Using Multisim Online

Video Transcript

Hello, I'm Lois Gray from North Highland College UHI and in this video I'm going to show you how to use Multisim Online to draw your circuits and simulate them. So this is for anyone who's had problems in downloading Multisim - you can use an online simulator instead. So I'll show you where that is first of all. The easiest way to find it it's probably just to do a Google search so we'll just type in Multisim Online and then it's this online simulator so if we click that, your simulator opens. Now you will need to create an account with National Instruments and you'll need to login; use your student credentials to do this and if you have problems with that please contact the Unidesk as it's not something I can really help with anyway.

So it's really easy to use you just click create. So okay, you can see I'm already logged in here; click

create circuit (I'm just going to try and move this video thing out of the way so it's not on the way of the menus there) and then you'll see on the left hand side here we have our components which you can click and drag onto the blank canvas. You don't have quite the same range of options as you do with

Multisim downloads, but certainly enough for your course works.

So we're going to build an inverting op-amp today so I'll show you how to do that. So here's our op-amp circuit. You can see when you hover over you get a little tab telling you what these components are, so if we click and drag that you can also open the menu, as you can with Multisim normal. I think if you click that you can see the options that you have which is very similar to your virtual components

in the downloaded version of Multisim. OK, so we're just going to use virtual components. As you know these are components that are perfect; they don't have manufacturing tolerances, they don't have temperature drift, gain variations, they're just perfect components. So there's our inverting op-amp by the way. For your courseworks you can use virtual components as well - that's fine you don't need to use actual or simulations of real components.

So we want some resistors for our inverting amplifier, so all I'm doing is left clicking and dragging, putting them on. Now when you left click and drag you'll see that there's these four menu options here: you can delete, you can copy the component (that might be quicker than trying to find it in the menu if it's a more complicated component like a 555 timer or something like that), you can rotate the component and you can also flip it just using this button here. So this one this is going to be our bias resistor. If you remember, the bias resistor helps to reduce the effect of input current bias by ensuring both the input terminals see a similar impedance. So we're going to just rotate that and that's going to be connected to our non-inverting terminal to balance the parallel combination of R1 and R2. We're going to use a sine wave source to simulate the circuit, so if I just click in, oh actually yeah, click and drag is fine. I could have picked a voltage source but I think it comes up automatically. As well, anyway, if you want a current source you'll have to open the menu and then find your source from there, but the voltage source is the one that comes up automatically.

Yeah so we want to use a voltage source so we can see both amplitude and phase changes at our op amp output. With a perfect inverting op-amp we would expect to see the same output: the same amplitude of output because we have a gain of 1 here. You'll remember the inverting op-amp gain is negative R2 over R1 so 1, 1k over 1k gives us 1, but we would expect to see a 180 degrees phase shift because it's an inverting op-amp. If it was a non-inverting op-amp we would see no phase shift.

Ok I'm just going to move that component over to the side – all those components to the side a little

and then we'll just connect them up (by the way you can zoom in and out with your mouse scroll button). We’ll zoom in a bit now to connect these circuits up; you just left hover over your contact and

you should get a cotton reel sign and you just click drag and then left click again to connect. So nice and easy although sometimes it's a little tricky to get the cotton reel, I found, but it does work. So let's just connect these up. This is our gain resistor - gain setting resistor - so that will go there. The other end of that will go to the output bias resistor - will go to the positive terminal. Okay, this is the wrong value so to change the value all you need to do is click on that. Double-click on it, rather, and a pane will open up on the right hand side here and we can just type in the new value. You can add in here some extra information: you can add temperature effects, you change your symbol and so on. I'm not gonna bother doing that, I'm just going to show you the basics; you can play with doing these sort of things yourself, and then we can just hide that pane with the arrow up at the top here and you can see that to change the value, so that's now the parallel combination of R1 and R2.

As with Multisim downloaded, you do need an earth point so I'm just going to grab an earth point from the menu and then connect that, so we'll connect that to the bottom of our bias resistor and we'll also connect it to the negative terminal of our source (although with it being AC it's not really a negative terminal). So that's all ready to go. You can edit the source if you wish. I don't know - do we want to edit that? You have all the options for editing - they come up in the right hand plane. I don't know, let's change the frequency - let's say 500 Hertz just to show you how to edit this. So that's fine and then we'll just close that pane again. So to see the signals what you have to do is put these analysis points on. So it comes up automatically as a voltage but you do have the option of current, digital outputs, and so on. Anyway we're just going to use voltage, so what you do click and drag there and then hover over your point that you want to test, and just let let go of the left mouse button at that point and you'll see here we have a green symbol so that will show up as a green trace in the scope. You will also see the values change in this little box here as you simulate. I'll show you that when we actually simulate. Let's put another one on the input so we can see input and output on the same scope trace. So you'll see this one's blue. Blue will be input, green will be output. That's pretty much it.

To simulate you just click this button here - this Run button; it doesn't run for a specific time - it runs until you stop it. So we'll run it for a little while then stop it and then look at our scope. When I click run you can see the values changing and then if we click stop that's it run - and by the way, if you have made any errors like forgotten your ground point, what will happen is you'll get a box will pop up on the left here telling you what your error is, ok, and then to see the waveforms we just go to this thing called the graph. You can see quite clearly the time base is completely wrong there; it's far too long so we can change that by just clicking on this arrow - oh sorry no clicking on the tool pane here - and you get up your normal scope choices. So we don't have a trigger - we don't really need a trigger because it's an AC signal. You could set a trigger if you want you can set the level of your trigger rising or falling edges, same as with any normal scope; you just move this out of the way. So what we want to do is we want to change the time base, perhaps to 10 milliseconds, perhaps, so the x-axis is going from zero to ten milliseconds. I'm not sure if that's gonna work or not, let's see if that made a difference – I think you have to click here. Yeah, so that's fine so you can see quite clearly the two signals input and output: blue’s input, green’s output and they're doing exactly as we'd expect. You can of course change your voltage minimum and maximum as well here. If you want to export this to a copy of something you can put into your report, just click on this button here and you have the option of save: saving it as a specific waveform. Um streamed, which I think will stream it to the National Instrument’s server but the key function I think for you guys is the export where you can export it as an image to the schematic or to a graph.

So I'll just show you to prove that this is actually working, let's just change the gain and rerun it. So I'm going to change the gain to two by changing this resistor value. You've got a slider down here, or you can just set it up in here, which I think is quicker. By the way, if you want to come out of any menu you have to left-click somewhere in a blank space and that takes you out of the menu. That's true for components and for the right and left menu bars as well. So we'll close that so we've got more space, and then you'll see it says those are out of date because we've edited the circuit so if we just run that again and then stop that and then look at the graph. So quite clearly our green waveform is much bigger now, which is our output waveform and should be double the size. So we probably want to change our minimum and maximum. I’ve not tried this ‘zoom all’ button - I wonder if you can do that - what does that do? No that doesn't work - okay never mind, so we'll just reset that to what we had before which was 10 milliseconds and it's automatically zoomed the y-axis for us. So that's fine and the reason it went - when I did the zoom all - the reason it went to a long time base was because we had simulated it for that length of time anyway. So hopefully you can see there that the green waveform amplitude is indeed twice the blue one. You can see it, put some cursors on that if you want to make measurements as well. I'm sure there's a way of doing two cursors but I'll let you find that out

for yourselves. Regardless you can do it, you can put your own cursors on and check that okay.

That's all then. I hope you find that nice and easy to use if you can't get Multisim download it will certainly let you complete your courseworks anyway. Bye for now