An Engineering Manager’s First Experience with Multisim
by Anonymous, Analog Devices, Inc.

IN THIS MINI TUTORIAL
A first-person account of an Analog Devices, Inc., manager’s initial experience with the NI Multisim™ Component Evaluator–Analog Devices™ Edition simulation tool to design a simple op amp.

INTRODUCTION
As an engineering manager with several decades of experience in analog circuits, I am embarrassed to admit that until now I had never actually used SPICE or any other simulation software. I recently had the Analog Devices edition of Multisim installed on my PC, and now I am going to see if I can get a simple op amp circuit to work. Therefore, with my office door firmly locked, I open Multisim and begin my quest into the world of analog simulation.

GETTING STARTED
Of course, Multisim offers plenty of tutorials under the Help menu; however, like most engineers, I decide to bypass that step and dive right into the program. How difficult could it be? I open the program and begin by looking at the menus. The Place menu item (see Figure 1) seems like a good starting point.

After clicking Place, I choose Component (see Figure 2); now, I can choose an op amp. I pick the OP282GP.
Later, I learn that there are multiple ways to place a component. Right-clicking anywhere on the grid displays a window that easily allows you to Select a Component as shown in Figure 2.

This is a good time to think about saving. I save my file, naming my circuit AMP Block1.

**CONNECTING THE POWER SUPPLIES**

As I look at the part displayed on the screen, I decide to connect the power supplies. Though I do not see a menu that looks like it provides power supplies, I learn later that this can be accomplished via the Component menu.

When I right-click on the menu bar, a drop-down box supplies choices to select power source components. Unsure what to do, I move the cursor over the possible supplies. One is Digital Ground and another is Ground. I know I will need an analog ground, so I choose Ground. I place it on the drawing.

Next, I choose VCC and click on Pin 8 of the amplifier; it automatically connects. VEE should work the same. It does, but it looks funny (see Figure 5) on top of the Pin 4 line. I drag it lower and it looks better (see Figure 6). I have + and − 5 V on my supplies. The part will work with these values, so I leave them alone for now.
ADDING RESISTORS

My interest is amplifiers, and an inverting gain amplifier is about as simple as it gets; I am going to put in a few resistors. Again, I select the Place menu that I used for the amplifier. However, I notice at the left of the screen a selection of various components. From Family, I choose RESISTOR (see Figure 7) with a value of 10 kΩ, and then click OK and place the resistor connecting it to Pin 2. The menu automatically pops up, and I place another 10 kΩ to the same node. Again, the menu pops up. I need one more resistor for the noninverting input and choose a 5 kΩ resistor, even though this is a JFET op amp, and I may have done better to leave it off. I can check later to see if this increased the noise and was or was not a good decision.

![Figure 7. Selecting Resistor Values](image7)

This seems like a good time to connect the 5 kΩ resistor to ground. I click on it and drag to the ground symbol. It connects. If I want to rotate it, I can do a Control-R. The 10 kΩ feedback resistor is sitting on top of the amplifier, so I drag it lower. Now I find out it is not connected to the input; it does not connect. After looking at the menus, I decide to select Place and then Junction. After some playing around, I am finally able to connect three components to the same node (see Figure 8).

![Figure 8. All Resistors Placed, but Some Connections Missing](image8)

At this point, I think I am getting close. I am going to add another junction so that I can have an output.

FINISHING UP

After that, I'm almost done. I do not know how to connect inputs and output or how to do anything. I decide to toggle the Simulation switch. I get a note at the bottom of the window (see Figure 9).

![Figure 9. No Errors Present in the Circuit](image9)

Not too impressive, but not too bad. There are no errors. I suppose I need an input and something to measure the output. After looking through the menus, I select Place > Component > Sources, and chose SIGNAL_VOLTAGE_SOURCES. Then, I chose AC_VOLTAGEW. I connected this to the 10 kΩ input resistor.

Next, I need a way to view the output. I find an oscilloscope by selection Simulate > Instruments. I chose the oscilloscope and connect one channel to the output of the AC_VOLTAGEW and the other to the output of the amplifier. I ground one oscilloscope channel. Later, I find that this is not necessary, in most instances. I think I am ready to try this now (see Figure 10).
When I click the Simulate button in the upper right nothing happens.

Then, I double-click the oscilloscope and —surprise—this thing works (see Figure 11)!

With a quick change of scales, I get the results shown in Figure 12.

I have completed my first simulation. Although it is only a simple inverting amplifier, I am ready to try more. I could easily make a filter. On the other hand, maybe I will try to see if the noise is low enough for my application. I can at least unlock my office door now.

**REVISION HISTORY**
1/12—Revision 0: Initial Version