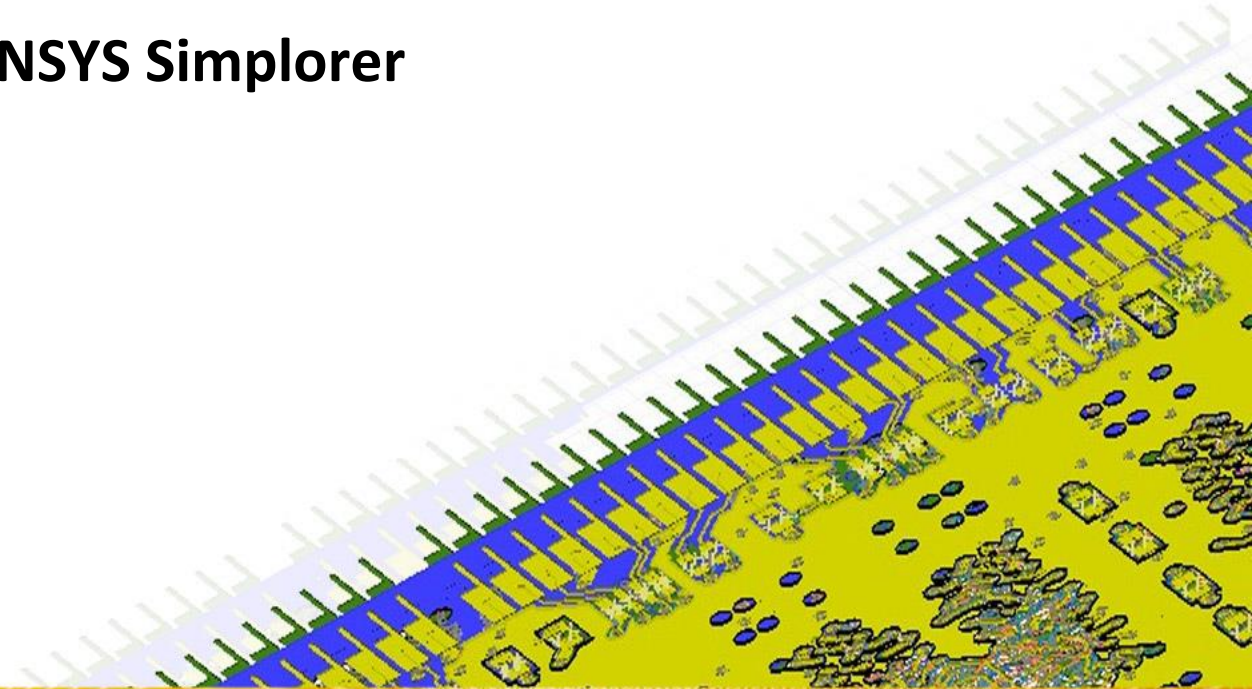




# Workshop 4.2: Non-linear Components

## Introduction to ANSYS Simplorer



# Overview

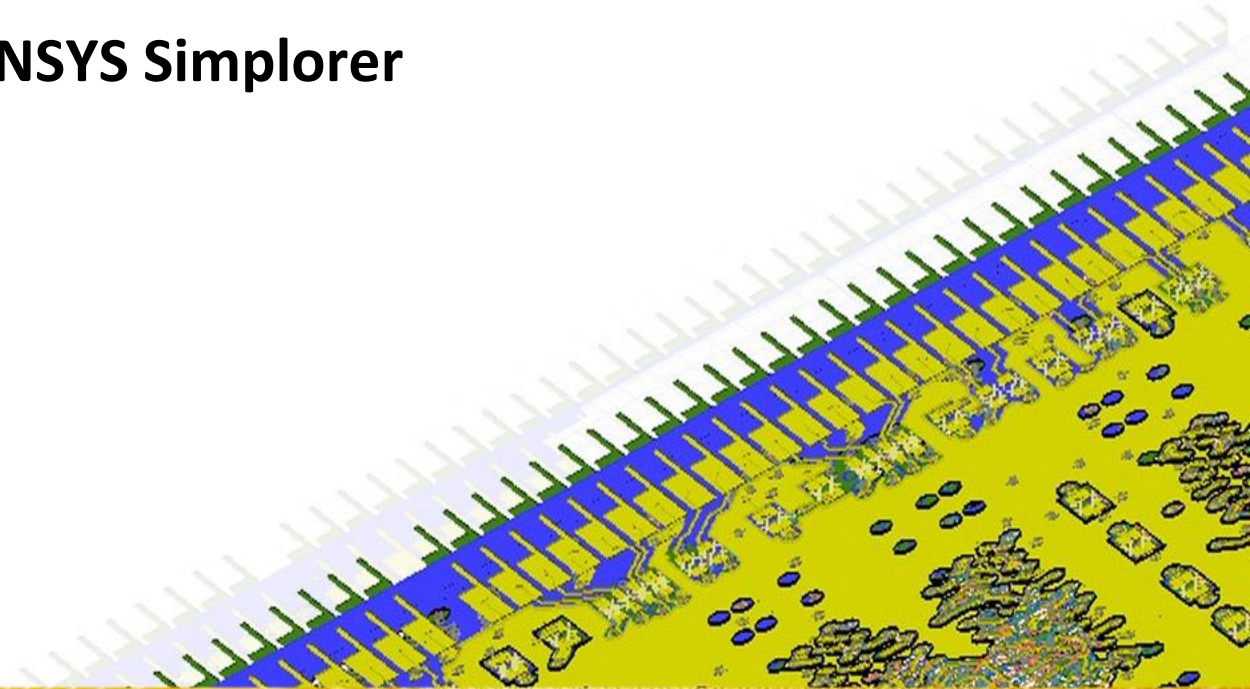
- **Non-Linear Components**

- In the first section of this example we will learn how to create a non-linear resistor that will be used to model a Varistor. An alternate method will also be evaluated using a non-linear voltage dependent current source
- In the second section, the Varistor will be used in a switched inductive application to show its usefulness as transient protection for the switch. When the current through an inductor is interrupted quickly via a switch, there will be a very large induced voltage due to the relationship  $V_L = L di/dt$
- In particular we will learn
  - How to use a dataset for defining non-linear component characteristics
  - How to verify correct behavior comparing results
  - How to use the non-linear component in a circuit to protect a power electronic component





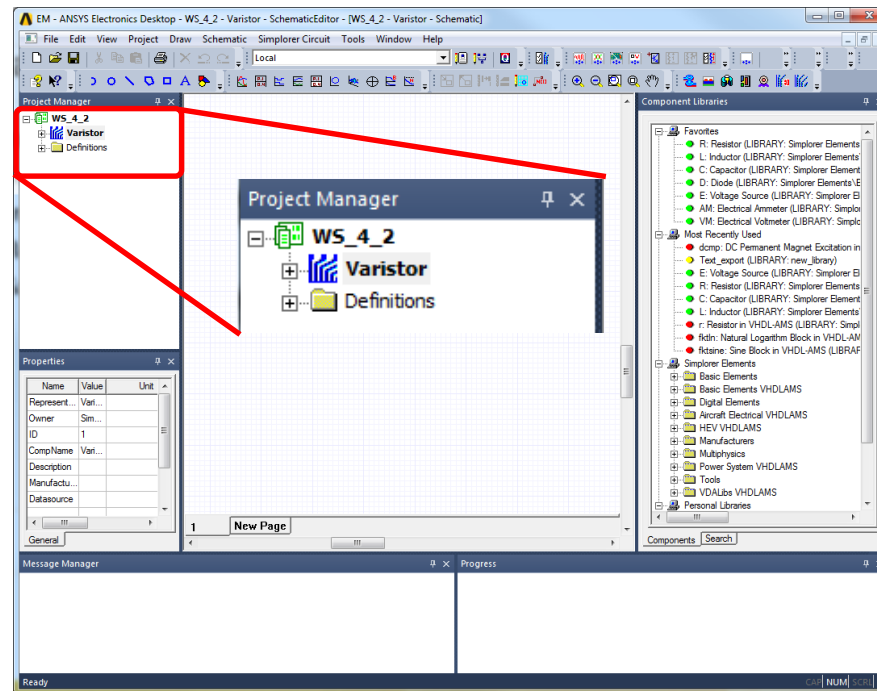
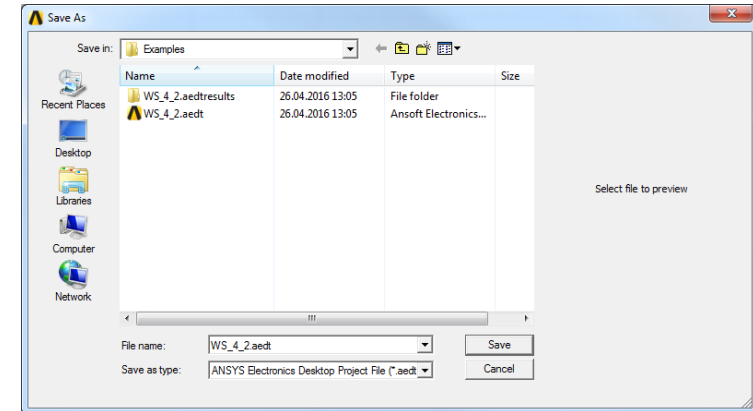
# Varistor

## Introduction to ANSYS Simplorer



# Insert a Simplorer Design

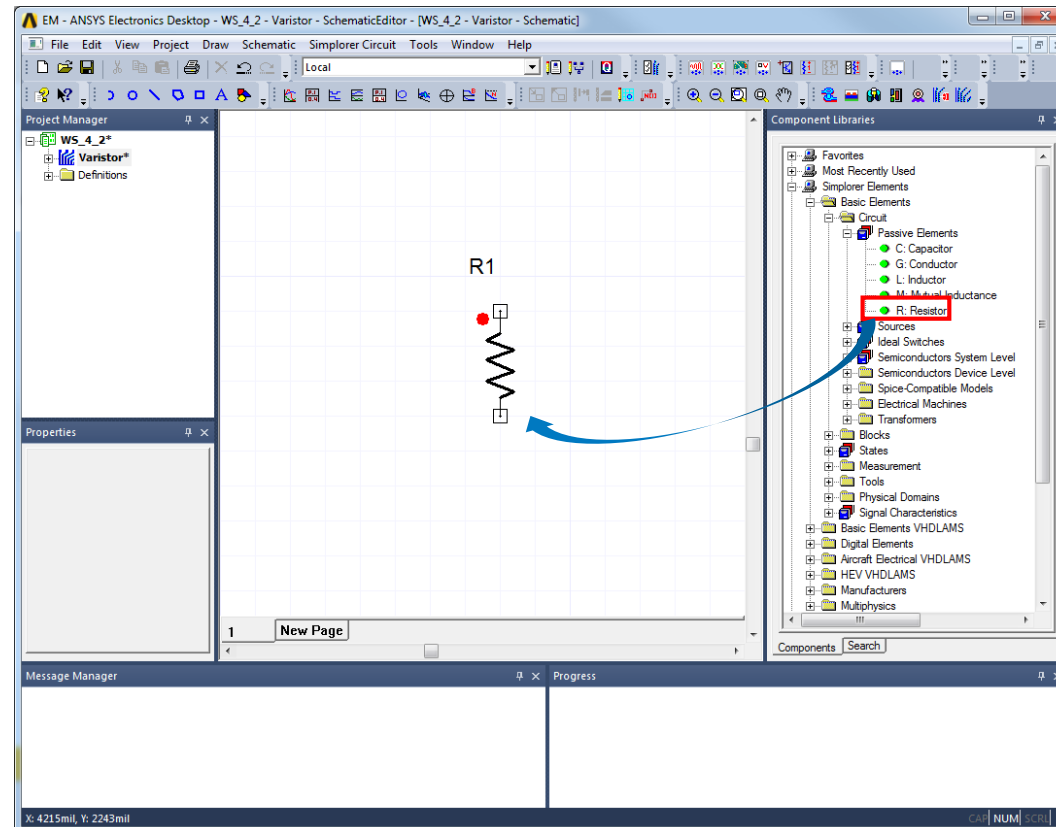
- Launch the Electronics Desktop 2016
  - Save the Project as **WS\_4\_2.aedt**
  - Insert a Simplorer Design using the icon 
  - Rename the Design as **Varistor**
  - Save again the project using the icon 



# Insert Components

- Resistor

- In Component Libraries window *Simplorer Elements* → *Basic Elements* → *Circuit* → *Passive Elements*
- Select the **Resistor** and drag and drop it into the Schematic. Press **Esc** key to exit the insert mode

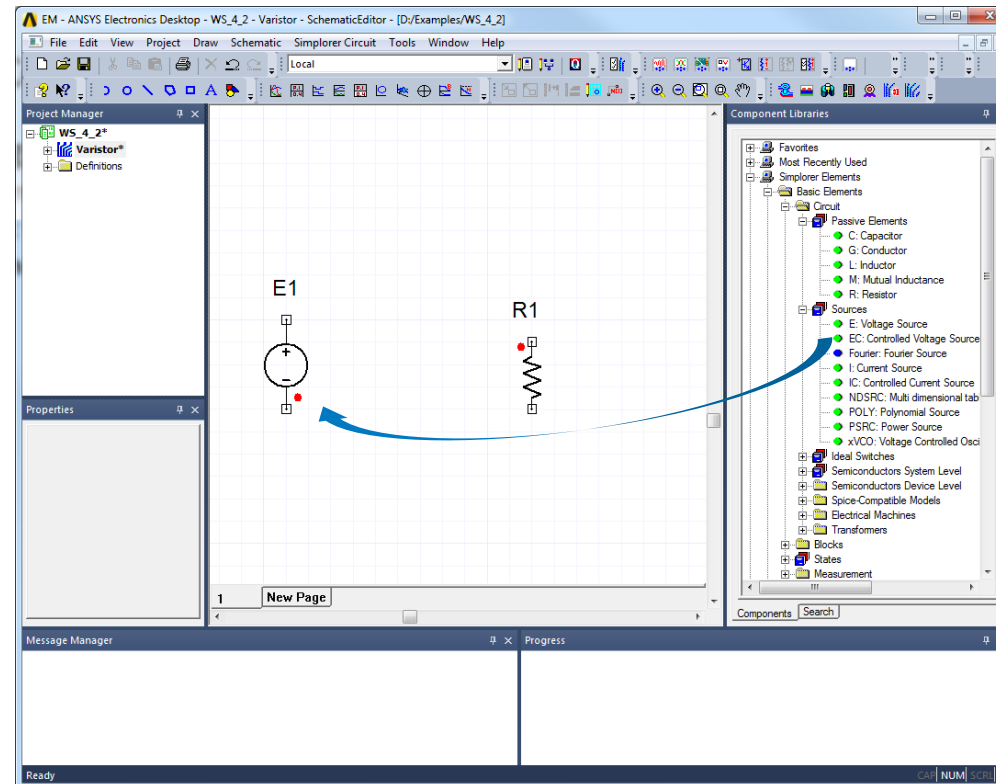




# Insert Components

- Voltage Source

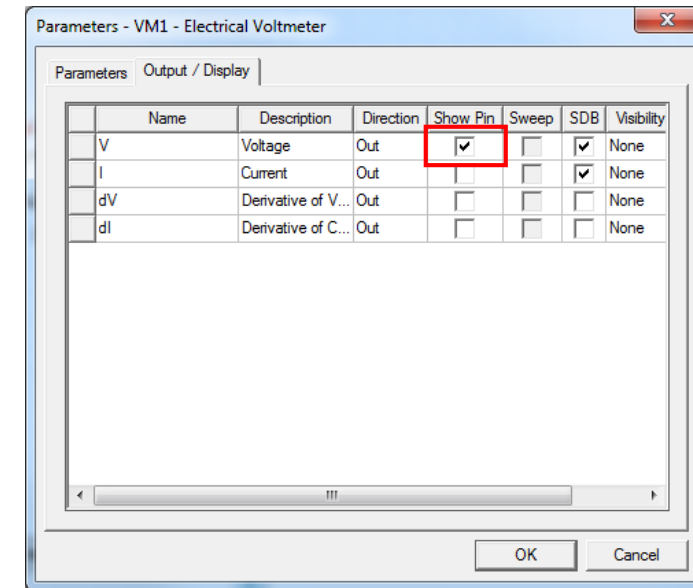
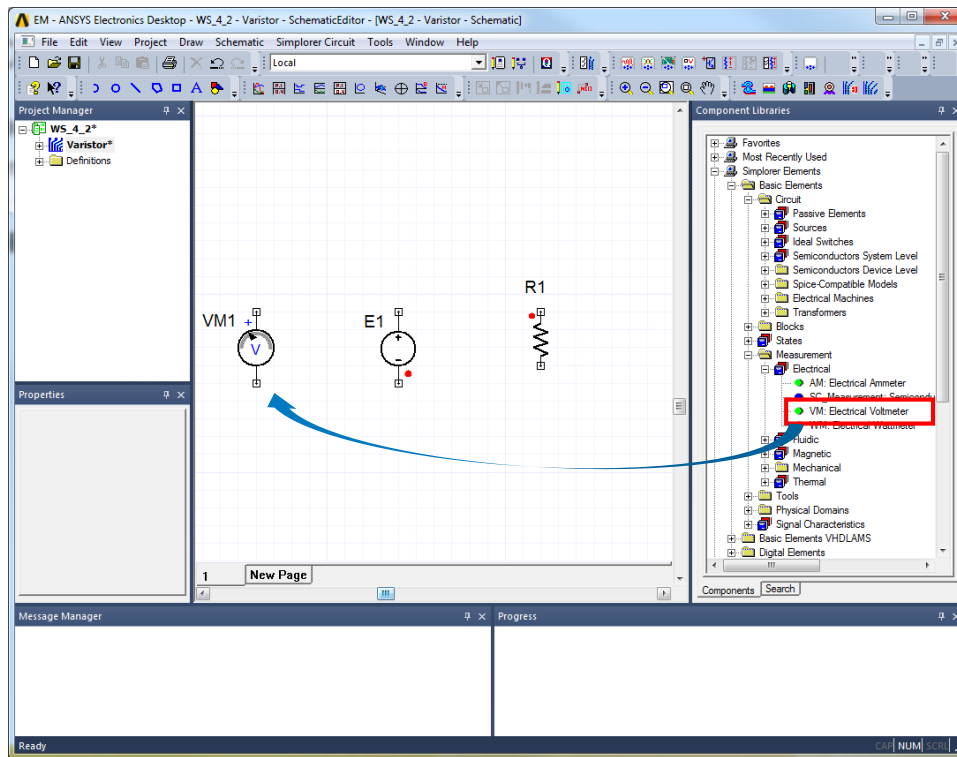
- In Component Libraries window *Simplorer Elements* → *Basic Elements* → *Circuit* → *Sources*
- Select the **E: Voltage Source**, drag and drop it into the Schematic. Press **Esc** key to exit the insert mode
- Use the shortcut **Ctrl+D** to fit the view



