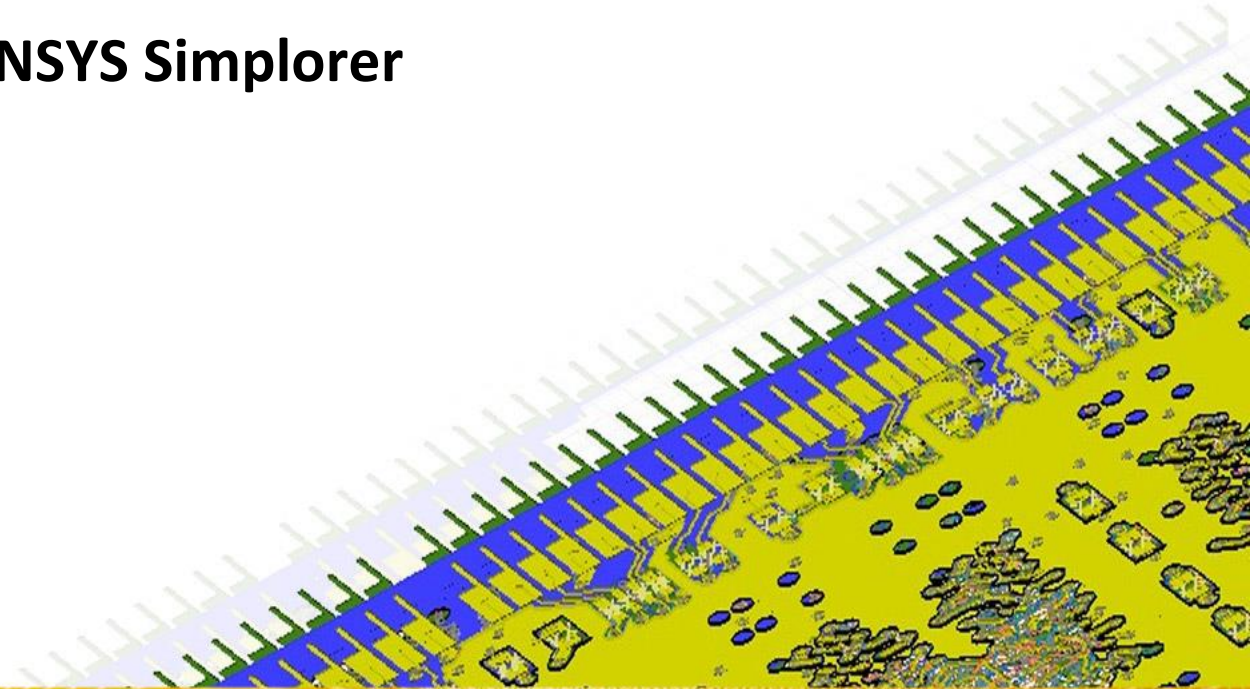




# Workshop 1.1: Switching circuit

Introduction to ANSYS Simplorer



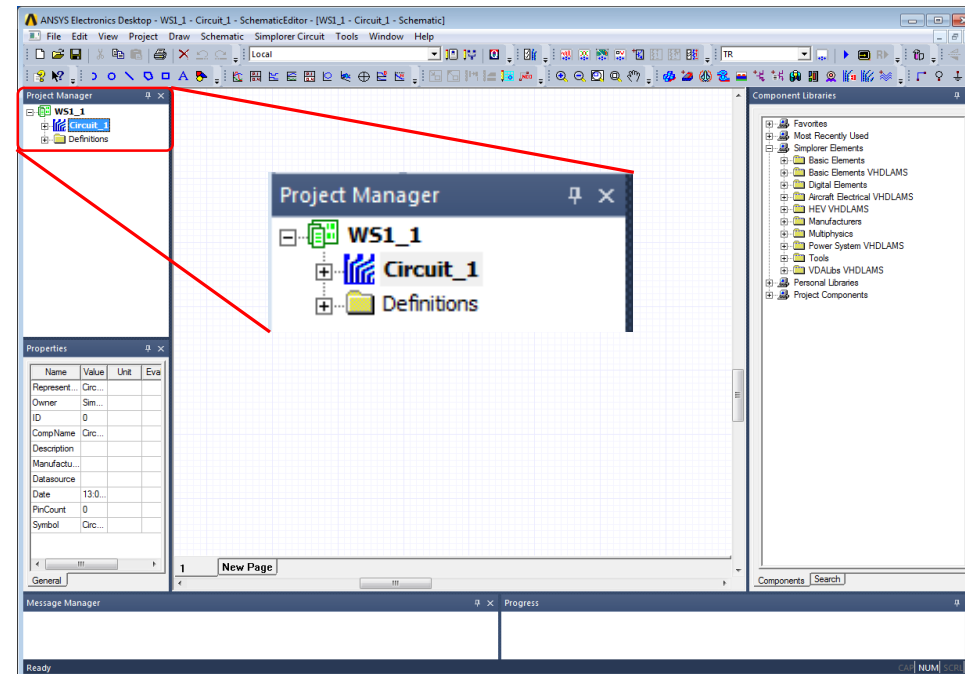
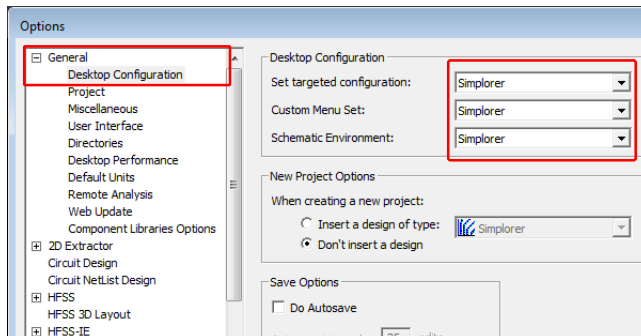
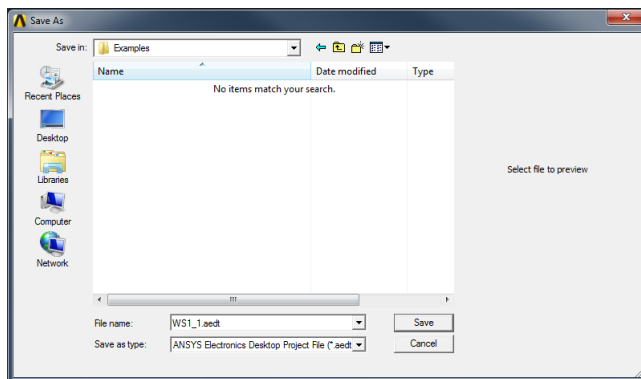
# Overview

- **Switching Circuit**

- In this example we will build a simple DC/DC switching converter feeding an R-L passive load. In particular we will learn:
  - How to place several components from different libraries in the schematic
  - How to connect all the components in order to create a complete circuit
  - How to control the closing/opening behavior of a power electronic switch
  - How to set-up the simulation analysis and to run the simulation
  - How to view the most important and interesting resulting signals/quantities
- Additionally we will set-up a parametric sweep and we will investigate the device behavior depending on parameter values

