

## Tutorial:

# SPICE Modeling Technique Using Multisim Circuit Components

---

*Estimated time to complete: 1hr*

## Introduction

Multisim users that are creating new components, often find themselves with the annoying situation of not being able to find a SPICE model to simulate the component. Even by looking into a manufacturer's website (such as Maxim®, Analog Devices®, Texas Instruments®, and so on) they are not able to get a simulation model.

There is one technique that will be presented with this tutorial that might offer an alternative solution for these users. An IC is by itself a complex circuit packaged in a small component, when manufacturer's create SPICE models, they are actually describing the internal circuit that makes that component. This proposed technique has helped us to be able to create new SPICE macro models for some components just by creating a Multisim circuit that describes the internal schematic of the component. In order to use this method, a thorough knowledge of the underlying circuit is required. In some cases, you can use circuitry that resembles the same behaviour to get approximate or equal results as if the SPICE model would have been built for the component.

The process is quite straightforward:

- a. Create the circuit equal to the internal behaviour of the IC
- b. Export the netlist file
- c. Modify and adapt the netlist file to behave like a SPICE macro model
- d. Create a new component using the netlist file as the simulation model
- e. Use/test the new component

For this demo script we will be using the **MAX738A** as the example IC, this is a 5V, Step-Down, Current-Mode PWM DC-DC Converter from Maxim Semiconductor®, if you find this component in Maxim's website (<http://www.maxim-ic.com/>) you will notice that there is no simulation model.



**Note** All files referenced in this demo script are included in the same folder location. There is no need to download files, although finding the files in the web or the samples folder of Multisim allows you to practice real situations encountered by the users.

