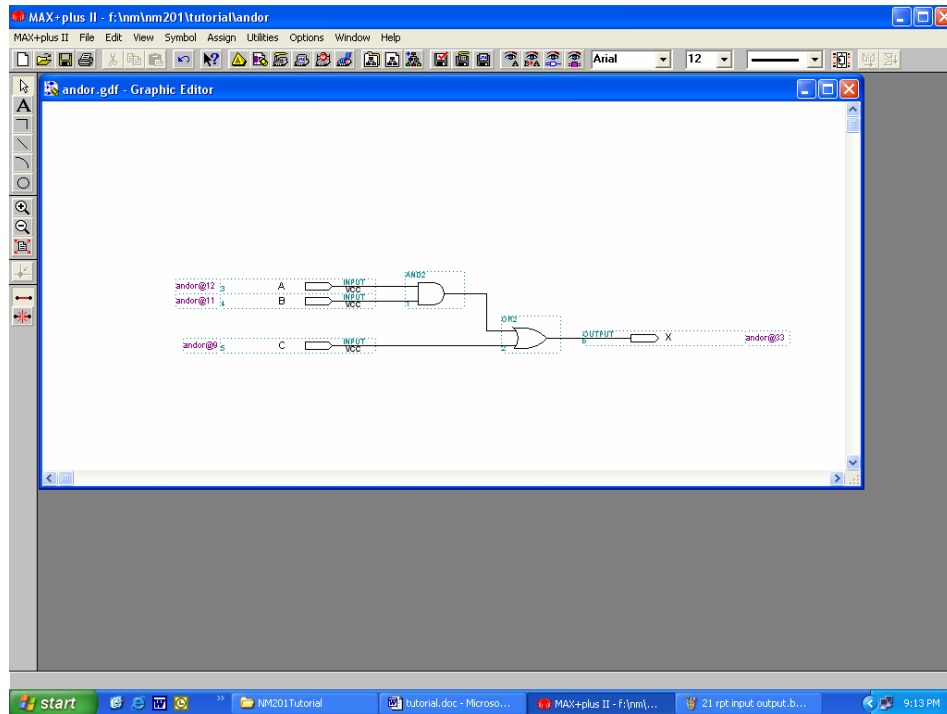
A, B, C pins are the inputs for this design.

X pin is the output pin for this design.

Close the andor report and click on the andor.gdf window. Click Assign – Back-Annotate Project.