# Insert a Simplorer Design

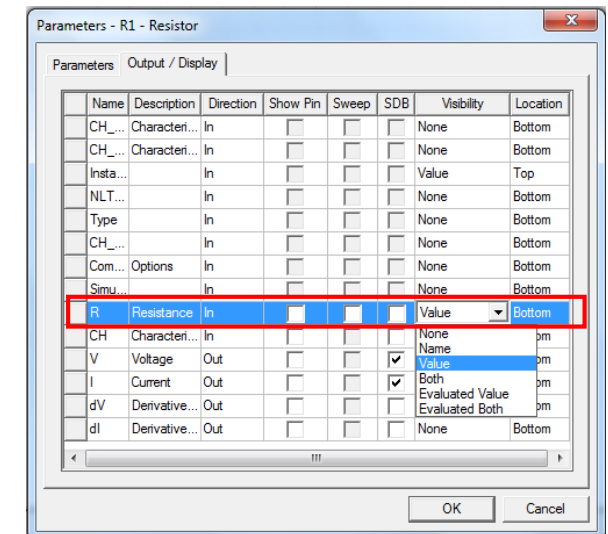
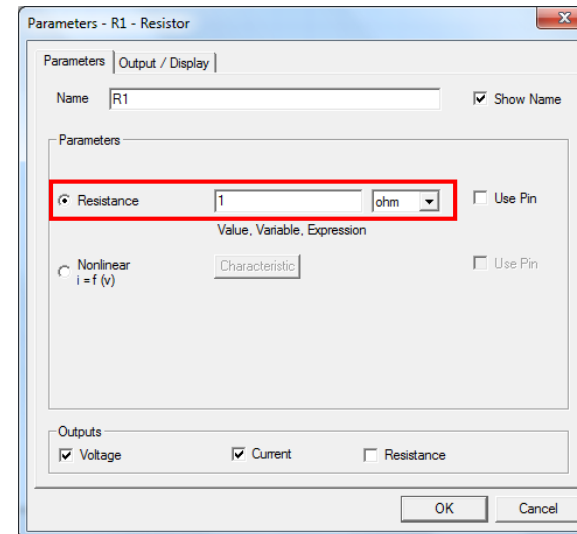
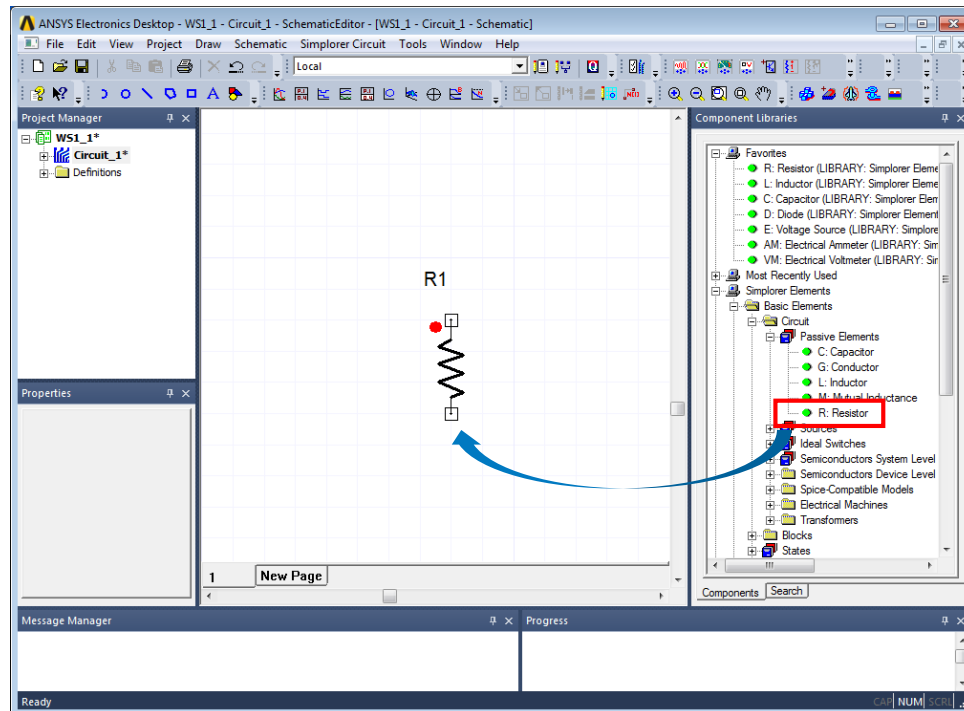
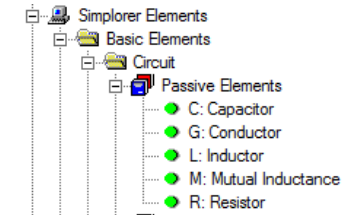
- **Launch the Electronics Desktop 2016**
  - Save the Project as **WS1\_1.aedt**. Go to menu item **Tools** → **Options** → **General Options General** → **Desktop Configuration**. Set all the fields to **Simplorer** and press OK
  - Insert a Simplorer Design using the icon  and rename the Design as **Circuit\_1**
  - Save again the project using the icon 