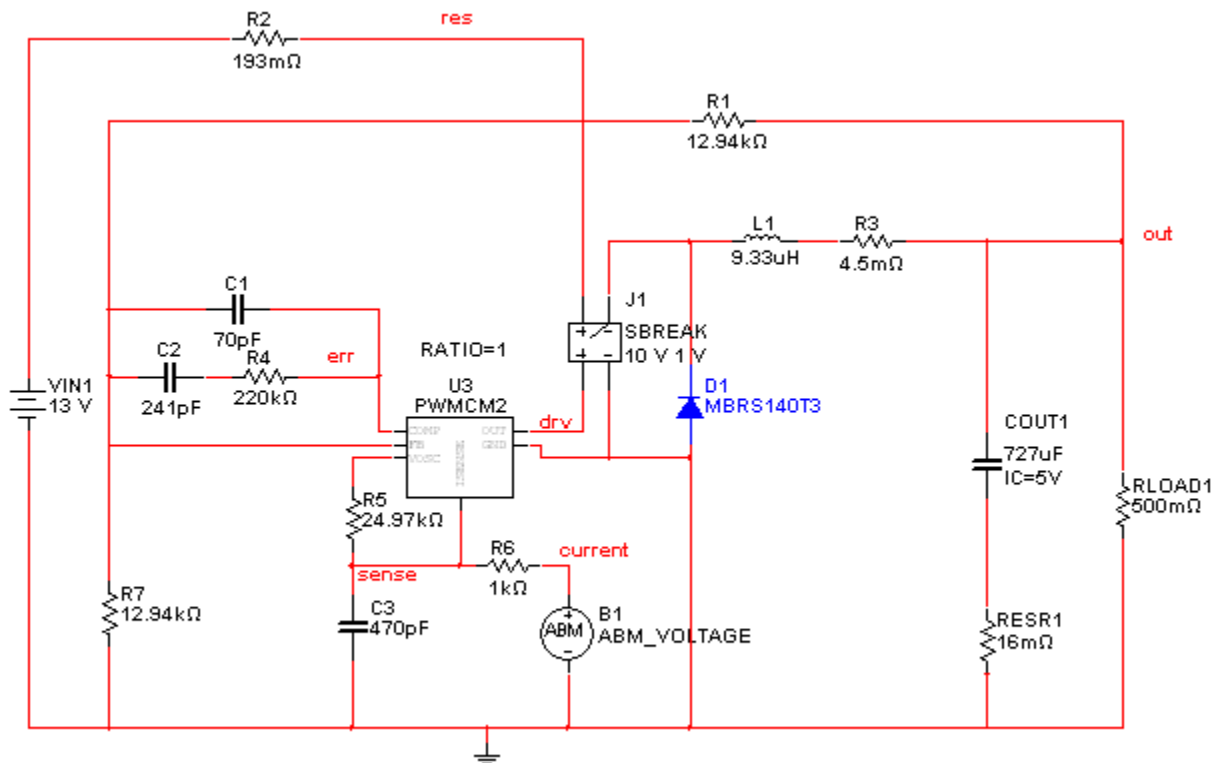
## Implementation

1. Get all necessary files.
  - ☐ Download the MAX738A datasheet from Maxim's website, or open the file MAX738A.pdf from this demo script's directory location.

- ❑ Open the datasheet and very quickly get familiar with the contents of the file and the component itself, look for: pin configuration, pin descriptions, test circuit and package information; these are all details needed when creating a new component.

The MAX738A is a buck converter working on current-mode. Since the datasheet does not include specific details of the internal circuitry values it will be impossible for you to build one. Instead use a similar circuit.

- ❑ In Multisim, select **File » Open Samples...**, open the folder **SMPS Circuits » Transient Analysis** and select the Buck\_CM.ms11 file. Click **Open**.
- ❑ Familiarize yourself with this circuit and save it to a different folder location since you will be editing it.



## 2. Confirm circuit behavior.

- ❑ In Multisim, place a **2-Ch Oscilloscope** virtual instrument and wire **Ch-A** to the positive terminal of **VIN1** and **Ch-B** to the **out** net.

- ☐ **Run** an interactive simulation and observe the oscilloscope graph to understand the circuit response.
- ☐ **Stop** the simulation and **delete** the oscilloscope.

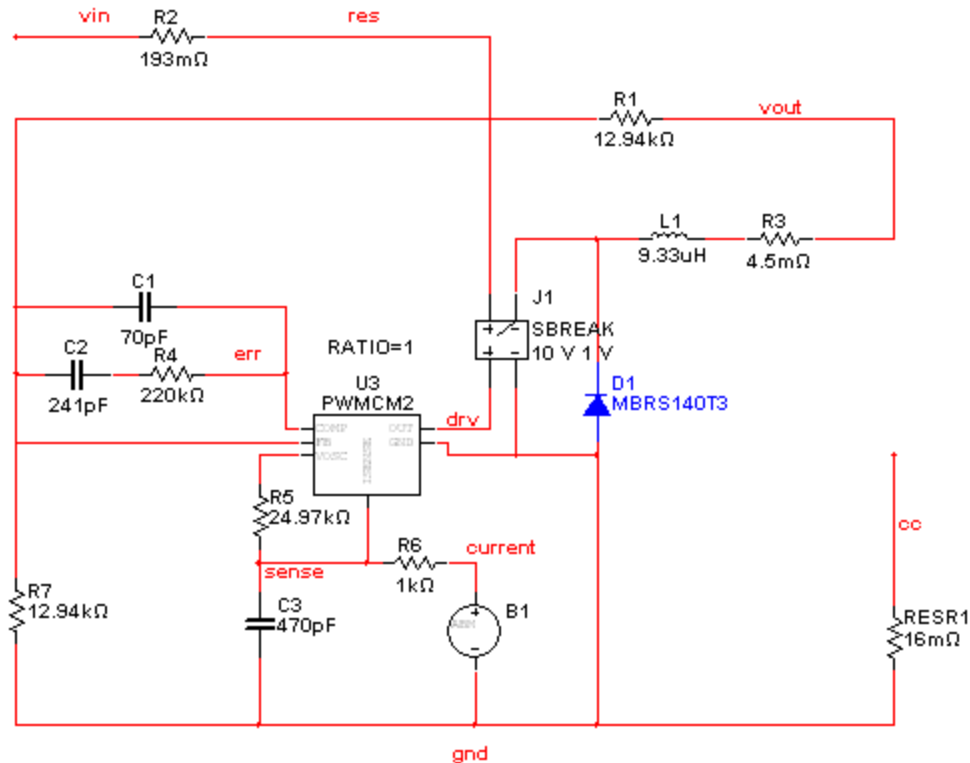
### 3. Prepare the circuit for netlist export.