# Insert Components

- Voltmeter

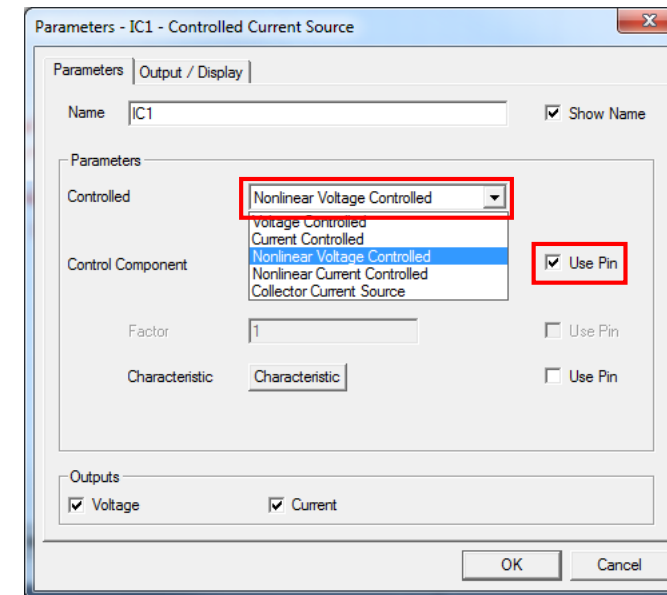
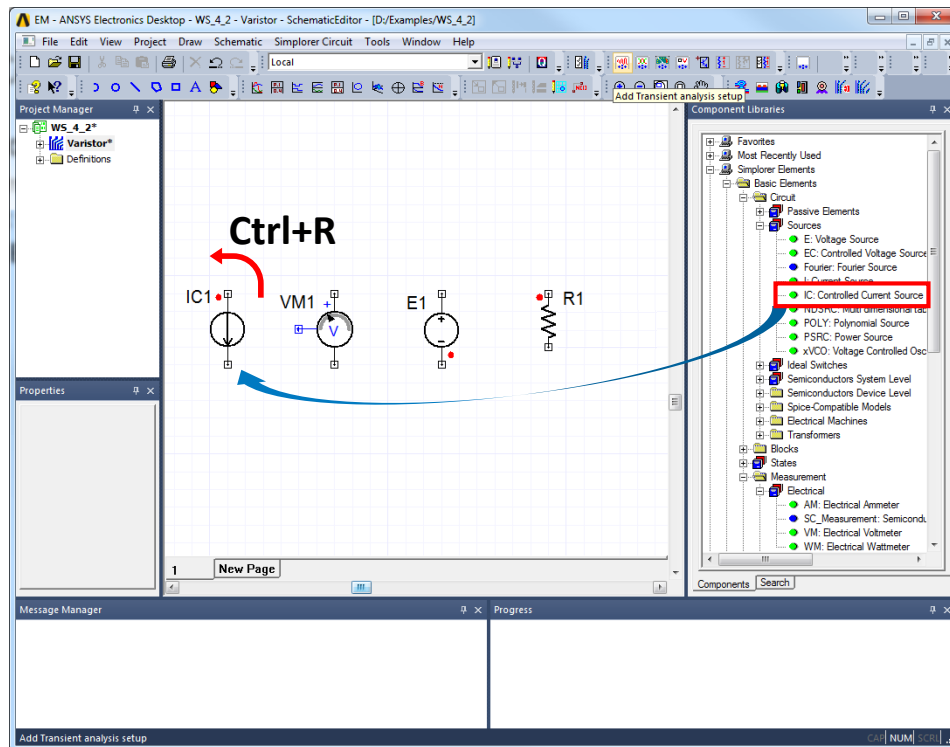
- In Component Libraries window *Simplorer Elements* → *Basic Elements* → *Measurements* → *Electrical*
- Select the **VM: Electrical Voltmeter** and drag and drop it into the Schematic
- Double click on the **Voltmeter**, select the **Output/Display Tab** and check **Show Pin** for the Voltage
- Press **OK**



# Insert Components

- Controlled Current Source

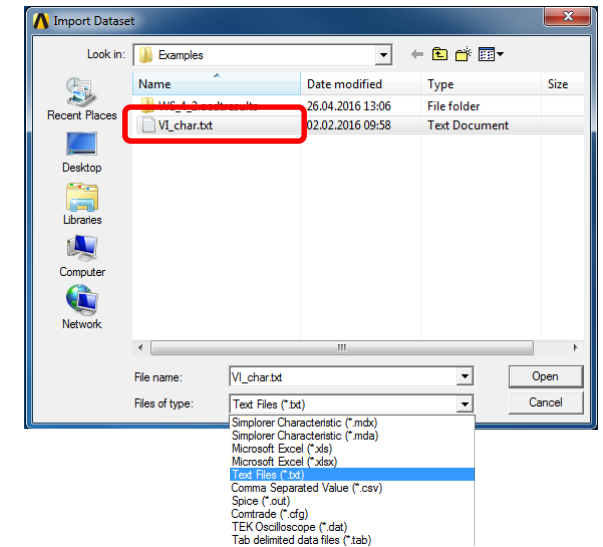
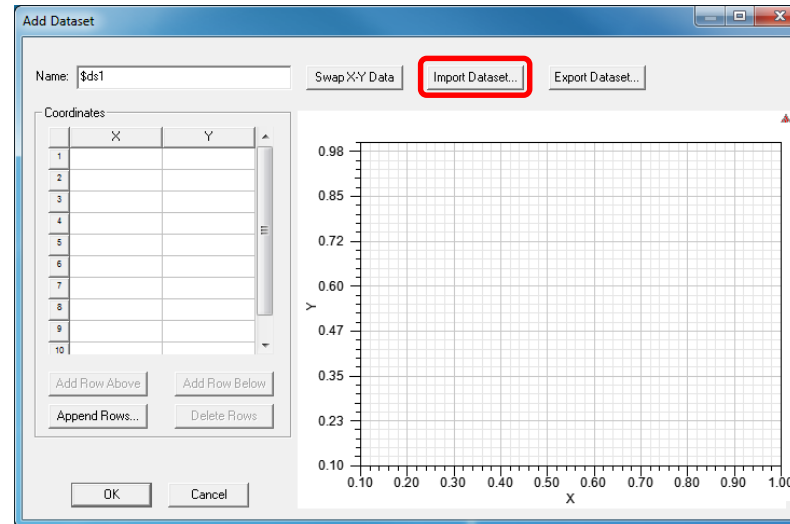
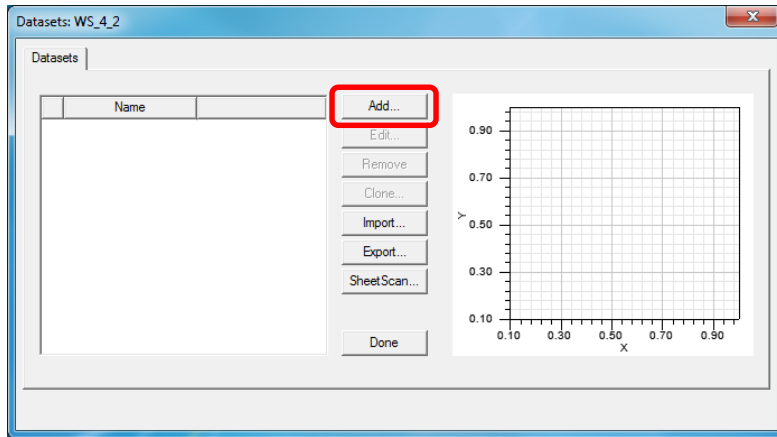
- In Component Libraries window *Simplorer Elements* → *Basic Elements* → *Circuit* → *Sources*
- Select the **IC: Controlled Current Source**, drag and drop it into the Schematic
- Press **Esc** key to exit the insert mode. Rotate it twice using the short key **Ctrl+R**
- Double click on the **Current Source**, select **NonLinear Voltage Control** and check **Use Pin**. Press **OK**





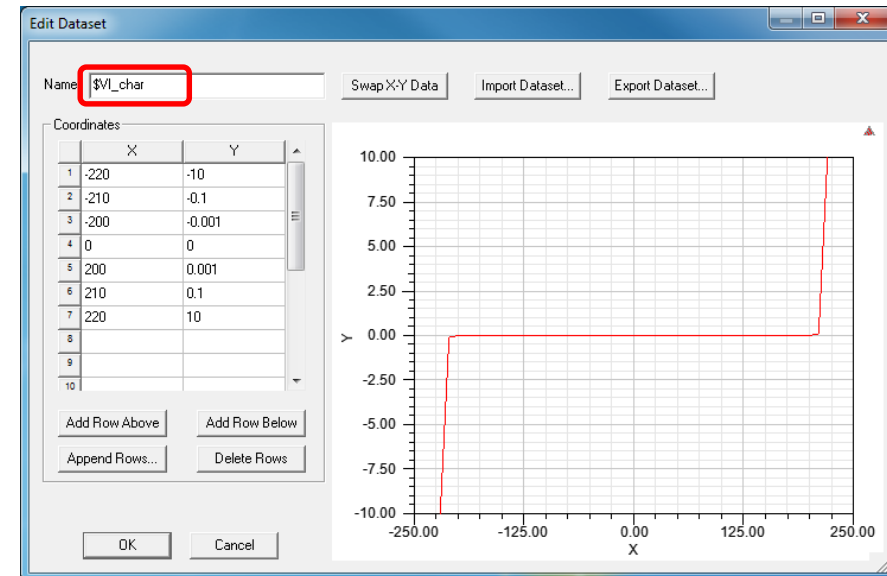
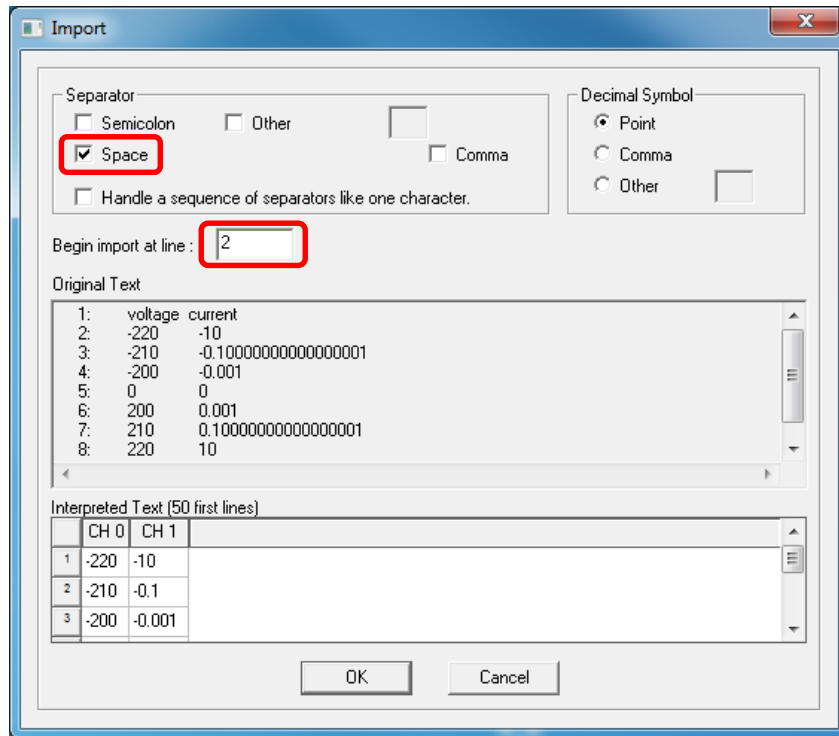
# Creating Dataset

- A dataset will now be created to represent the Non-linear resistance behavior
- Select the menu item **Project** → **Datasets**
- Press **Add** to create new Dataset
- Press **Import Dataset** to load an external file
- Locate the file **VI\_char.txt** under the folder **workshop\_input\_files** and select it. Press **Open**



# Creating Dataset

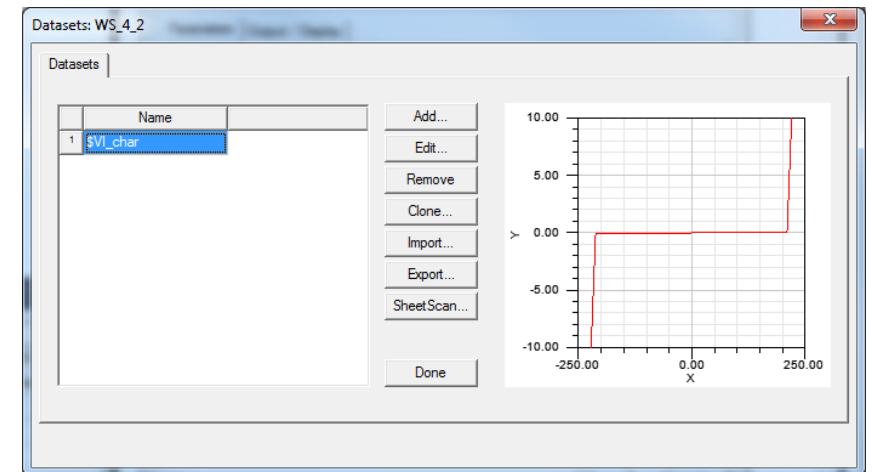
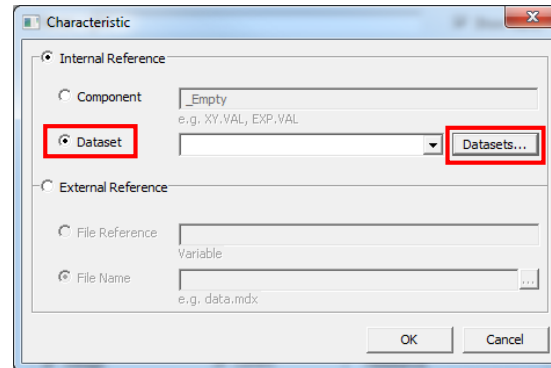
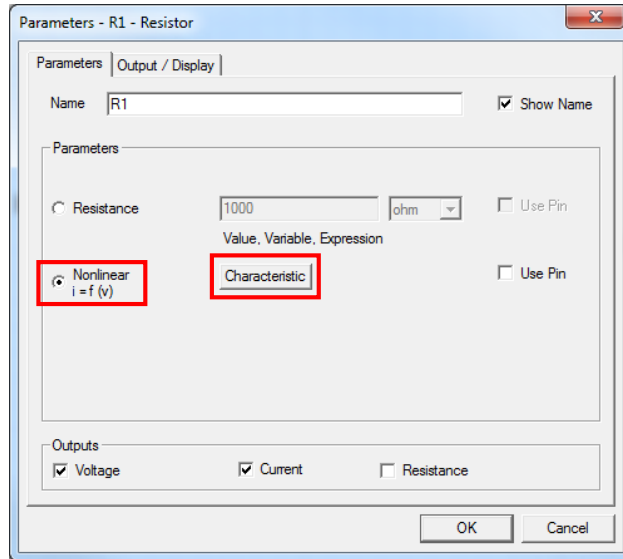
- Select **Space** as column separator and set Begin to import at line **2**
- Press **OK**
- Name the Dataset to be “**\$VI\_char**”
- Press **OK** and then **Done** to come back to the Schematic



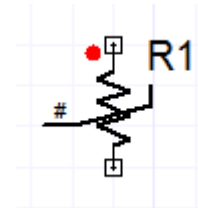
# Set Component Properties

- Resistor

- Double click on the **Resistor** and check **Non linear  $i=f(v)$**
- Click on **Characteristic**. In the Characteristic window, check **Dataset** and press the **Datasets** button
- In the Dataset window, select **\$VI\_char** and then press **Done**, **OK**, **OK** to come back to the Schematic



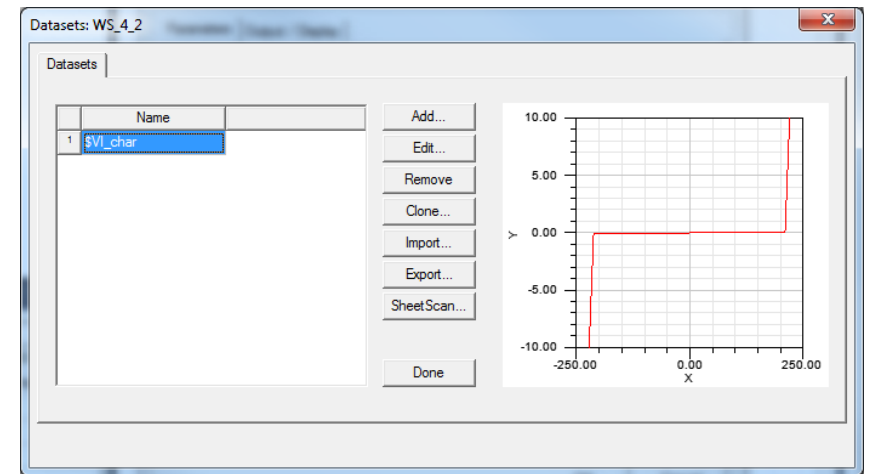
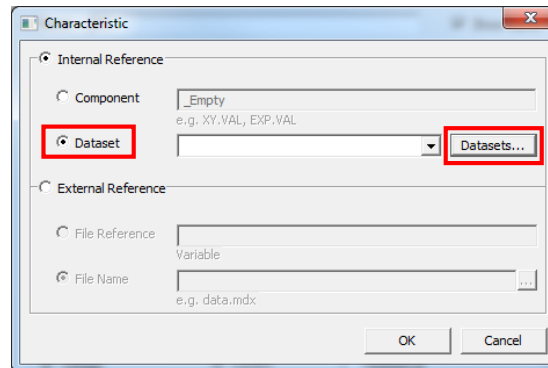
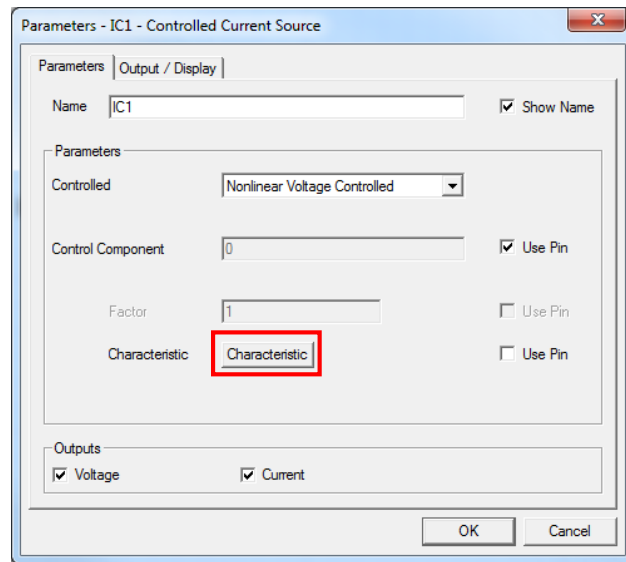
- Note the symbol of Resistor changes, showing the non linear behavior:



# Set Component Properties

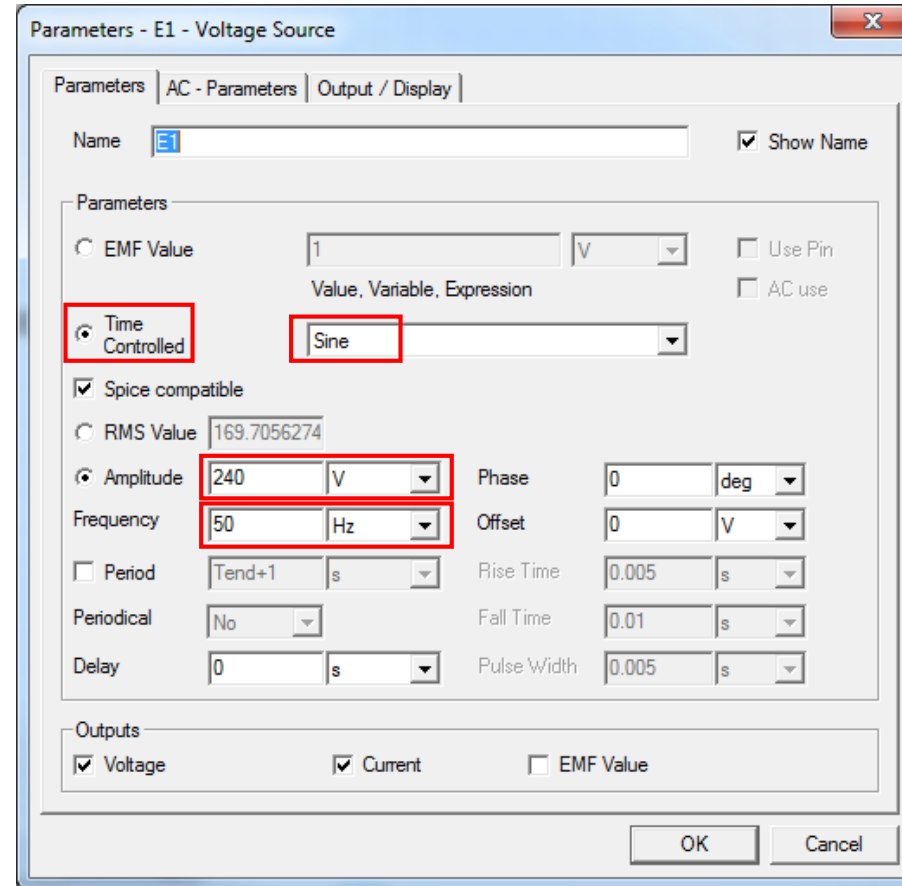
- Controlled Current Source

- Double click on the **Controlled Current Source** and click the **Characteristic** button.
- In the Characteristic window, check **Dataset** and press the **Datasets...** button
- In the Dataset window, select **\$VI\_char** and then press **Done**, **OK**, **OK** to come back to the Schematic




# Set Component Properties

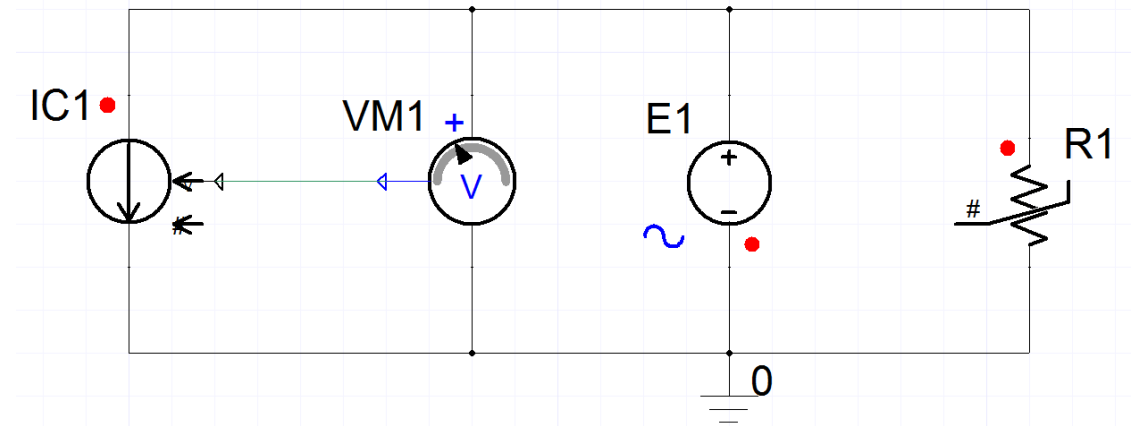
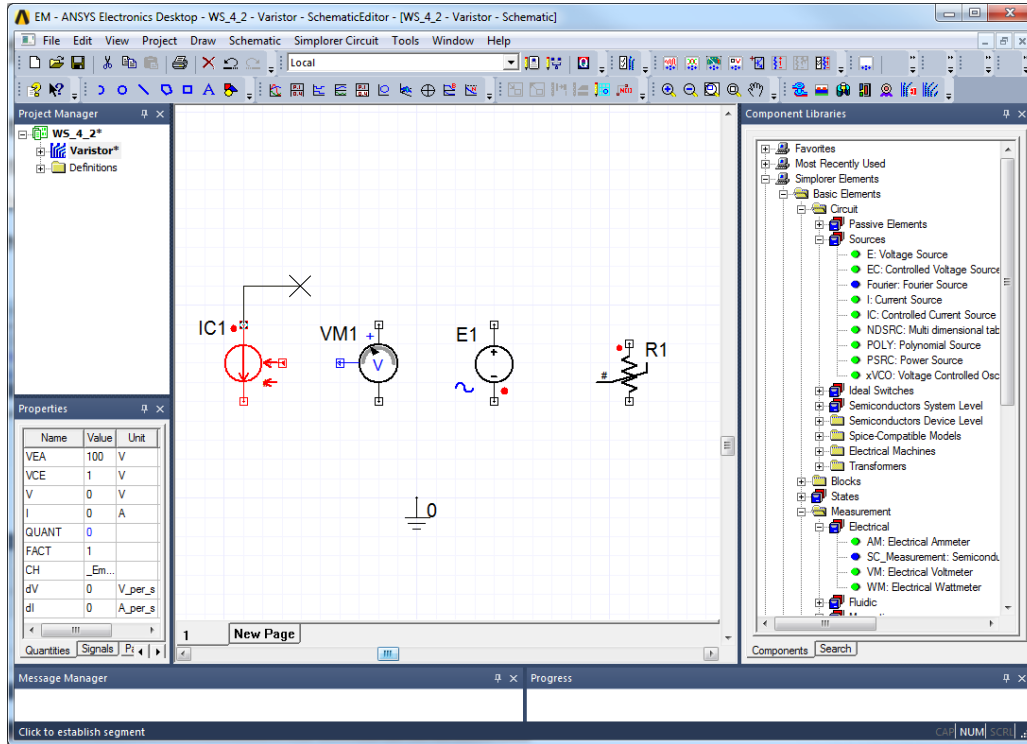
- Voltage Source
  - Double click on the **Voltage Source**
  - Set **Time Controlled - Sine**
  - Check **Spice Compatible**
  - Amplitude: **240 V**
  - Frequency: **50 Hz**
  - Press **OK**





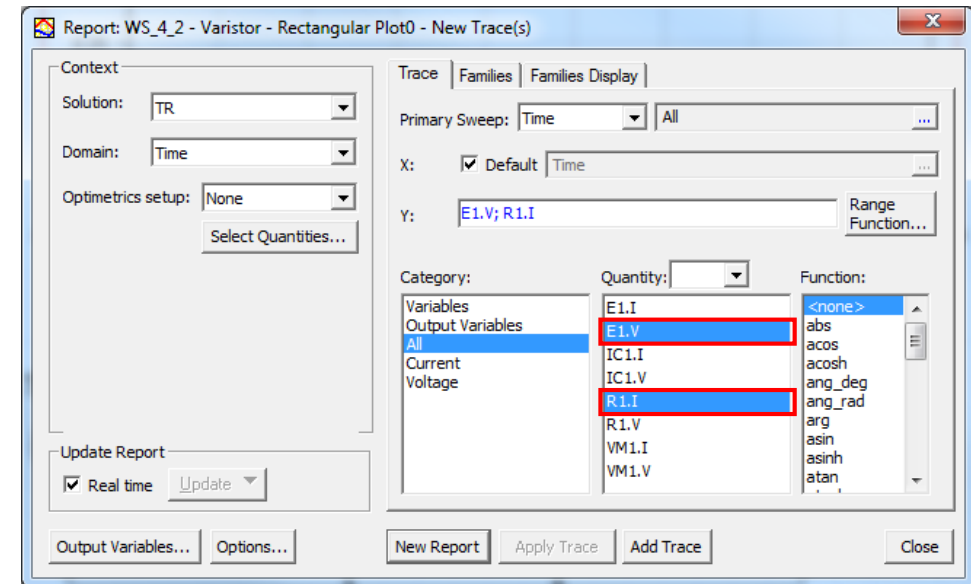
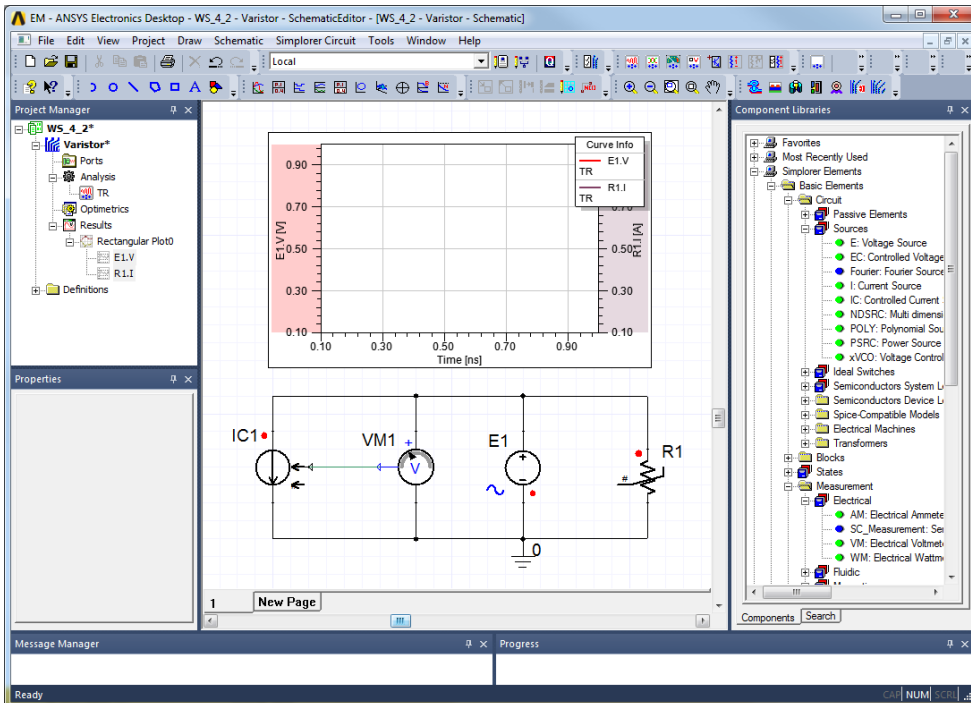
# Connect Components

- Place the mouse over one terminal of a component. The mouse pointer changes its shape becoming a cross. Press the **LMB** and move the cross till the connecting terminal of next component
- Add the **Ground node** clicking the icon  and placing it into the Schematic
- Connect all the components till completing the circuit as shown on the figure below



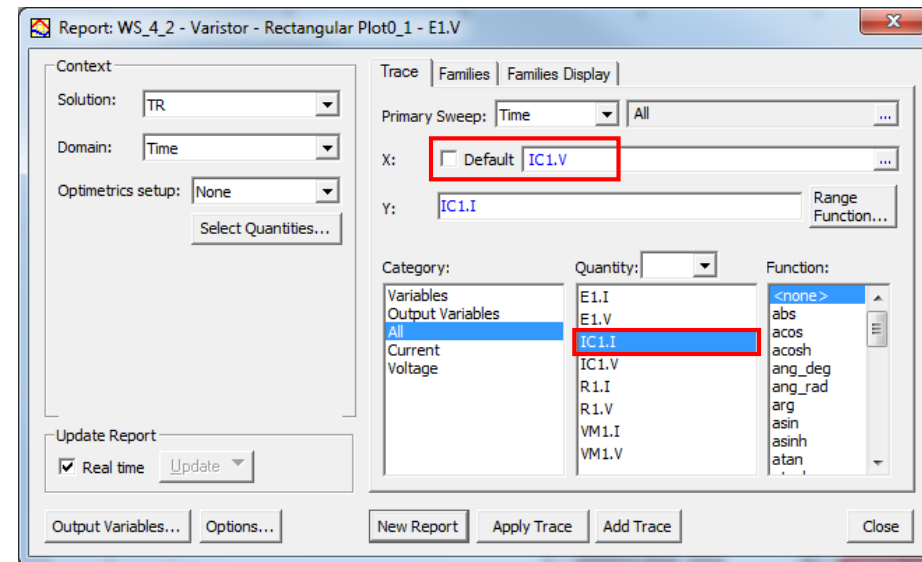
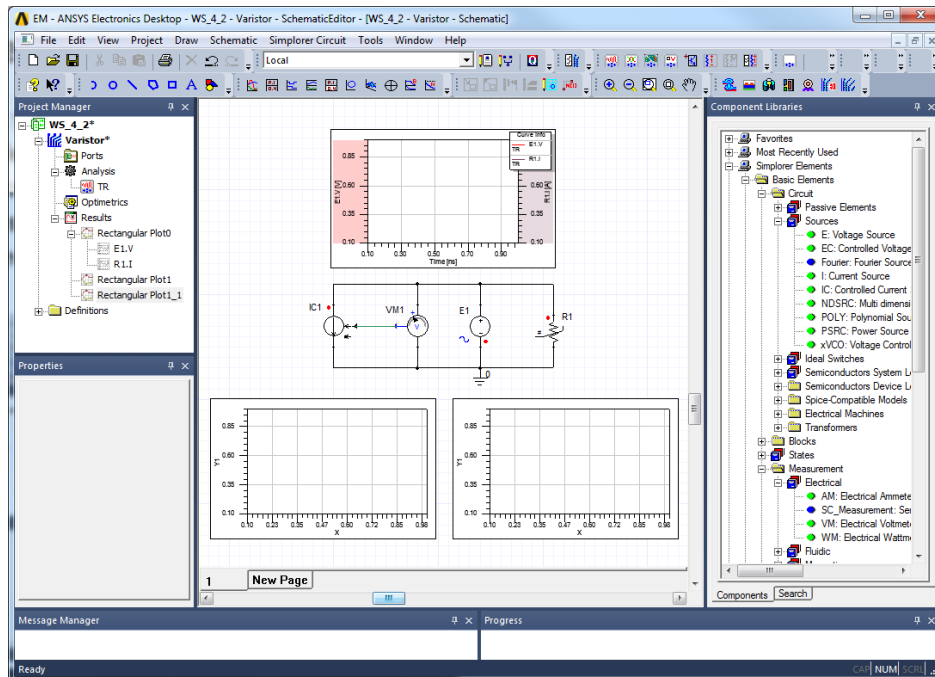
# Prepare the Postprocessing

- Select the menu item **Draw** → **Report** → **Rectangular Plot** and place the plot in the Schematic, for example above the Circuit. It is best to zoom out before placing the report, so you can resize it in relationship to the existing circuit
- Automatically the **New Trace** window pops-up
- Select the Voltage Source voltage and the Resistor current by selecting the quantities **E1.V** and **R1.I**
- Click on the **Add Trace** button and then **Close**



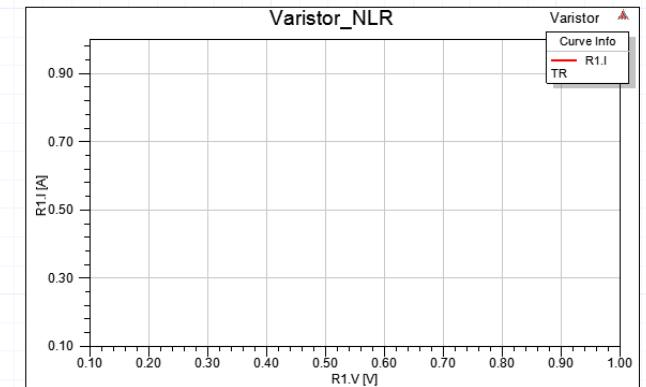
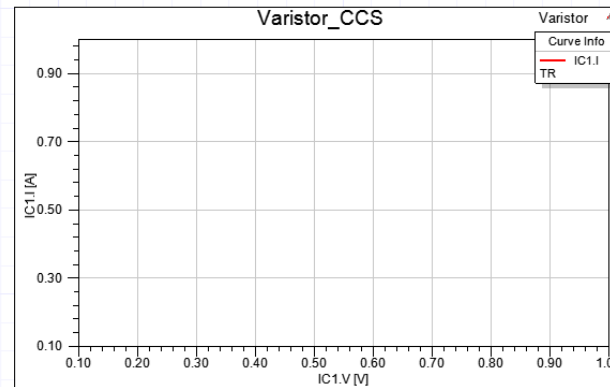
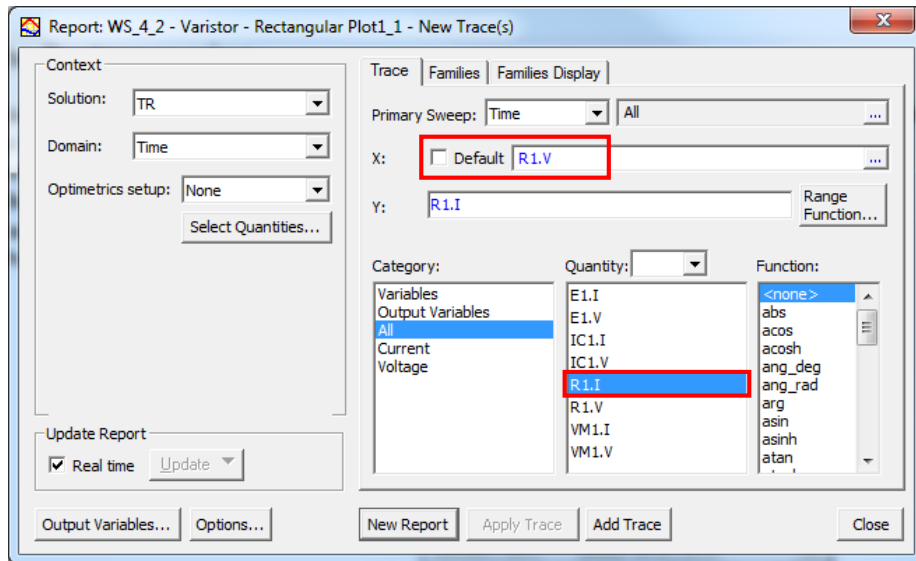
# Prepare the Postprocessing