# Insert Components

- Resistor

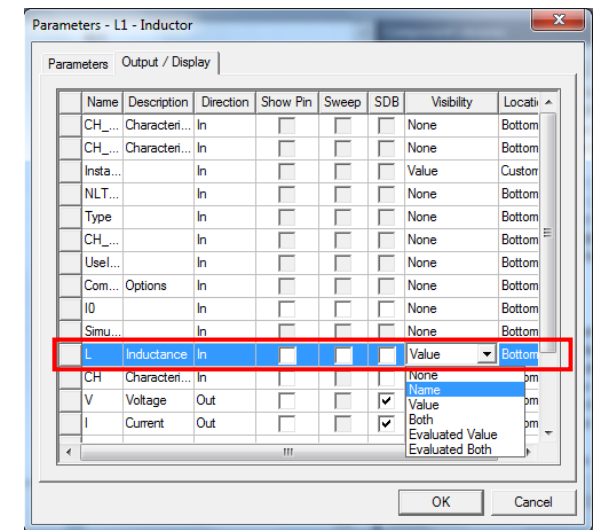
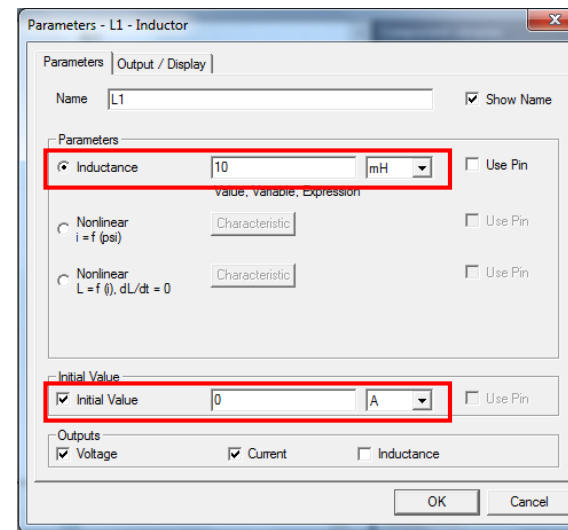
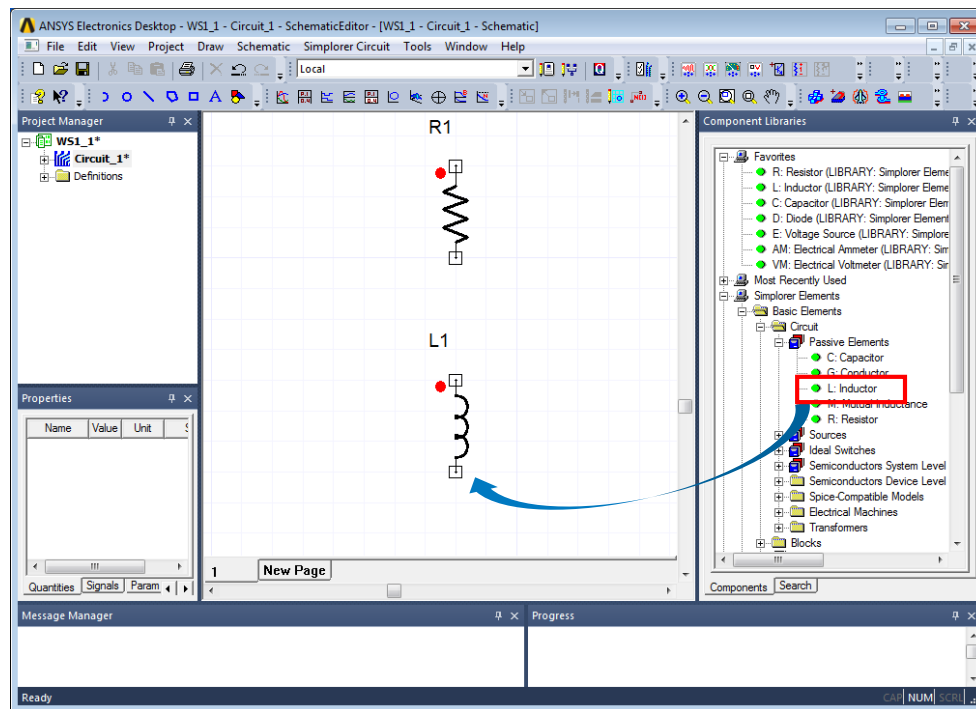
- In Component Libraries window *Simplorer Elements* → *Basic Elements* → *Circuit* → *Passive Elements*
- Select the **R: Resistor** component, drag and drop it into the Schematic
- Press **Esc** key to exit the insert mode
- Double click on the Resistor, change the value to **1  $\Omega$**
- In the **Output/Display Tab** under Visibility, select Value for Resistance



# Insert Components

- Inductor

- Select the **L: Inductor** component, drag and drop it into the Schematic
- Press **Esc** key to exit the insert mode and use the shortcut **Ctrl+D** to fit all
- Double click on the **Inductor**, change the value to **10 mH** and the initial value to **0 A**
- In the **Output/Display Tab** under Visibility, select Value for Inductance