With some intuition by looking at the datasheet (p. 10) of the MAX738 you can determine that **VIN1**, **COUT1** and **RLOAD1** from the Multisim circuit are all external components from the IC circuitry. **VIN1** is the input terminal for the voltage supplied to the IC, **COUT1** is the compensating capacitor which in the datasheet is a 330 pF component, and **RLOAD1** is the load to which you will supply the output voltage.



**Note** When exporting a netlist using the technique presented here, the Multisim circuit must only have the IC's internal circuitry.

- ☐ Save the file with a different name such as:  
`Buck_CM_export.ms11`.
- ☐ Double-click the net connected to the positive terminal of **VIN1** and name it `vin`. Extend a mid-air wire to the left side from this net.
- ☐ Double-click the ABM source **B1**, in the **Value** tab change `in` for `vin` inside the expression. Click **OK** to exit.
- ☐ Delete the ground reference located at the bottom. Extend a mid-air wire to the left side of the circuit from this net. Double-click the net and change the name to `gnd`.
- ☐ Delete **VIN1**.
- ☐ Double-click the wire of the **out** net, and change the name to `vout`.
- ☐ Delete **COUT1** and **RLOAD1**.
- ☐ Extend a new mid-air wire from the top terminal of **RESR1**, name it `cc`.
- ☐ You can reference the `Buck_CM_export.ms11` file located in this tutorial's folder for the finished circuit.
- ☐ Save the file.



#### 4. Export the netlist.

Now that the circuit represents only the IC's internal configuration, you are now ready to export the netlist.

- ☐ Select **View » SPICE Netlist Viewer**, click the **Save** button in the Netlist Viewer toolbar, select a folder location and give the file a name such as MAX738\_temp.cir and click **Save**.
- ☐ Using Notepad or any other text editor, open the MAX738\_temp.cir file you just created and review it.
- ☐ You may close the SPICE Netlist Viewer at this point.



**Note** Multisim leaves this exported netlist in a raw and verbose SPICE format and additional cleanup and some SPICE formatting is necessary before you can import the model back into a component.

#### 5. Modify the netlist file so it can be used as a SPICE macro model.

In SPICE, all comments start with a \* character. You can delete comments or add comments as you need. The purpose of this step is to make the file readable and easy to understand its structure; you will also add the necessary expressions so it will behave like a SPICE macro model. You can reference the MAX738.cir file located in the same directory of this tutorial for a finished file.

- ☐ Save the file as MAX738.cir so you can make modifications.
- ☐ Delete any comment that you feel just clutters the body of the file, and add any comments that you might need in order to understand what this file is for if you had just opened it.
- ☐ Add the following line at the top of the file, just after the first set of comments:

```
.SUBCKT MAX738 gnd vin vout cc
```

The preceding line indicates that this is a SPICE subcircuit macro model. The macro model also has to be terminated at the end of this circuit's macro model netlist.

- ☐ Look for the **dd1** component (the comment: \*## Multisim Component D1 ##\* precedes this line), immediately after this component add the following line:

```
.ENDS MAX738
```

The **dd1** component represents the last component of the macro model netlist. The rest of the expressions and text below this point in the file are specific model information and subcomponents; think of this as a C language source file with the `main( )` function at the top followed by the functions used by `main( )`.

- ☐ You must now move some code from outside the `.SUBCKT` and `.ENDS MAX738` lines that you just added. Find and cut the following subcircuit text, located within the component J1 listing:

```
.SUBCKT VSwitchJ1 1 2 3 4
aS1 1 2 3 4 vsw0
R1 3 4 1e9
.model vsw0 v_switch ( Roff=1e+006 Ron=0.01
Voff=1 Von=10 )
.ENDS
```

- ☐ Now paste that text immediately after the `.ENDS MAX738` line. Save the file and close it. You can reference the included file with the demo script for a finished version if you are having trouble.