- Insert two more Plots in the Schematic as shown in figure. Double click on the lower left plot and define it so to display the Voltage-Current relationship for the Controlled Current Source:
  - Un-select the X axis default box and select to use the voltage across the Current Source **IC1.V**
  - Select the Y axis to be the current in the Current Source **IC1.I**, press **Add Trace**, then **Close**
  - On the same Plot, **RMB** → **View** → **Visibility**, in the Legends Tab check only the **Header** Box
  - In the Project manager Window, rename the Plot “**Varistor – CCS**” (Controlled Current Source)



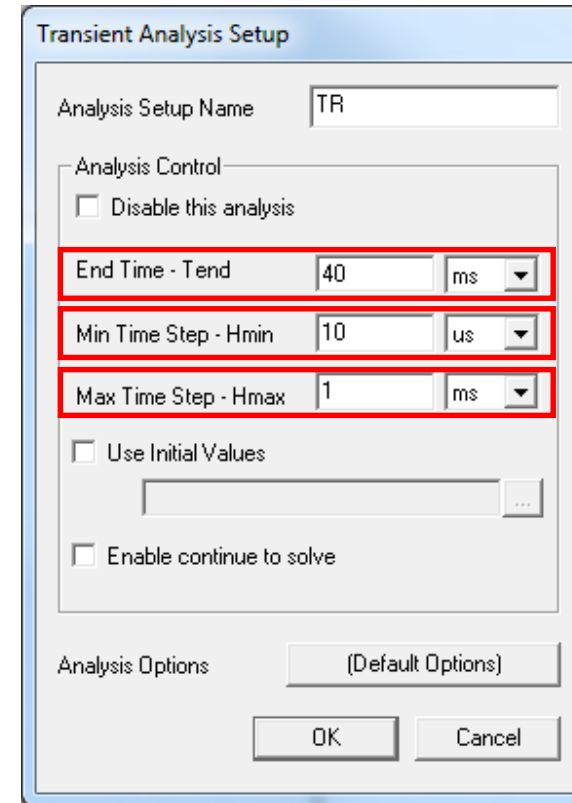
# Prepare the Postprocessing

- Double click on the lower right plot and define it so to display the Voltage-Current relationship for the non-linear resistor:
  - Un-select the X axis default box and select to use the voltage across the non linear resistor **R1.V**
  - Select the Y axis to be the current in the non linear resistor **R1.I**, press **Add Trace**, then **Close**
  - On the same Plot, **RMB** → **View** → **Visibility**, in the Legends Tab check only the **Header** Box
  - In the Project manager Window, rename the Plot “**Varistor – NLR**” (Non Linear Resistor)



# Setup the Transient Analysis

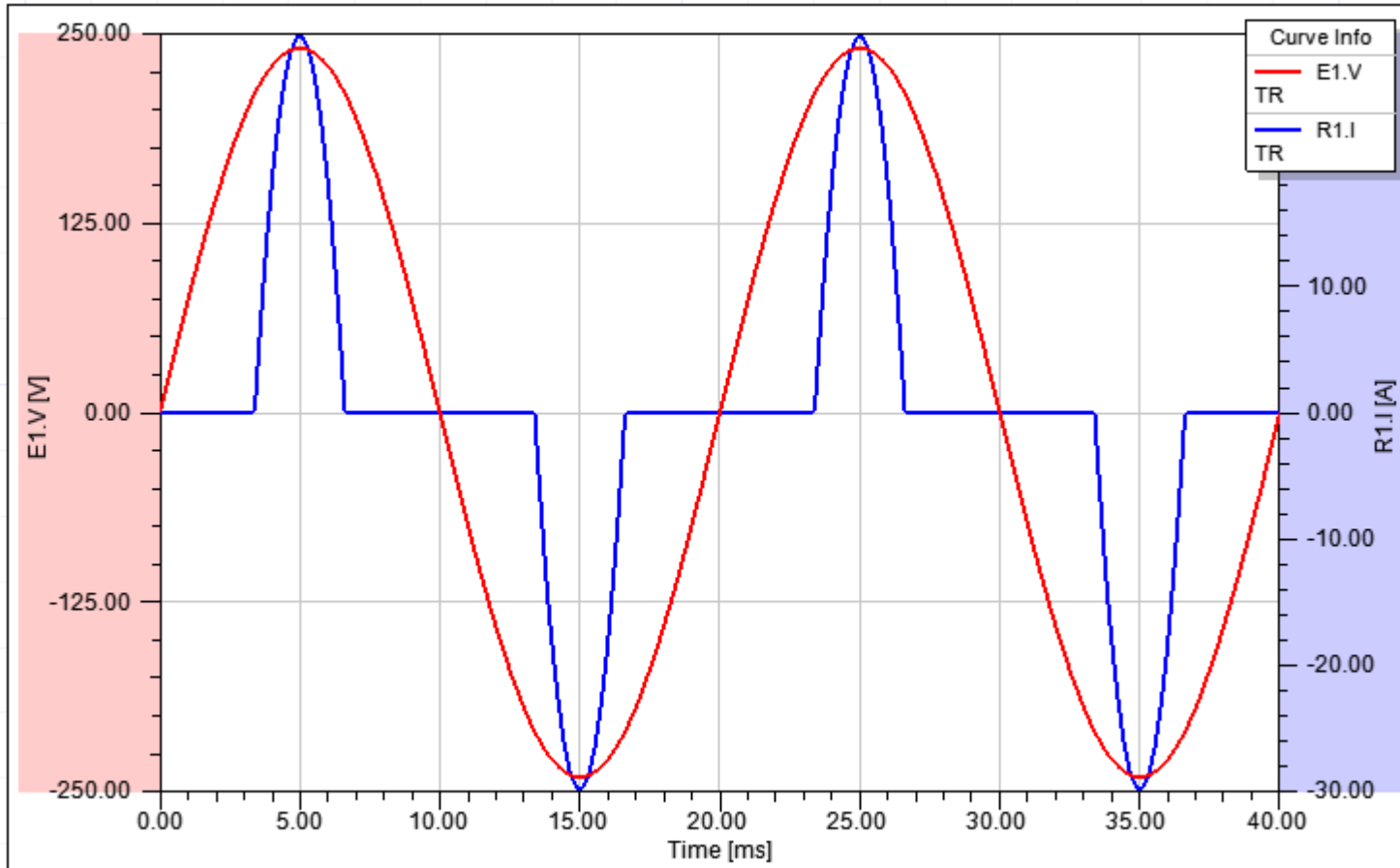
- In the present example it is enough to use the Default settings for the transient analysis
- In the Transient Analysis Setup window:
  - Tend: 40 ms
  - Hmin: 10  $\mu$ s
  - Hmax: 1 ms
  - Press OK





# Analyze and View Results

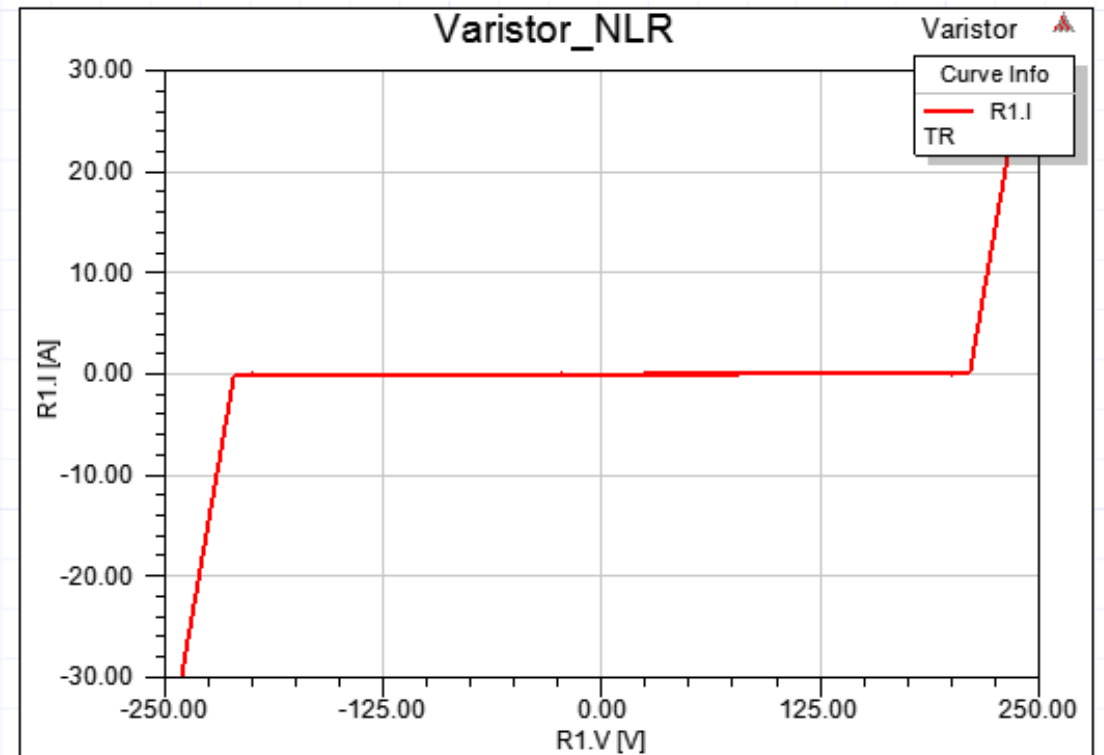
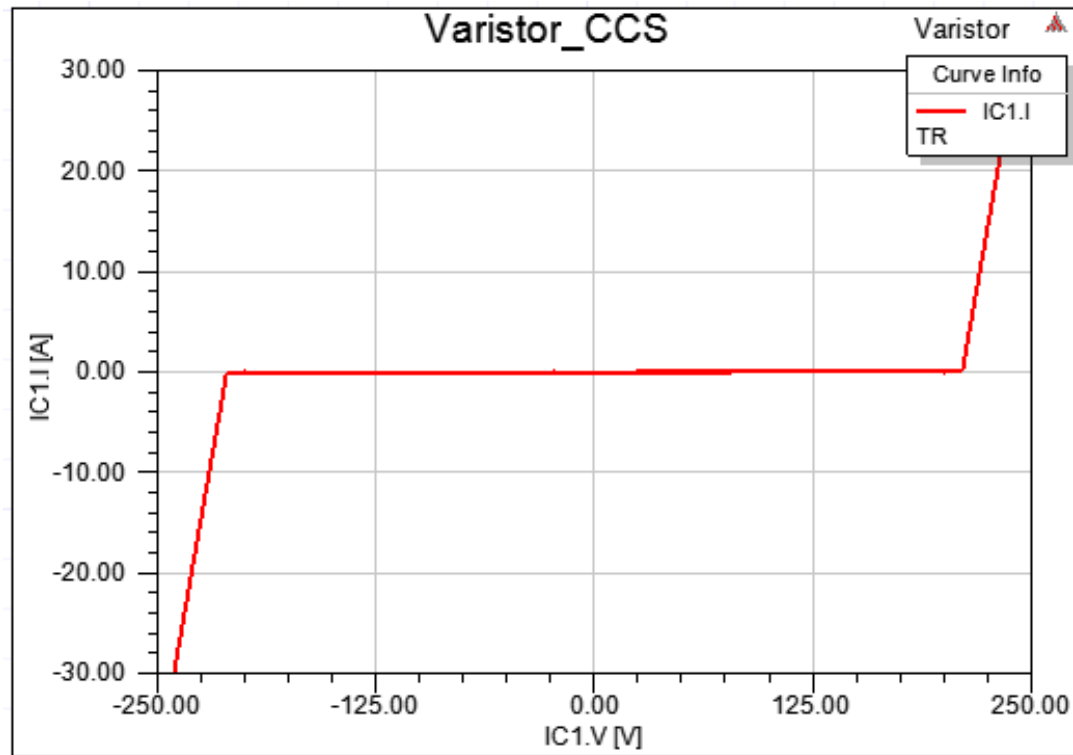
- From the Project Manager window, select the **Analysis** → **TR** → **RMB** → **Analyze** to run the Simulation
- The final result should look very similar to the following Figure:



*Note: Use RMB on the plot area and then “Edit in Place” to modify colors and thickness of plotted lines*

# View Results

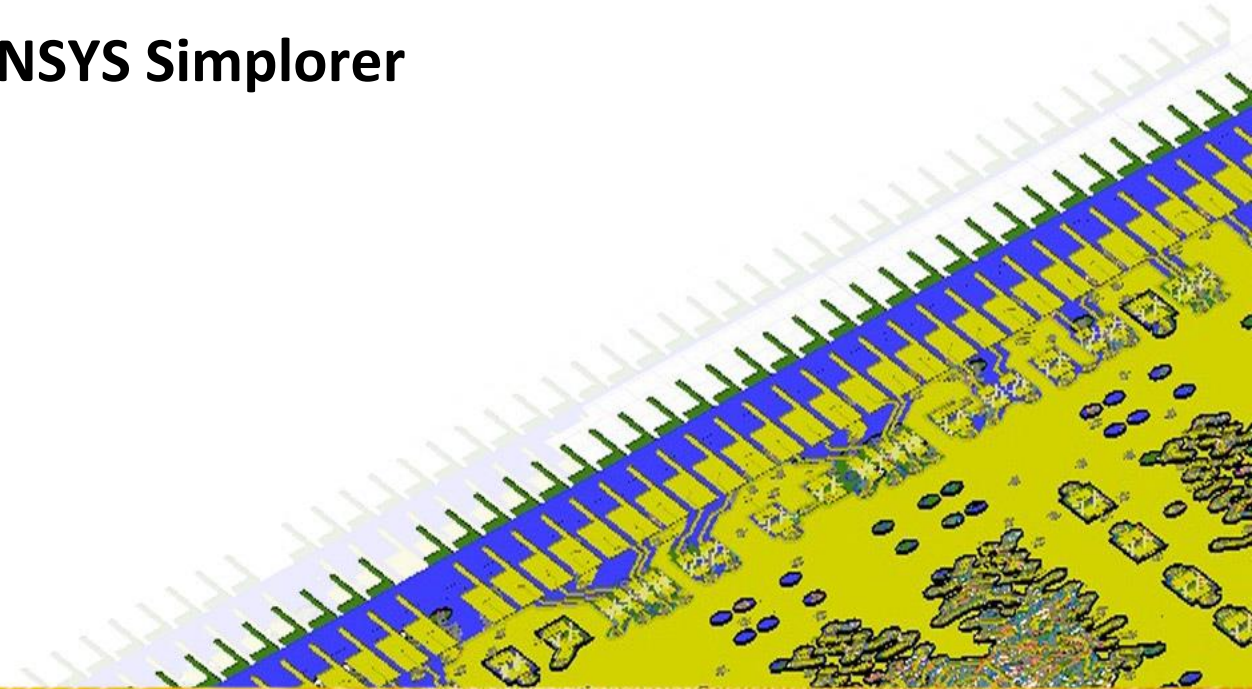
- As expected the two plots showing the non-linear behavior for both the non-linear current source and the non-linear resistor are identical