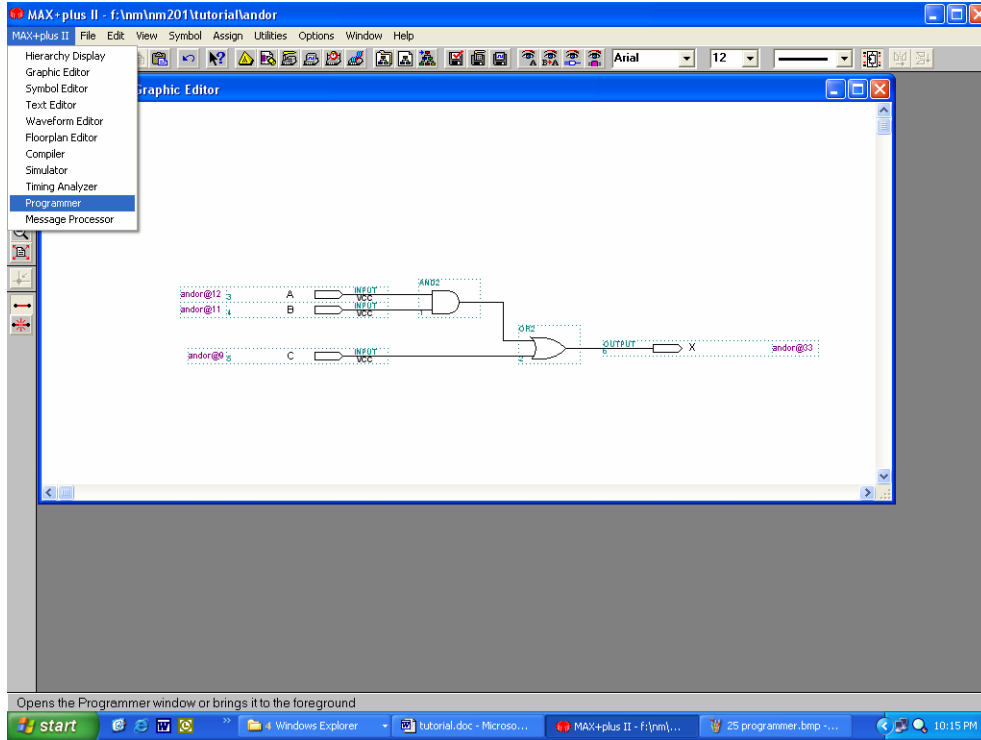


Back-Annotate assigns the pin numbers to the designs inputs and output on the graphical design.

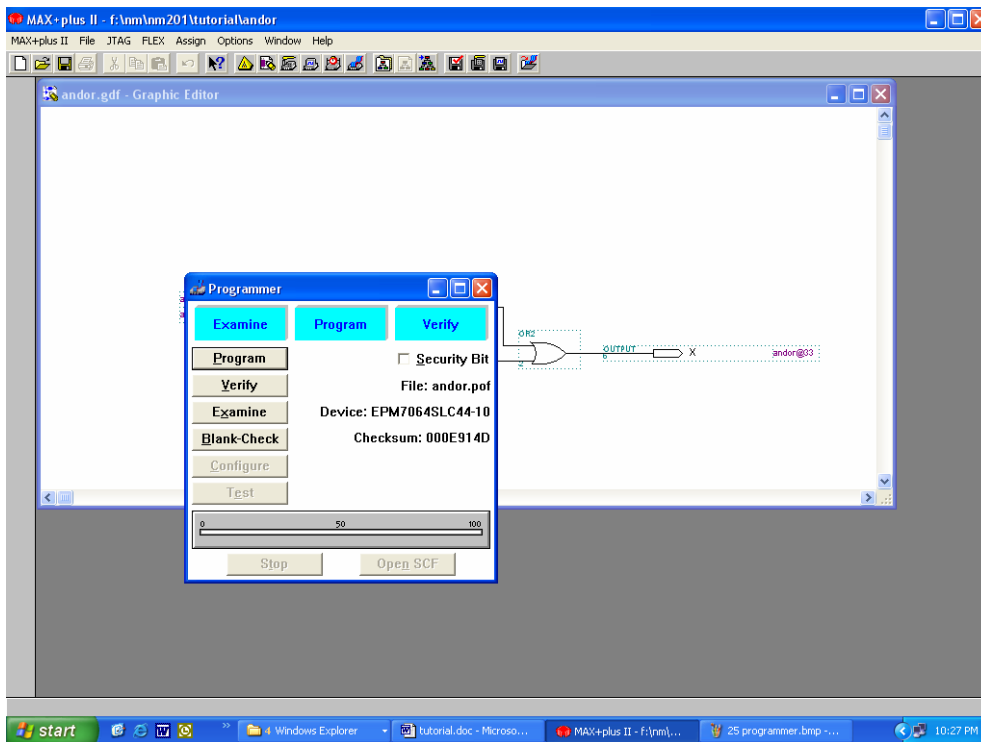


Connect the NM200 programmer to the NM201 development board. Make sure the red stripe of the cable is attached to pin 1 of each board. Connect the NM200 programmer to the parallel port of your computer. Plug the NM201 development board into a 9volt center positive supply of at least 500mA.