**Note** The following steps assume you are familiar with the Component Wizard in Multisim.

6. Create a new component using the exported netlist.

- ☐ In Multisim, launch the Component Wizard. Select **Tools » Component Wizard**.

- ❑ *Step 1.* Fill the following component information and click **Next**:

Component Name: MAX738A  
Function: 5V , Step-Down, Current-Mode  
PWM DC-DC Converter.

- ❑ *Step 2.* Click **Select a Footprint**. You need to select the SOIC-16 footprint. *Hint:* Use the **Filter** button with Footprint = SOIC-16. Back in the main dialog, set **Number of Pins** to 16. Click **Next**.
- ❑ *Step 3.* Click **Edit** to create a new symbol, use the datasheet as a reference (p. 1, 7). Make sure you then click **Copy to...** and copy the ANSI symbol to the DIN set.  
In the location of this demo script you will find an already made symbol you could use, is the file with name: MAX738A.sym.
- ❑ *Step 4.* Change the pin type if you like. In here we strongly suggest you add a NC hidden pin for the Not Connected pins present in the 16-pin package. Click **Next**.
- ❑ *Step 5.* Map the symbol and footprint as described in the datasheet (p. 7), and click **Next** when done:

<b>V+</b>	1, 15, 16
<b>SHDN</b>	2
<b>REF</b>	3
<b>SS</b>	7
<b>GND</b>	10, 11
<b>CC</b>	8
<b>OUT</b>	9
<b>LX</b>	12, 13, 14
<b>N.C.</b>	4, 5, 6

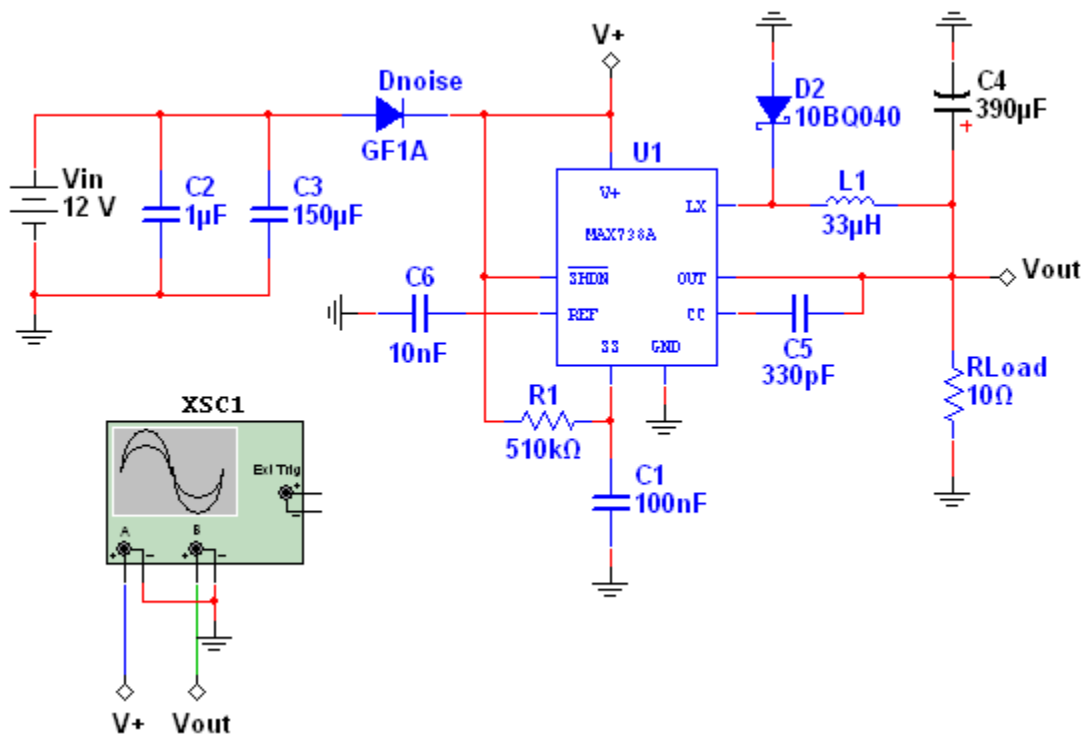
- ❑ *Step 6.* Click **Load from File**. Select the MAX738.cir file we created earlier and click **Open**, then click **Next** to continue.
- ❑ *Step 7.* Configure the symbol and model mapping as follows, then click **Next**:

<b>V+</b>	2
<b>SHDN</b>	NC
<b>REF</b>	NC
<b>SS</b>	NC
<b>GND</b>	1
<b>CC</b>	4
<b>OUT</b>	3
<b>LX</b>	NC
<b>N.C.</b>	NC

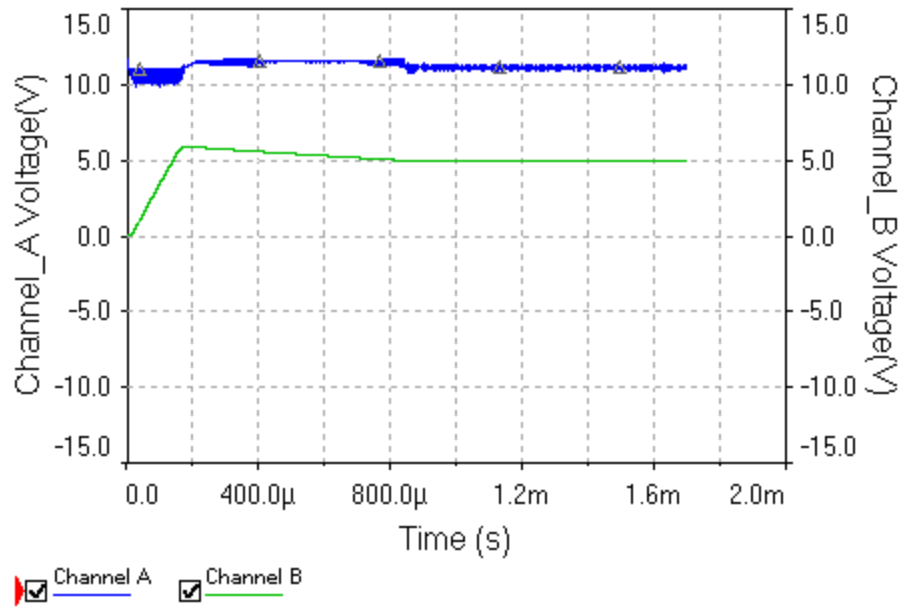
- ❑ *Step 8.* Save the component in the User or Corporate Database under any Group and Family you may want to create. Click **Finish**.

7. Test the new component.

- ❑ In Multisim, create a test circuit with the new component. Use the datasheet as the reference (p. 10, 11). You can also open the MacroModel\_EquivCircuit.ms11 circuit file included in the tutorial folder; the following image is the circuit screen capture.



- ❑ You can reference the datasheet or you can use the circuit with the original components of the Buck\_CM.ms11 file you deleted at the beginning of these instructions.
- ❑ In the MacroModel\_EquivCircuit.ms11 file included, an extra diode **Dnoise** was included to introduce noise in the input voltage. The output should still be regulated at +5V. Play around with the circuit and you'll see that **Rload**, **C4** and **C5** play a big role in the regulation of the voltage.
- ❑ The following graph shows a sample response with interactive simulation.



## Summary

As you may have noticed, pins **SHDN**, **REF**, **SS** and **LX** are not functional, since the model we built does not declare these inputs (as defined in the `.SUBCKT` line). However, using this type of technique to create inexistent models from sample circuits is a very useful and approximate way to simulate the components.

Please note that the most accurate models are typically built by the original manufacturer with extensive research and test involved; the method presented here gives you an alternative way of getting a component to simulate in your circuit, however it is not meant to be the most accurate method to simulate.

The finalized component is also included in the same location as this demo script as a component file (\*.prz) for version 11.0, is called MAX738.prz, you can import it into your **User Database** if you select **Tools » Database » Database Manager** and click the **Import** button.