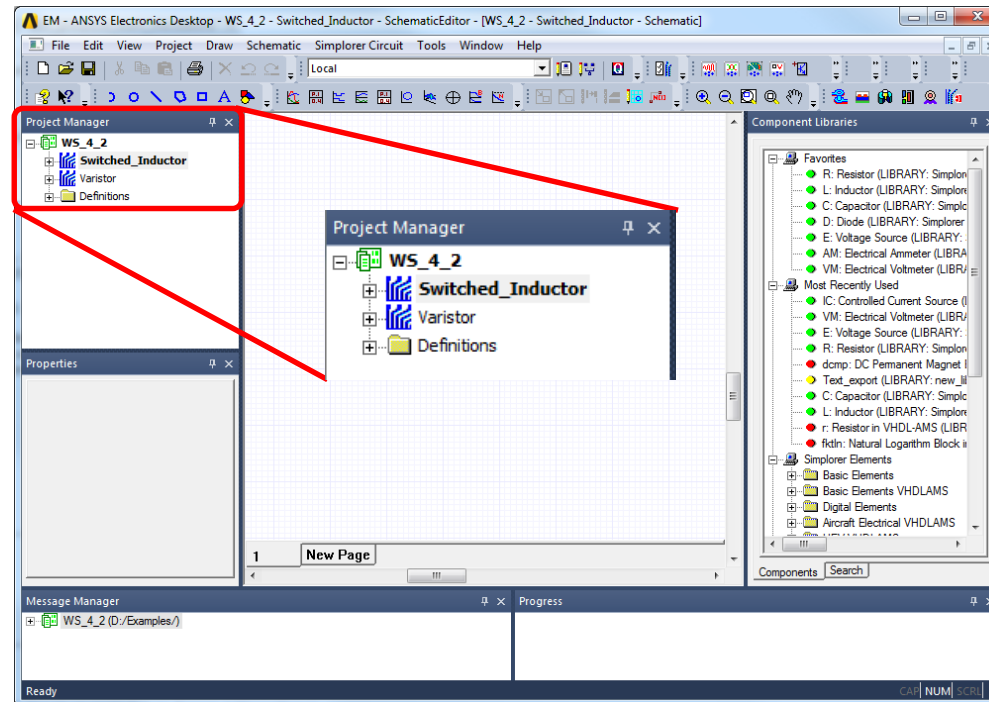
# Varistor as Switch Protection

Introduction to ANSYS Simplorer



# Insert a Simplorer Design

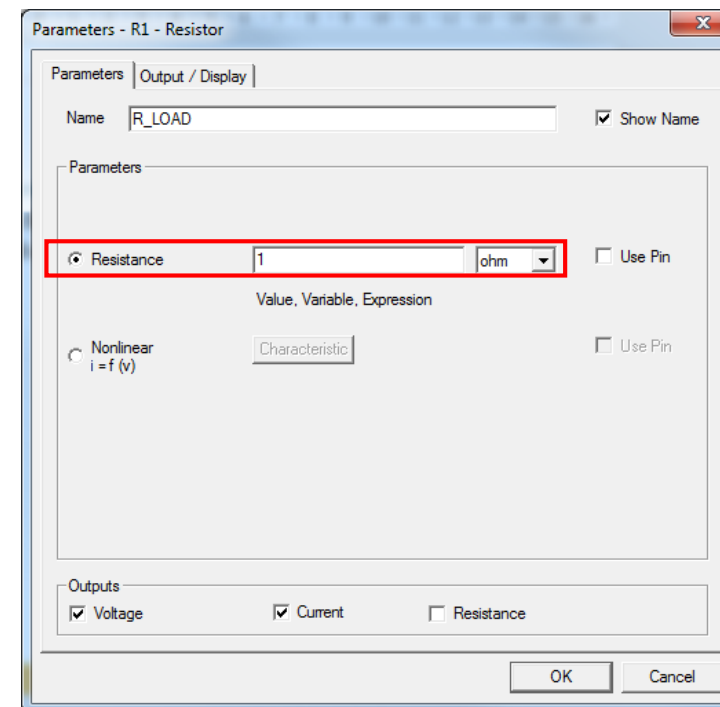
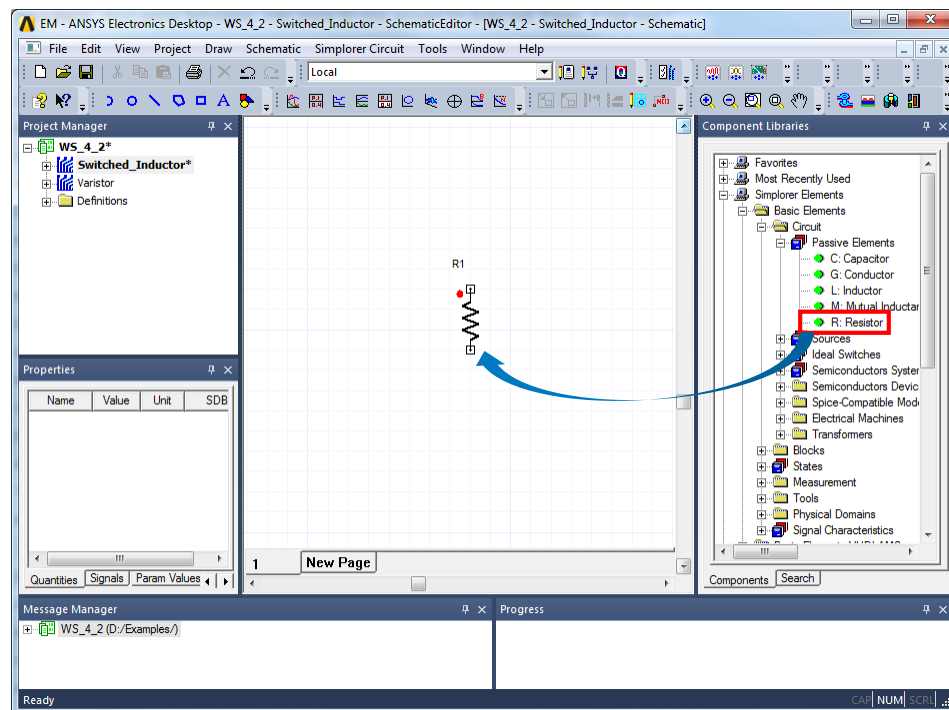
- Insert a Simplorer Design using the icon 
- Rename the Design as **Switched\_Inductor**
- Save again the project using the icon 



# Insert Components

- Resistor

- In Component Libraries window *Simplorer Elements* → *Basic Elements* → *Circuit* → *Passive Elements*
- Select the **Resistor** and drag and drop it into the Schematic. Press **Esc** key to exit the insert mode
- Double click on the **Resistor**, name it **R\_load** and set the value to **1  $\Omega$**

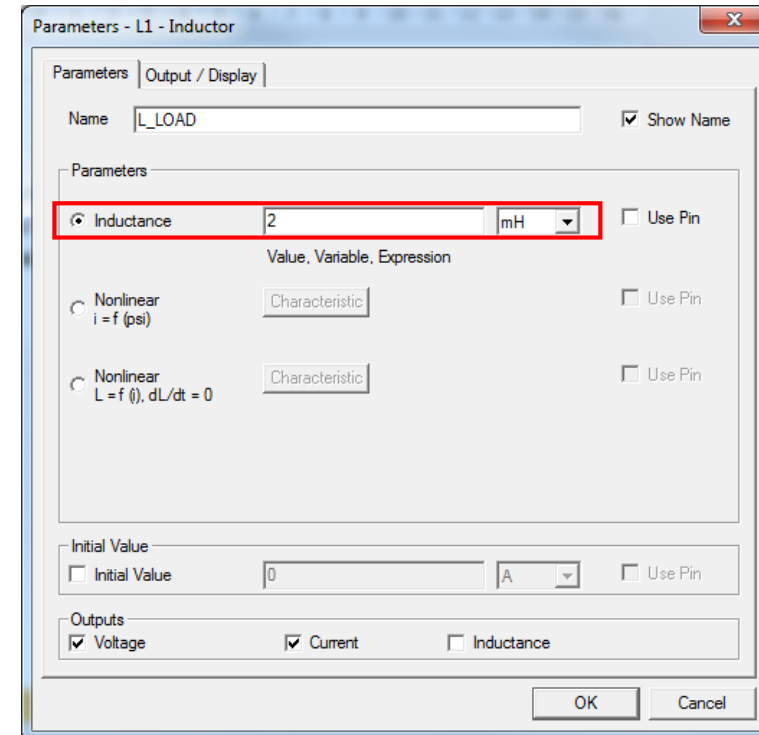
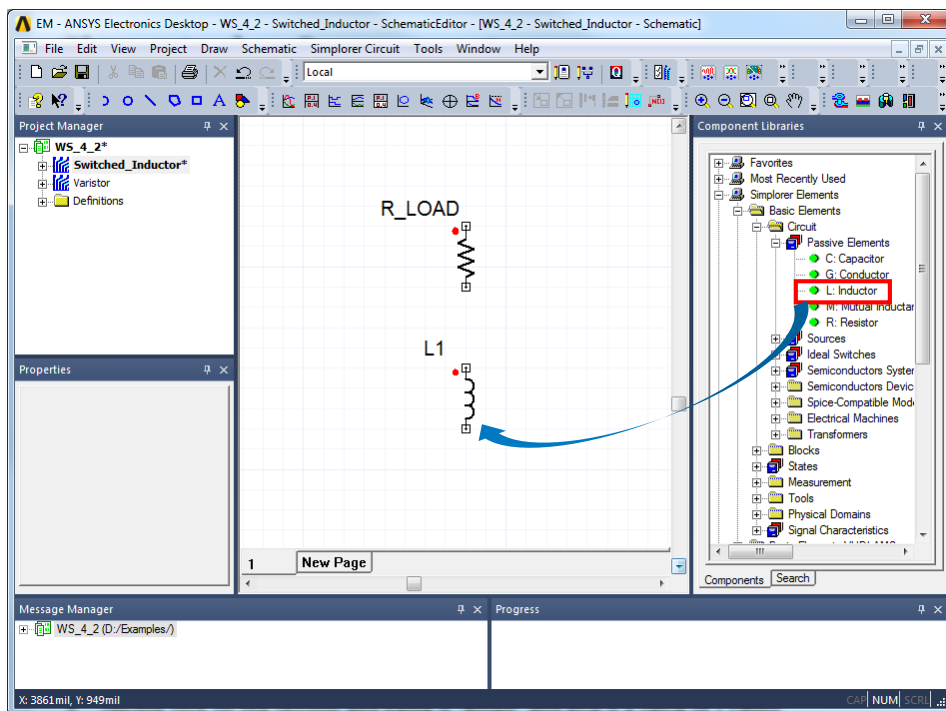




# Insert Components

- Inductor

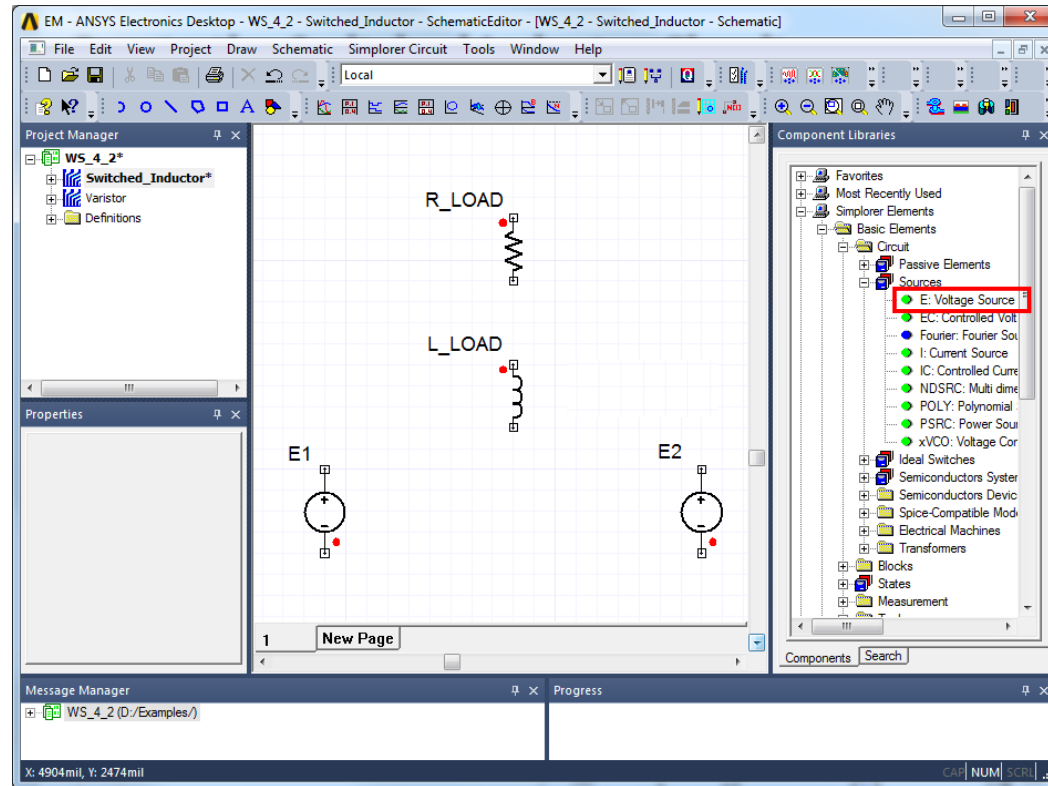
- In Component Libraries window *Simplorer Elements* → *Basic Elements* → *Circuit* → *Passive Elements*
- Select the **Inductor** and drag and drop it into the Schematic. Press **Esc** key to exit the insert mode
- Double click on the **Inductor**, name it **L\_LOAD** and set the value to **2 mH**



# Insert Components

- Voltage Source

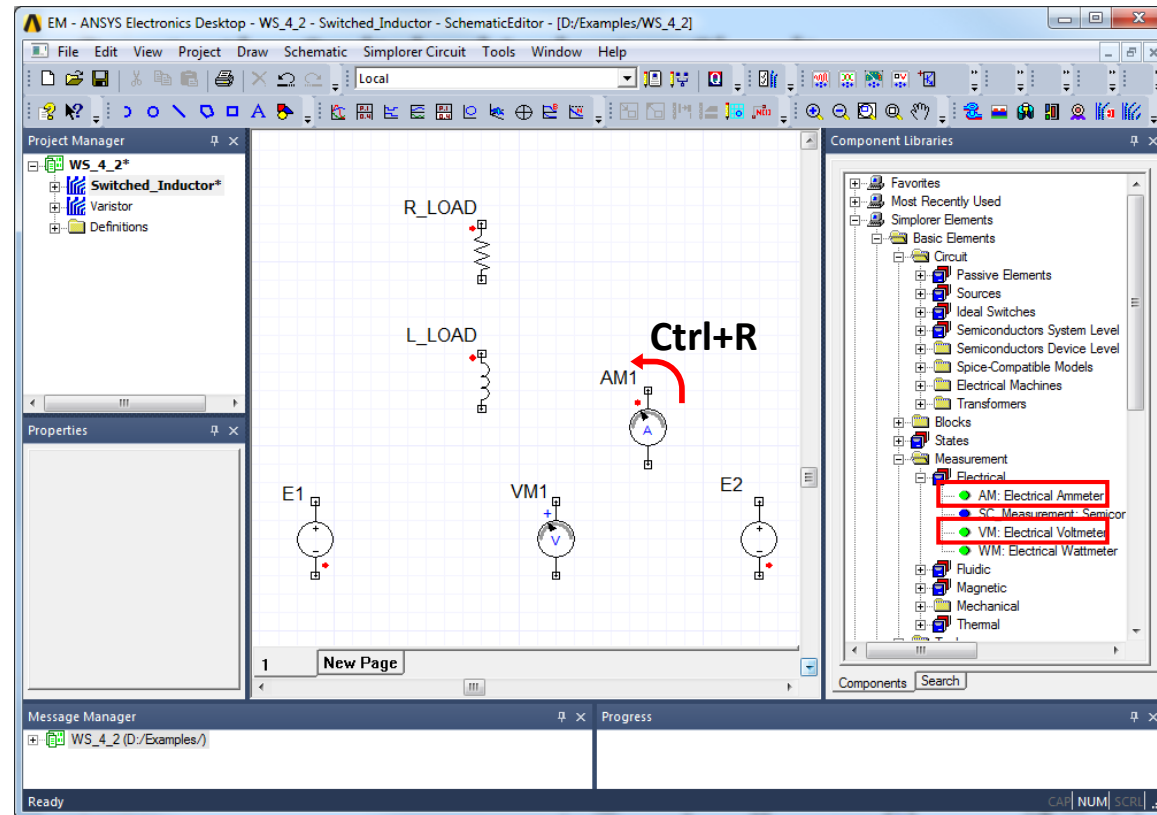
- In Component Libraries window *Simplorer Elements* → *Basic Elements* → *Circuit* → *Sources*
- Select the **E: Voltage Source**, drag and drop it twice into the Schematic. Press **Esc** key to exit the insert mode
- Use the shortcut **Ctrl+D** to fit the view



# Insert Components

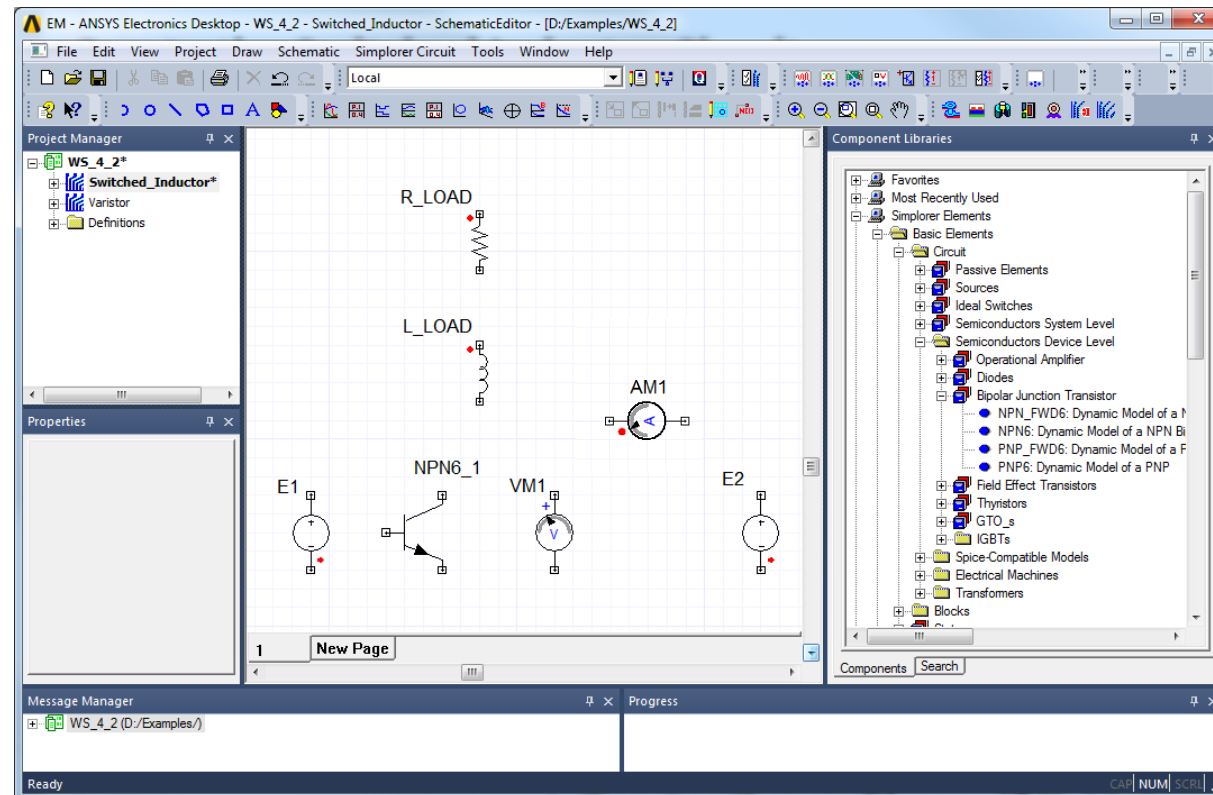
- Voltmeter and Ammeter

- In Component Libraries window *Simplorer Elements* → *Basic Elements* → *Measurements* → *Electrical*
- Select the **VM: Electrical Voltmeter** and drag and drop it into the Schematic
- Select the **AM: Electrical Ammeter** and drag and drop it into the Schematic. Rotate it using **Ctrl+R**
- Press **OK**