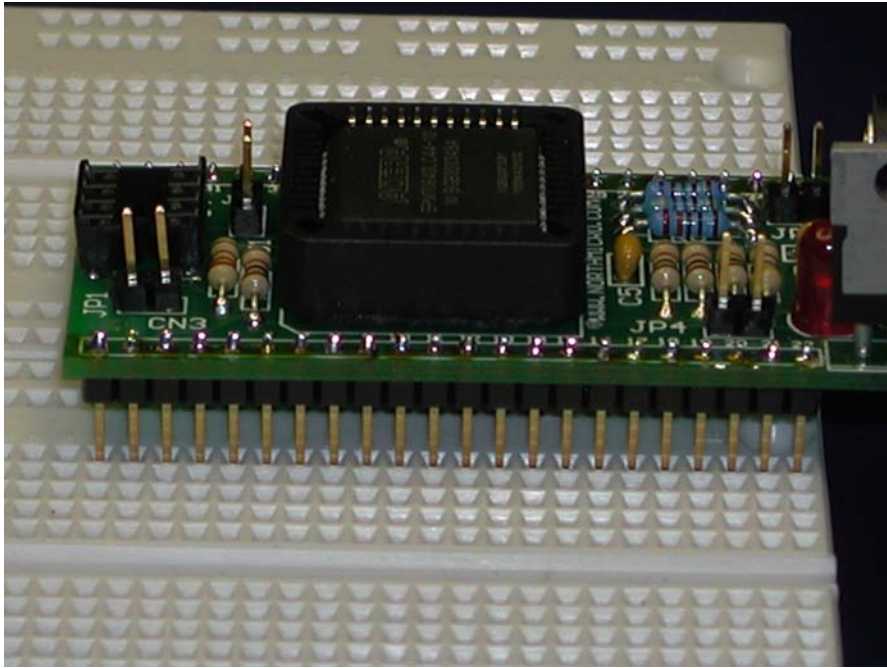
Click on the MAX+PLUSII menu to the left of the File menu. Click on Programmer.



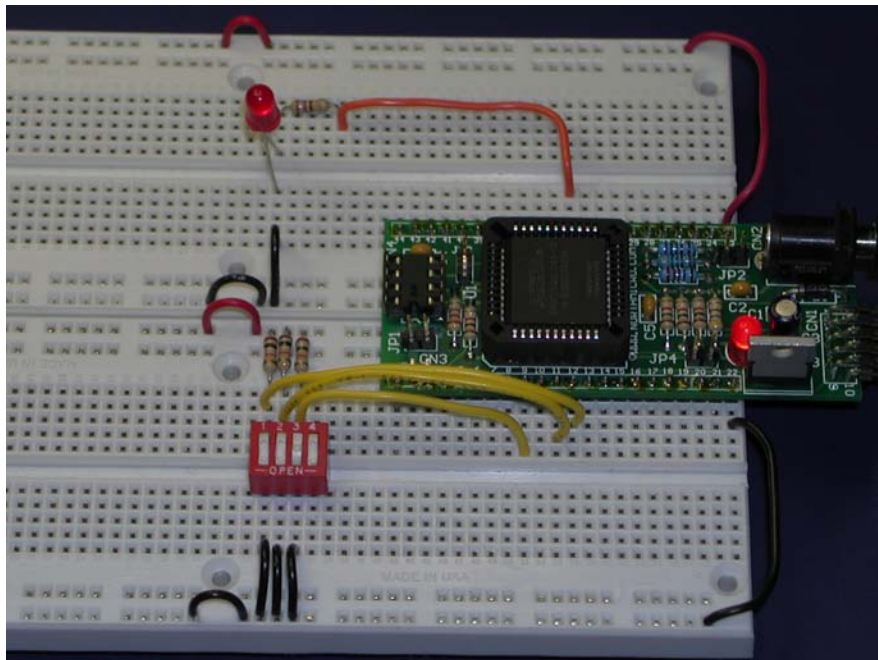
Click on program.



When programming is complete, unplug the development board from the power and programmer. Plug the development board into 2 breadboards. Be careful to align the holes and starting at one side, firmly press the development board into the holes.



Wire 3 bits of a 4 bit dip switch as the inputs to the development board. Wire an active high LED to the output. The NM201 Development Board is providing 5volts (or 3.3volts for the NM201K3) and ground to the breadboard via pin 23 (5volts) and pin 22 (GND).



Congratulations! Your design is complete.