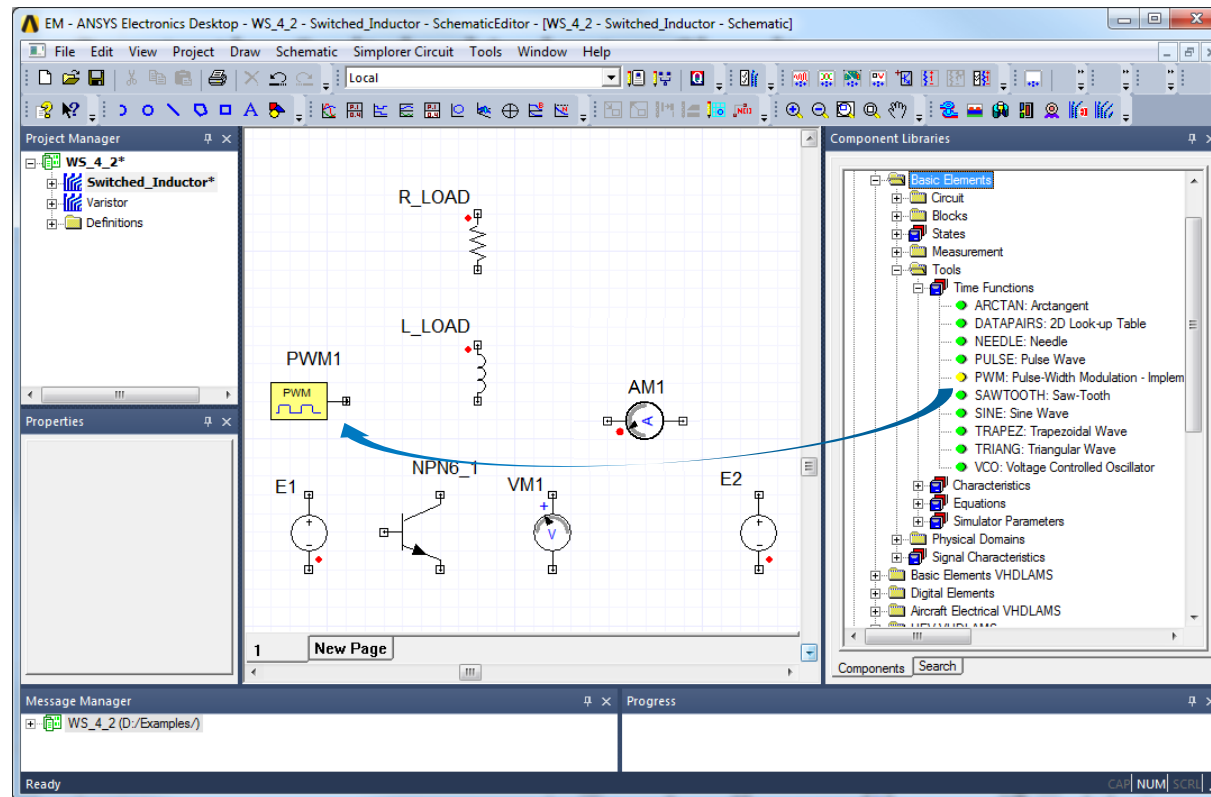
# Insert Components

- BJT
  - In Component Libraries window *Simplorer Elements* → *Basic Elements* → *Circuit* → *Semiconductor Device Level* → *Bipolar Junction Transistor*
  - Select the **NPN6: Dynamic Model of a NPN Bipolar Transistor** and drag and drop it into the Schematic



# Insert Components

- PWM Source
  - In Component Libraries window *Simplorer Elements* → *Basic Elements* → *Tools* → *Time Functions*
  - Select the **PWM: Pulse Width Modulation** and drag and drop it into the Schematic

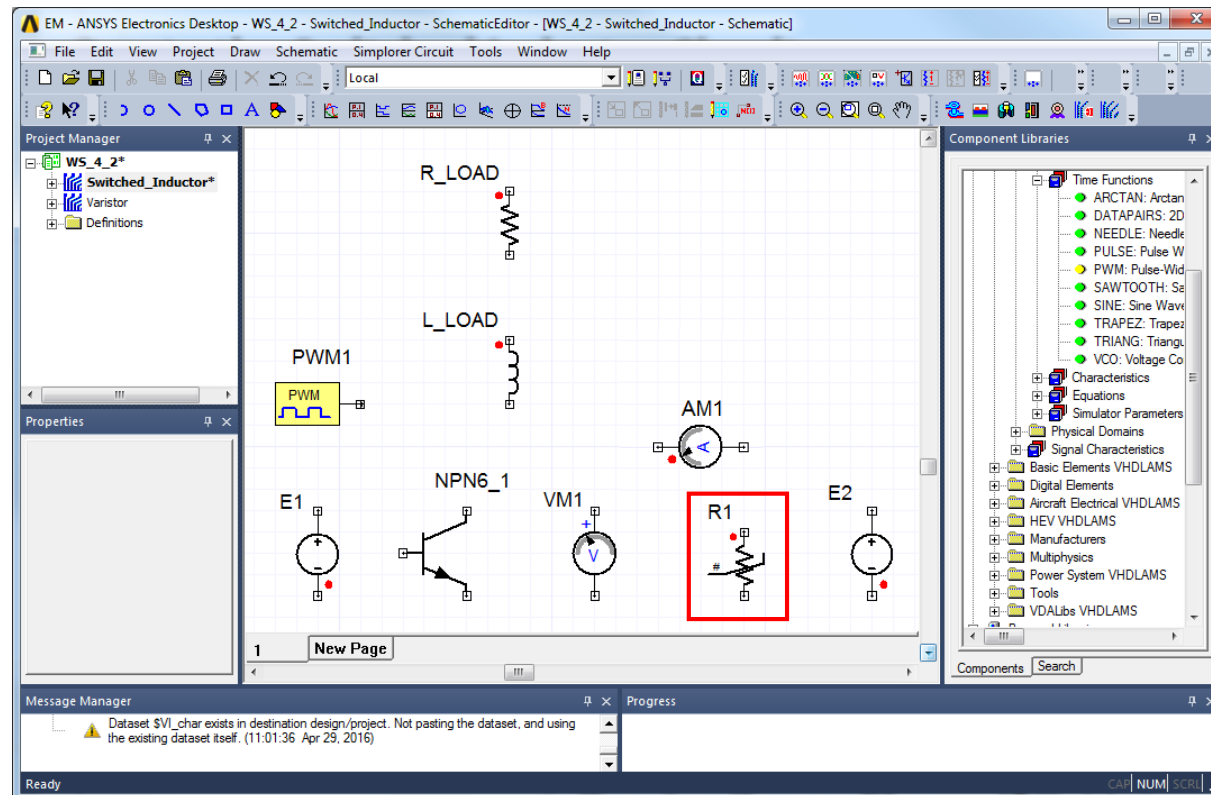




# Insert Components

- **Varistor**

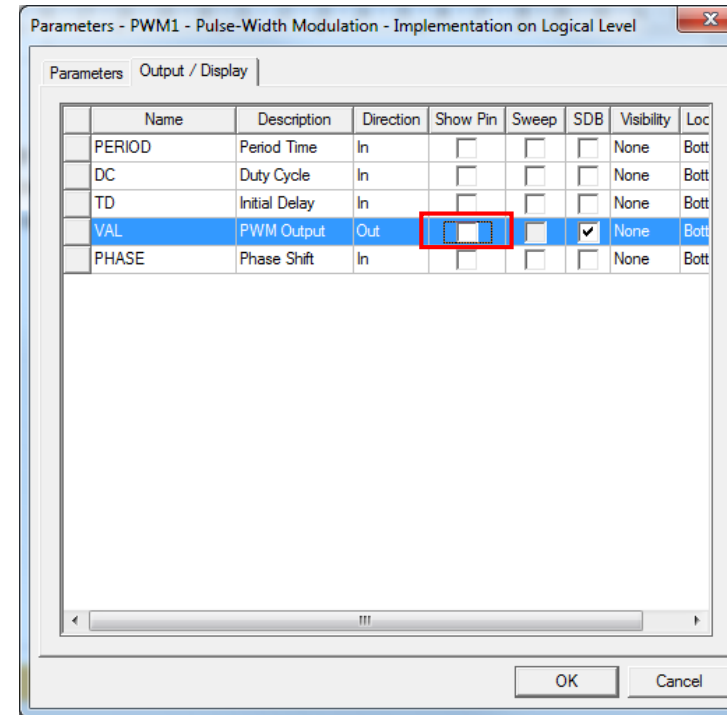
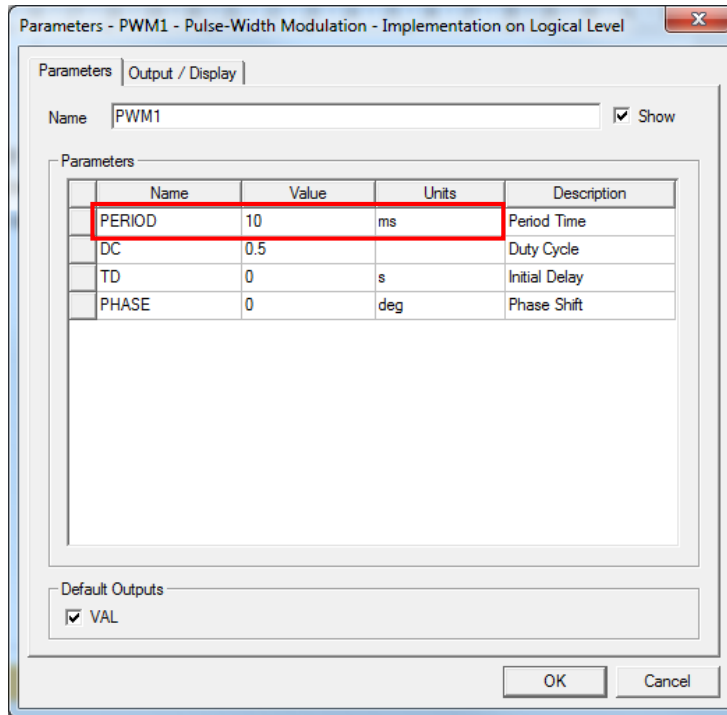
- Add the Varistor model that was created in the previous section by opening the previous design “**Varistor**”, selecting the **R1** component, copying it (**Ctrl+C**), then re-opening the new design “**Switched\_Inductor**” and pasting it (**Ctrl+V**)



# Set Component Properties

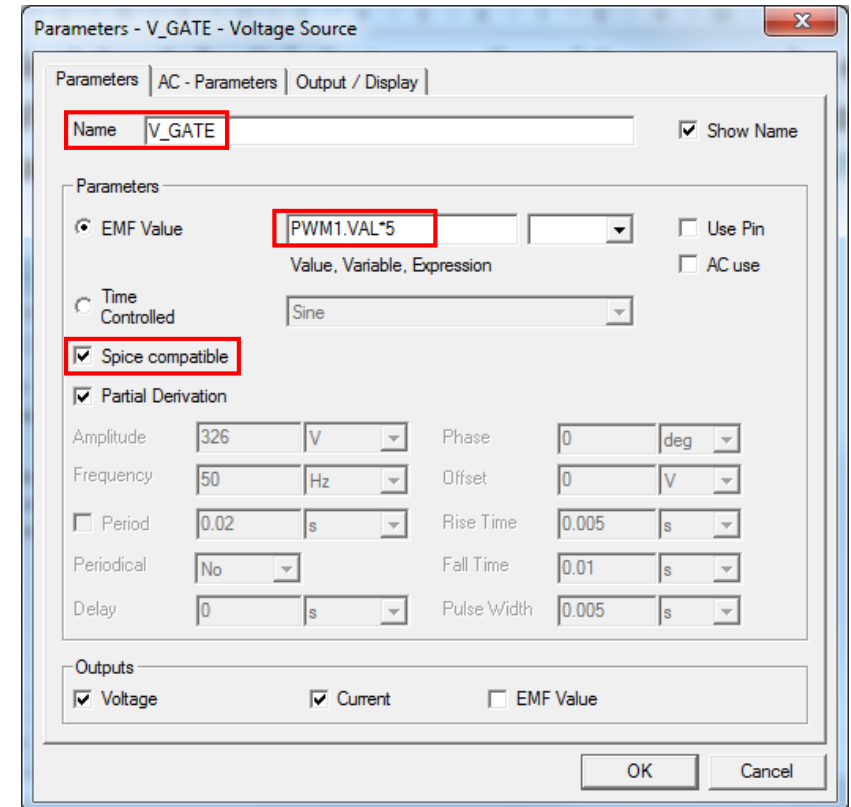
- PWM Time Function

- Double click on the **PWM1** Time Function. Define the period to be **10 ms** and leave the duty cycle to be default of **0.5**
- Select the “Output/Display” tab for the PWM block, and un-check the box “**Show Pin**” for the Output “**VAL**” (this will remove the output pin from the symbol)
- Press **OK**



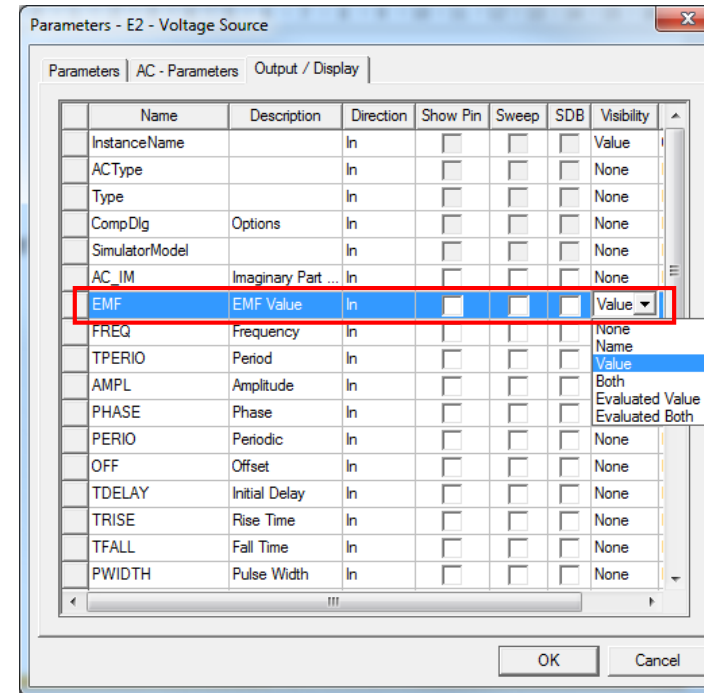
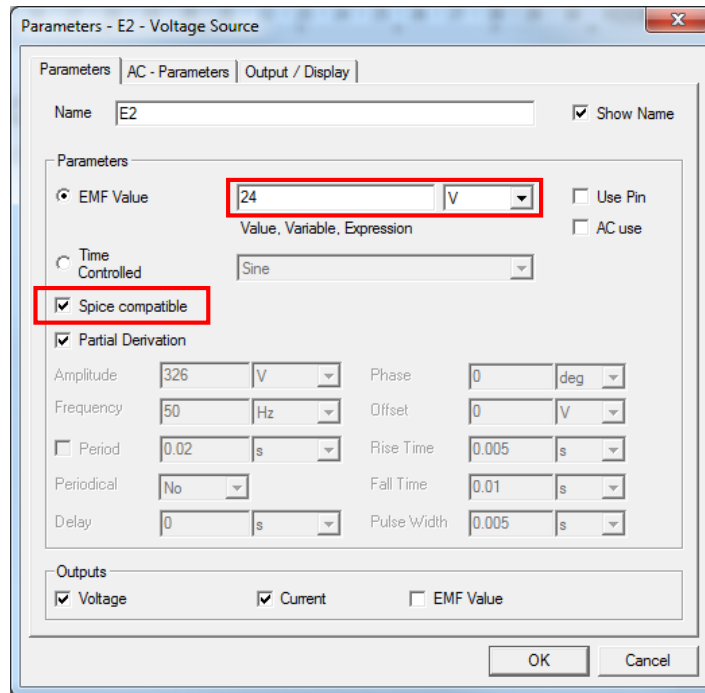
# Set Component Properties

- **E1 Voltage Source**
  - The **E1 Voltage Source** is supposed to drive the BJT, providing a sequence of pulses coming from the PWM time function block
  - Since the output of the PWM block by default is between 0 and 1, may not be enough to turn on the BJT's gate
  - The output of the PWM block will then be scaled to 0 to 5V in the Voltage Source Generator
  - Double click on the **E1 Voltage Source** and name it **V\_GATE**
  - Set the EMF value to be **PWM1.VAL\*5**
  - Select **Spice Compatible**
  - Press **OK**




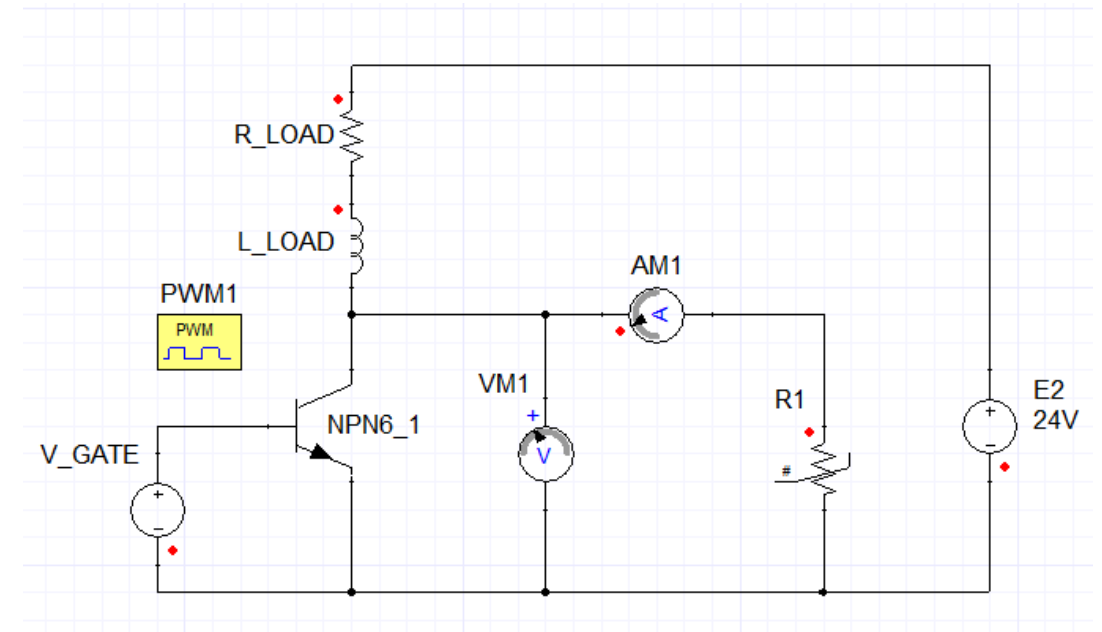
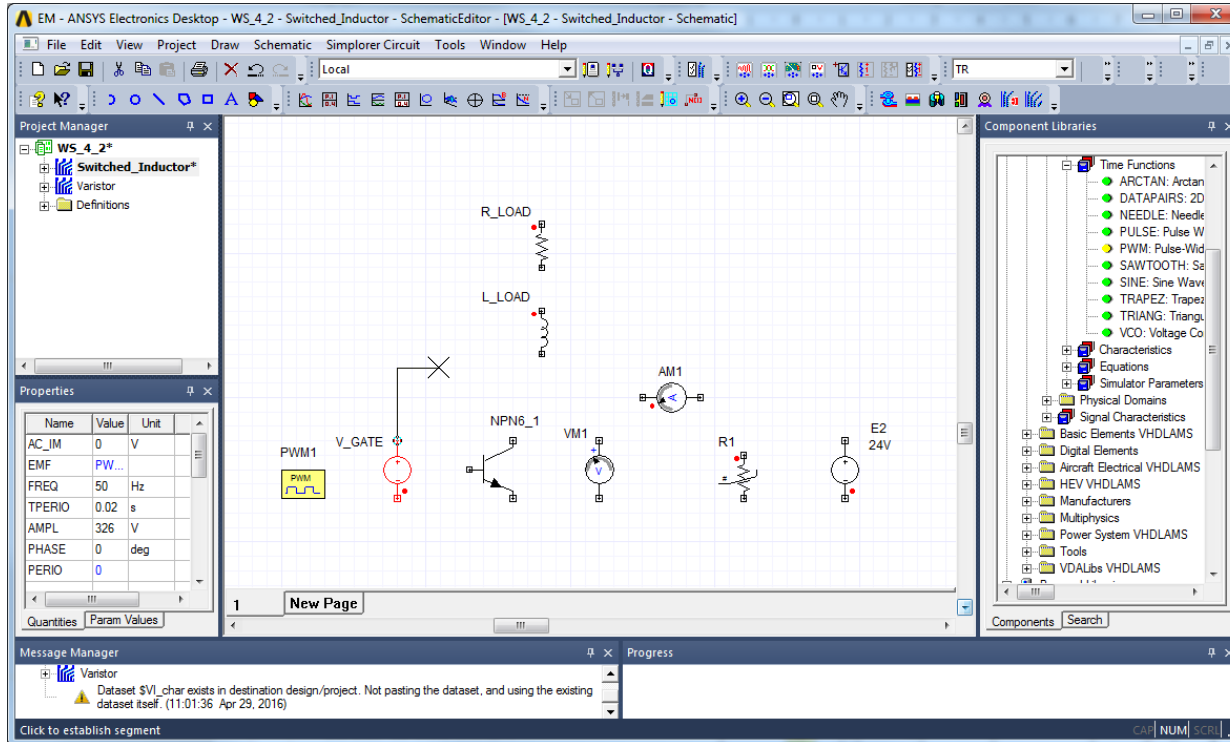
# Set Component Properties

- E2 Voltage Source
  - Double click on the **E2 Voltage Source**
  - Set the EMF value to be **24 V**
  - Select **Spice Compatible**
  - In the **Output/Display Tab** under Visibility, select Value for **EMF**
  - Press **OK**



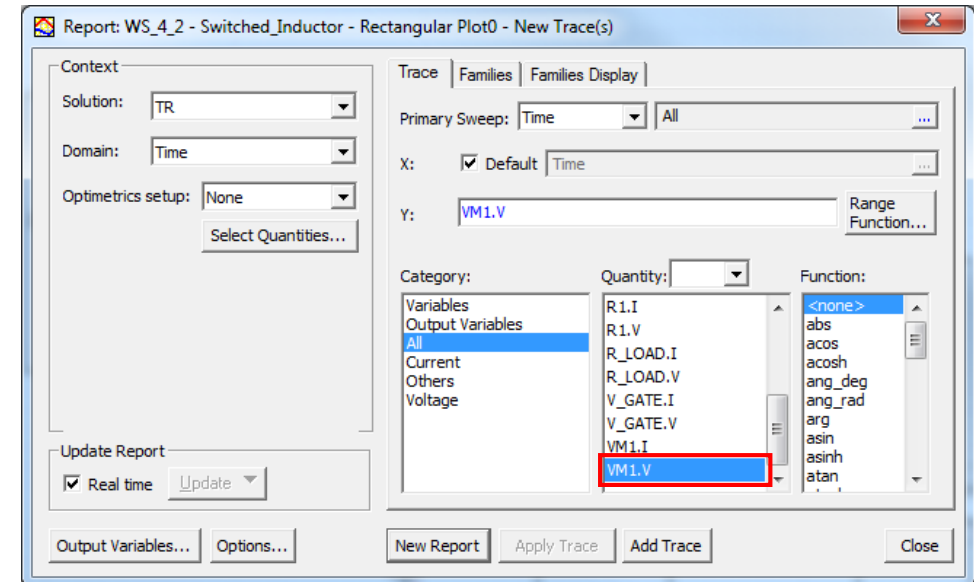
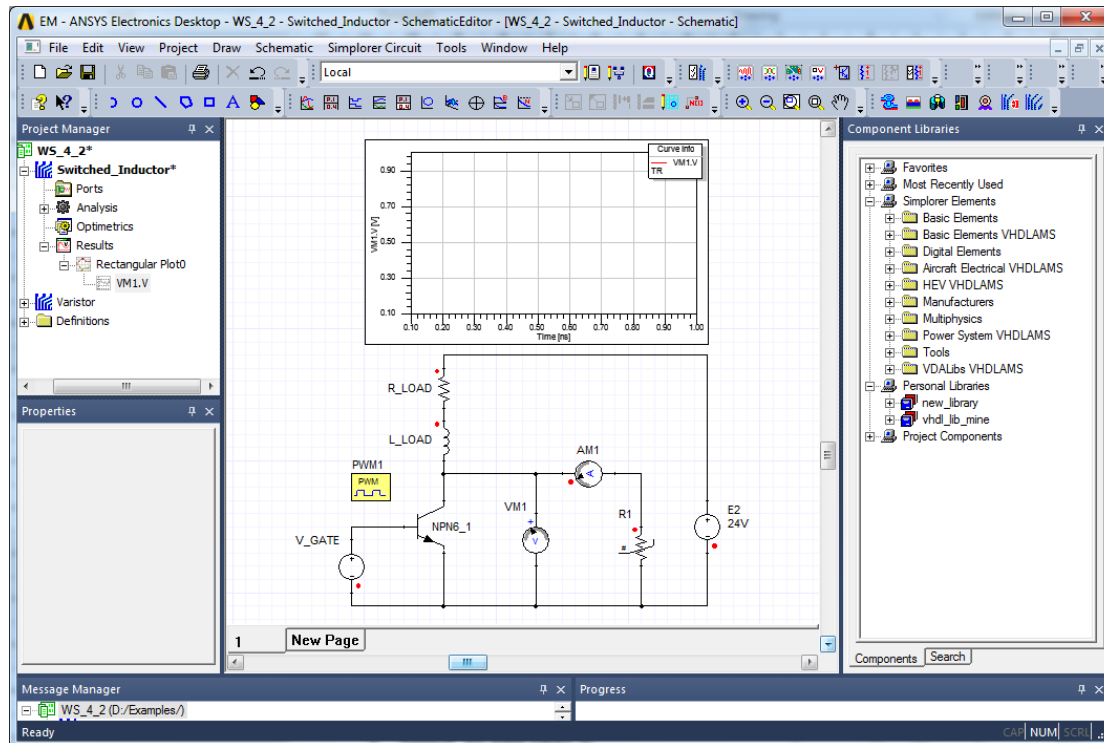
# Connect Components

- Place the mouse over one terminal of a component. The mouse pointer changes its shape becoming a cross. Press the **LMB** and move the cross till the connecting terminal of next component
- Add the **Ground node** clicking the icon  and placing it into the Schematic
- Connect all the components till completing the circuit as in figure



# Prepare the Postprocessing

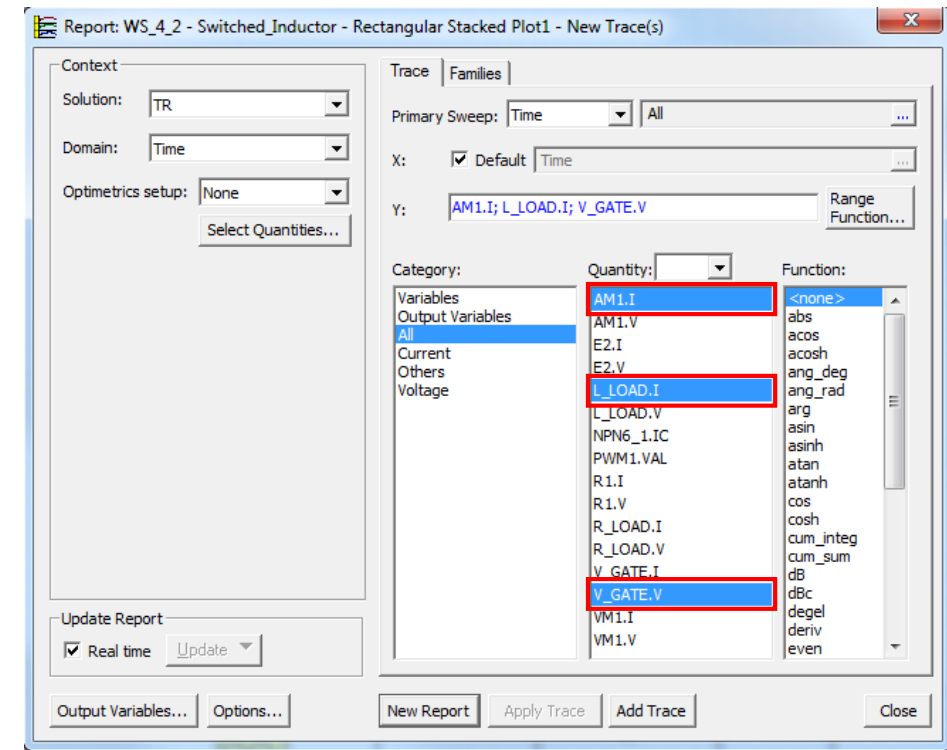
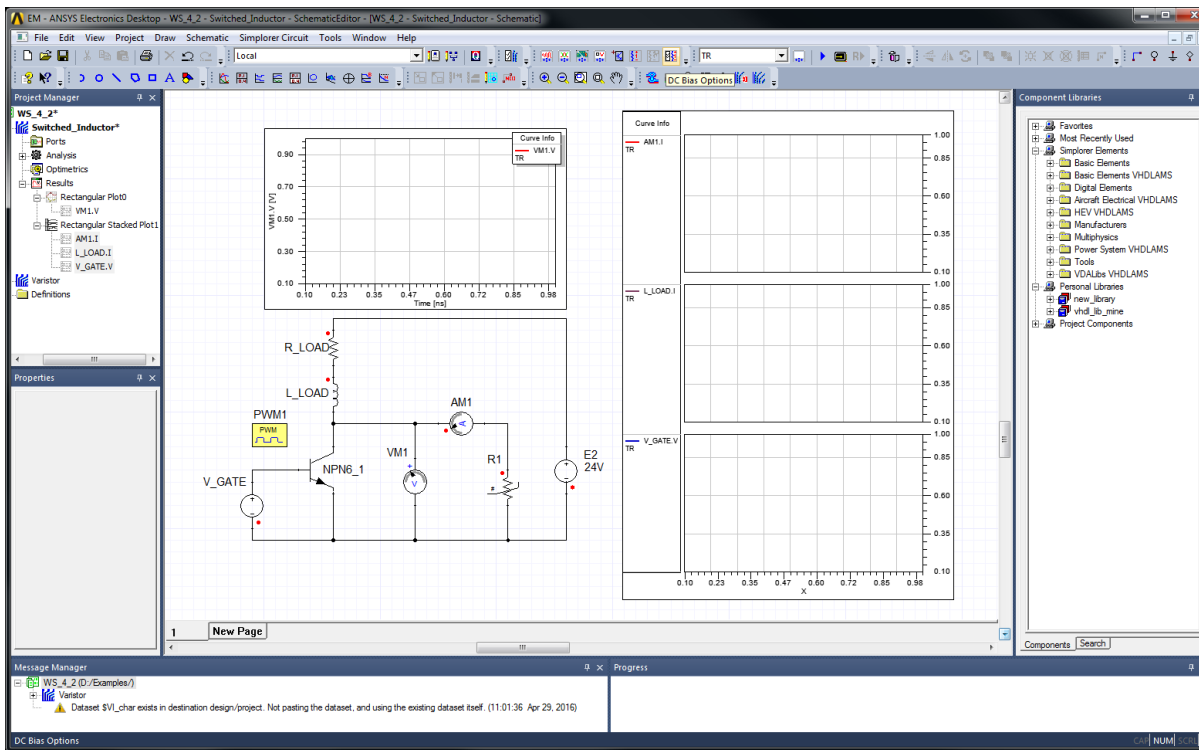
- Select the menu item **Draw** → **Report** → **Rectangular Plot** and place the plot in the Schematic
- Automatically the **New Trace** window pops-up
- Select the Voltmeter signal by selecting the quantity **VM1.V**
- Click on the **Add Trace** button and then **Close**





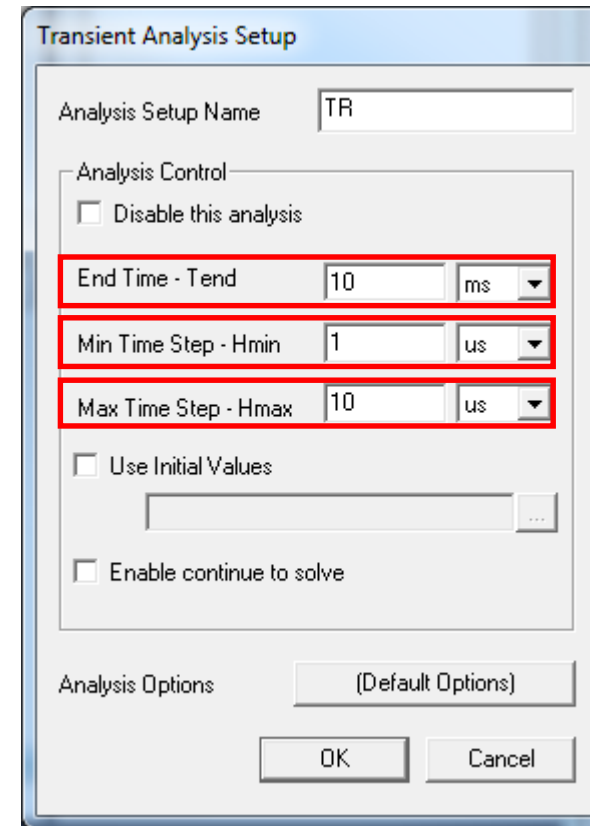
# Prepare the Postprocessing

- Select the menu item **Draw** → **Report** → **Rectangular Stacked Plot** and place it in the Schematic
- Automatically the **New Trace** window pops-up
- Select the 3 signals **V\_GATE.V**, **AM1.I** and **L\_LOAD.I**
- Click on the **Add Trace** button and then **Close**



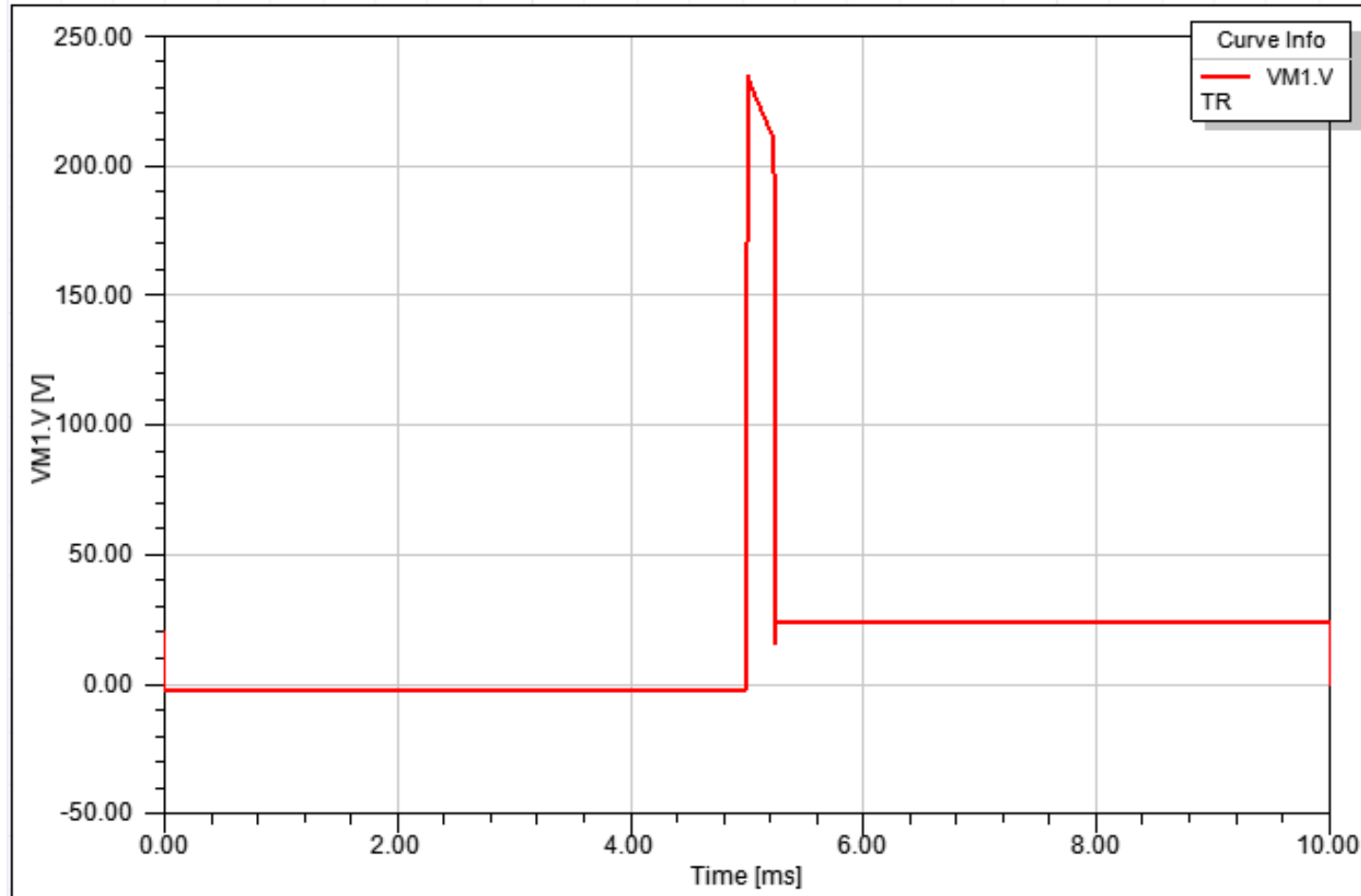
# Setup the Transient Analysis

- In the Transient Analysis Setup window:
  - Tend: 10 ms
  - Hmin: 1  $\mu$ s
  - Hmax: 10  $\mu$ s
  - Press OK



# Analyze and View Results

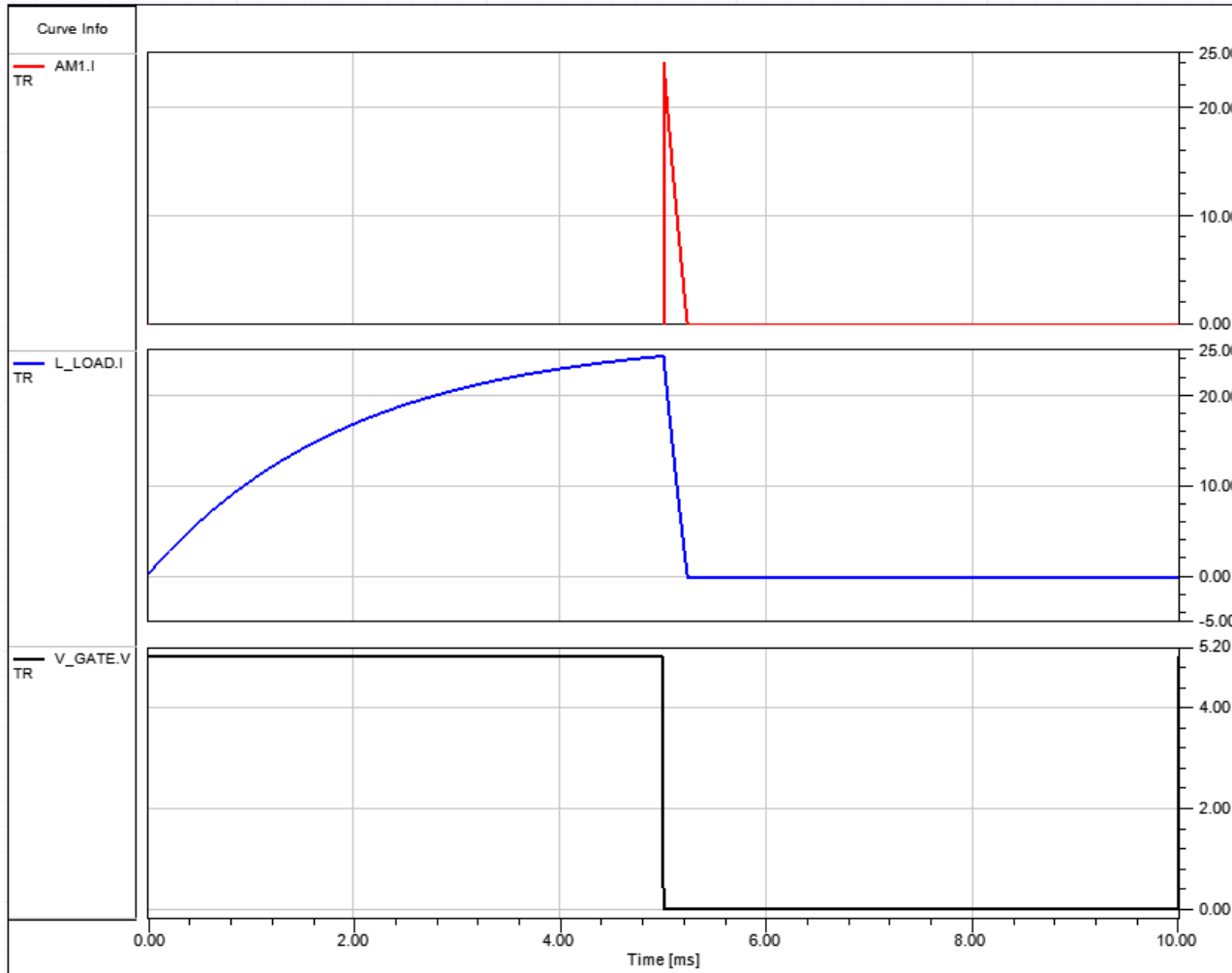
- From the Project Manager window, select the **Analysis** → **TR** → **RMB** → **Analyze** to run the simulation
- The final result should look very similar to the following Figure:



*Note: Use RMB on the plot area and then “Edit in Place” to modify colors and thickness of plotted lines*

# View Results

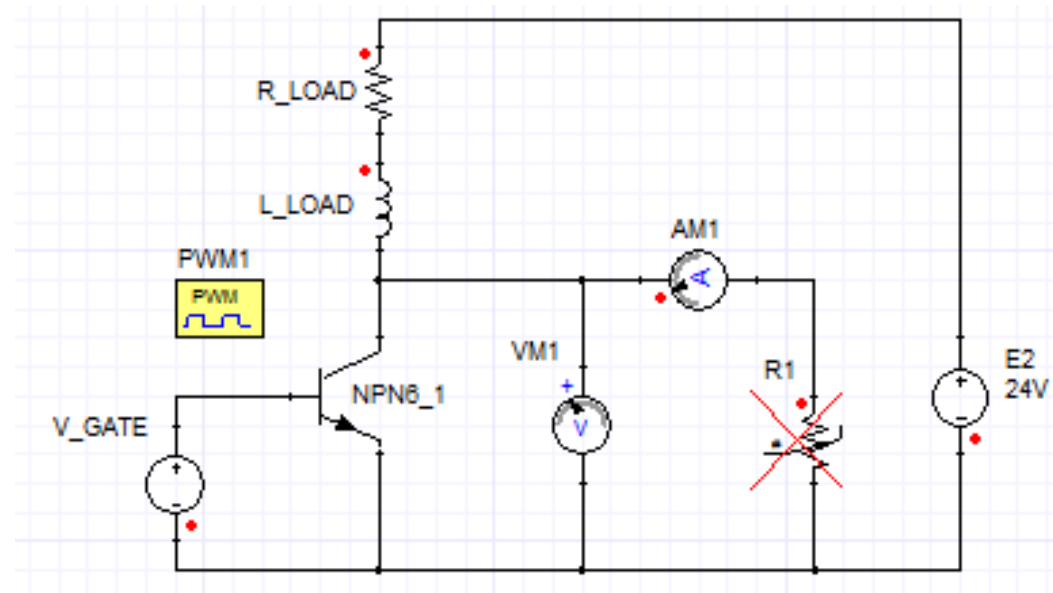
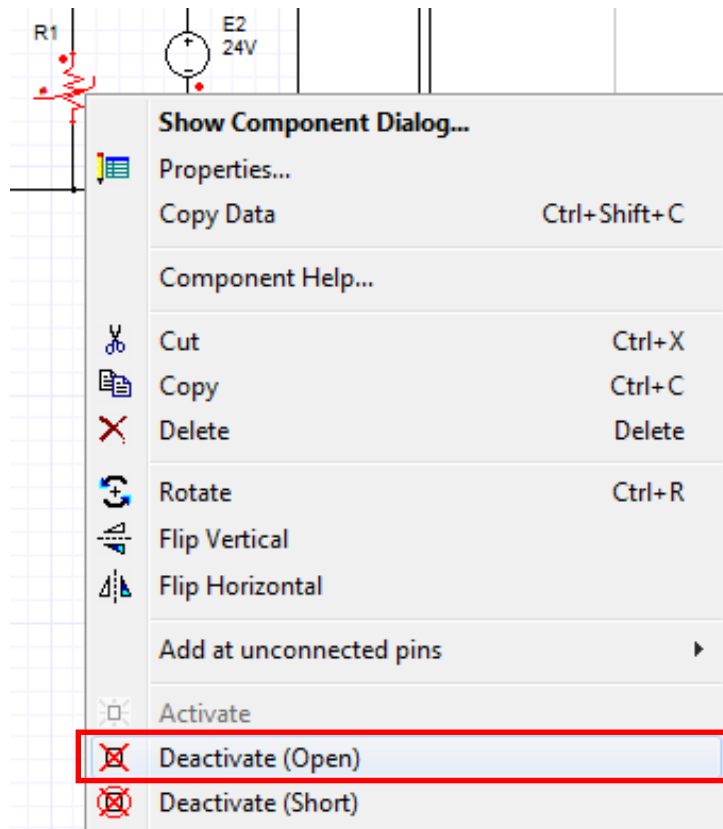
- The rectangular stacked plot should look similar to the following one:



**Note from the results, when the switch is turned off, the Inductor current will decrease rapidly yielding the voltage spike across the switch. Note the Varistor in this case will provide a path for the inductor current during turn off to limit the voltage spike magnitude to under 250V.**

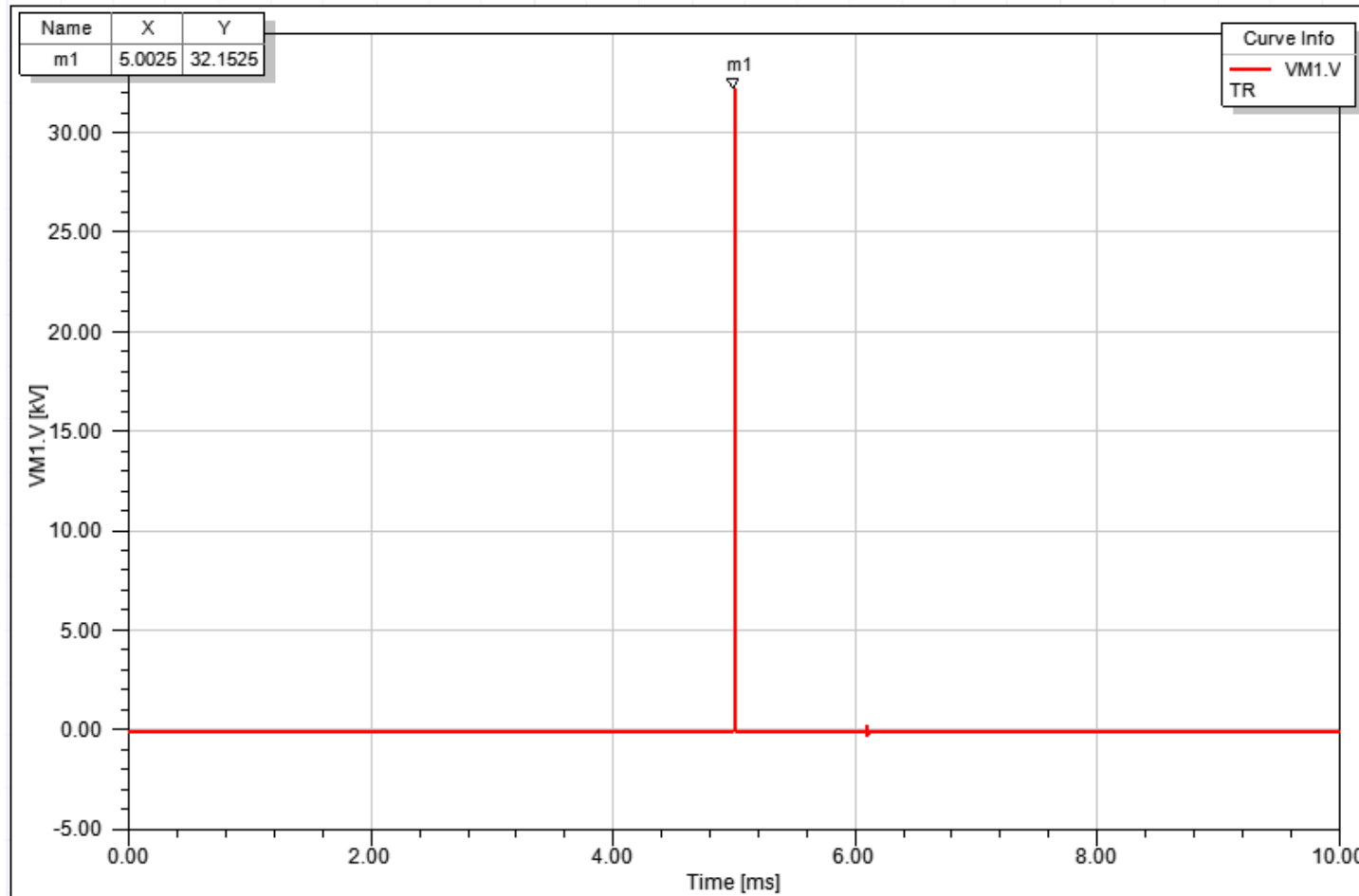
# Further Analysis

- In order to better analyze the effectiveness of the Varistor presence, it is enough to de-activate it, run again the simulation and compare the obtained results with the previous ones
- To deactivate the Varistor, **RMB** on **R1** and **Deactivate Open**



# Analyze and View Results

- From the Project Manager window, select the **Analysis** → **TR** → **RMB** → **Analyze** to run the simulation
- The new results should look very similar to the following ones:

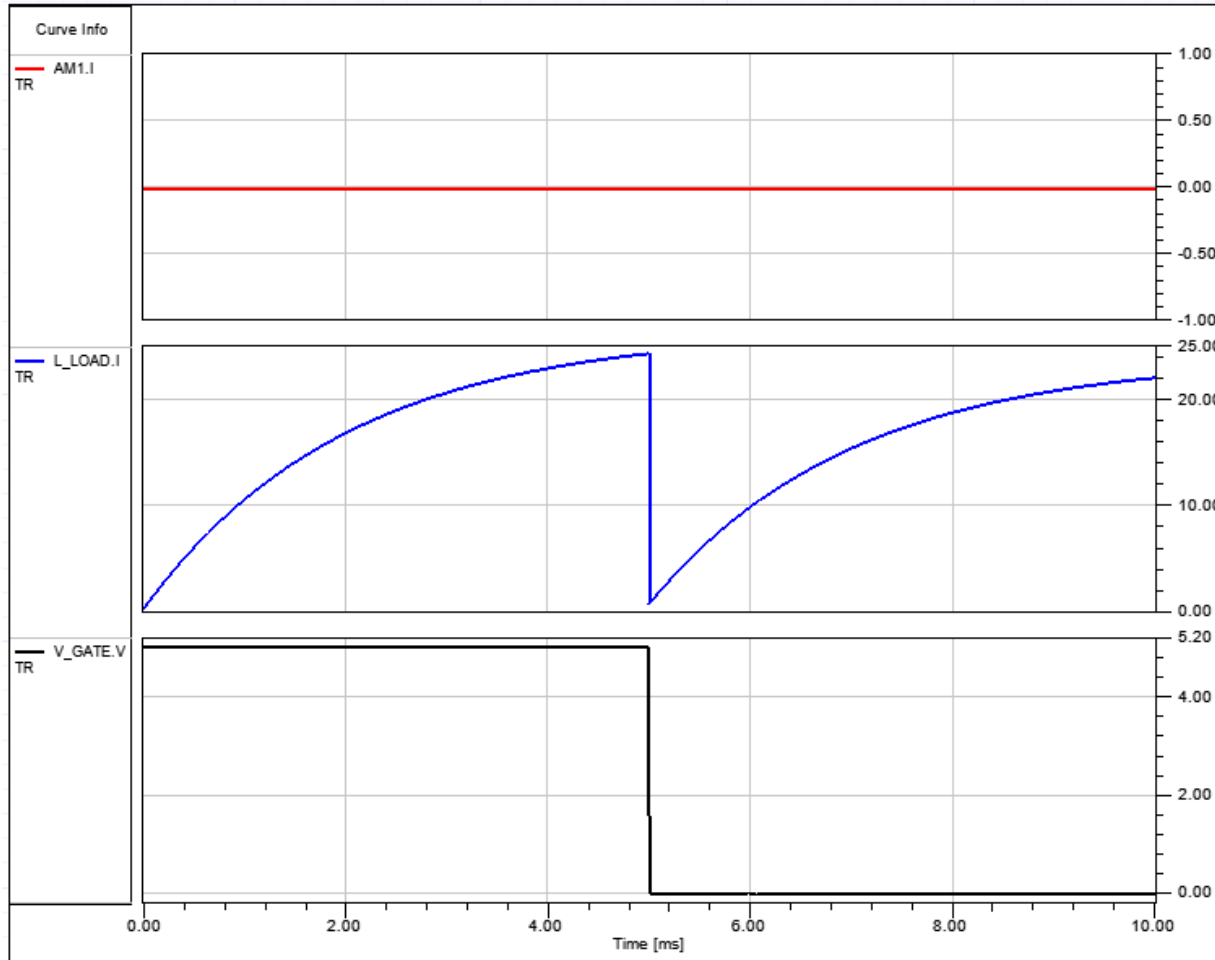


Note the spike behavior and the maximum value reaching in the present case around **32 kV**, more than enough to definitely damage the BJT

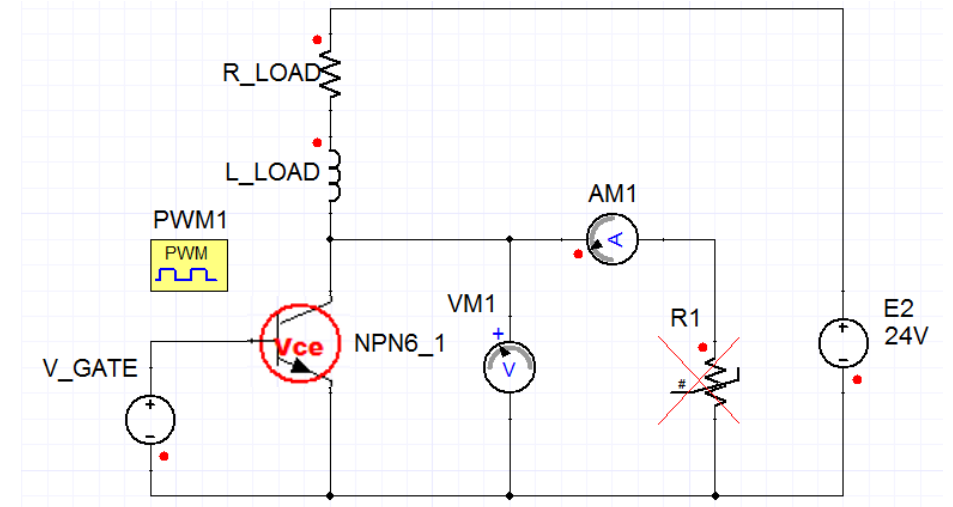


# View Results

- The rectangular stacked plot should look similar to the following one:



**Note the inductor current starts to ramp up again even though the switch is being turned off ( $V_{GATE} = 0$ ), this is a result of the large voltage spike across the BJT causing it to break down and turn on again**



# Saving the Project

- This completes the workshop
- Save the file with the name **WS\_4\_2** in the working folder