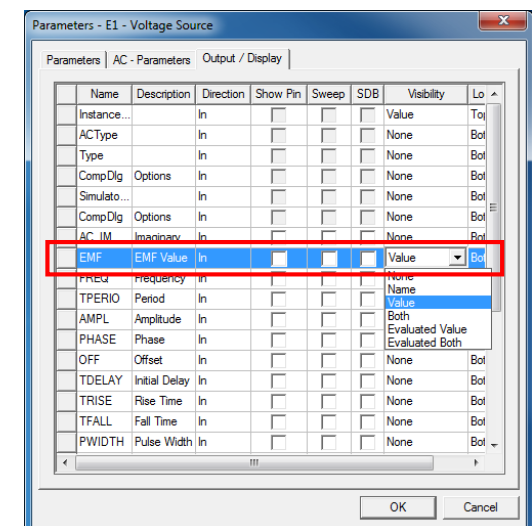
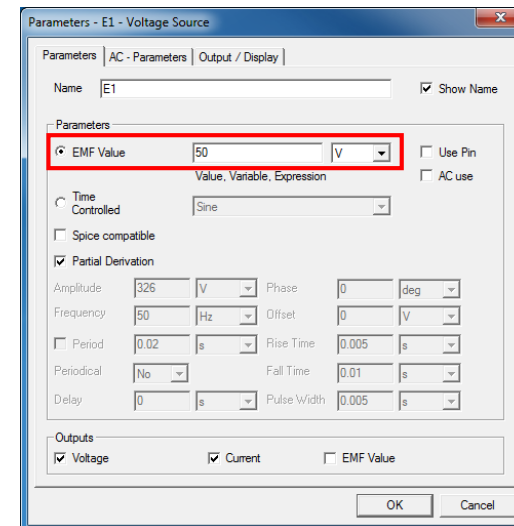
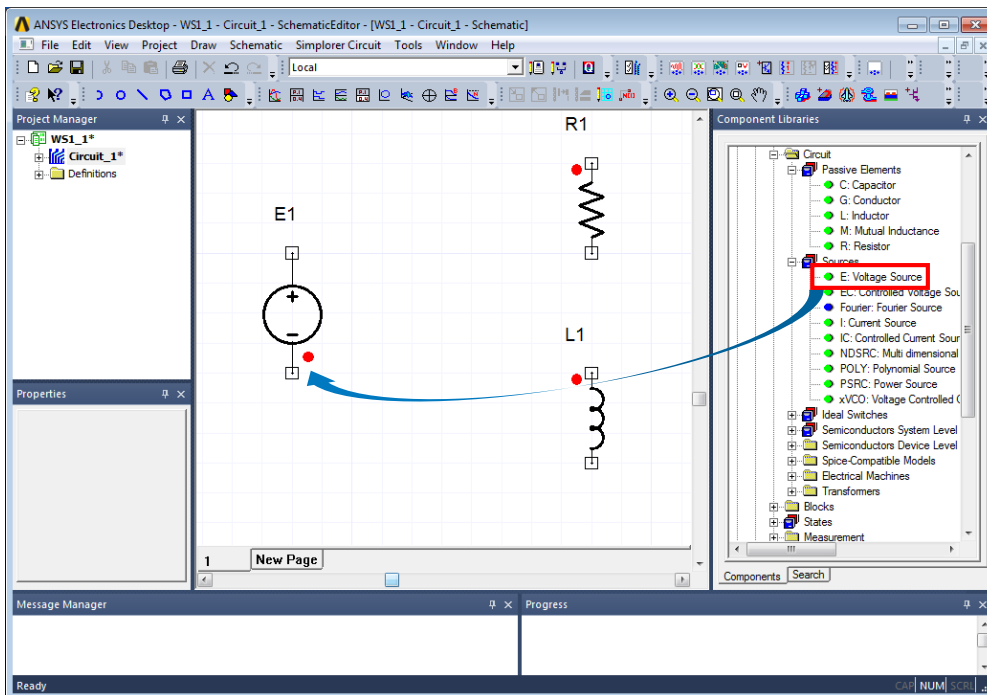




# Insert Components

- 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 and use the shortcut **Ctrl+D** to fit all
- Double click on the **Voltage Source** and change the value to **50 V**
- In the **Output/Display Tab** under Visibility, select Value for **EMF**

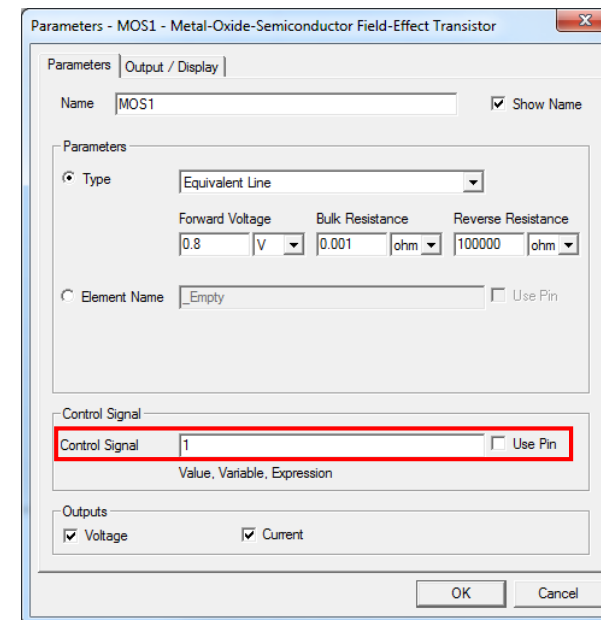
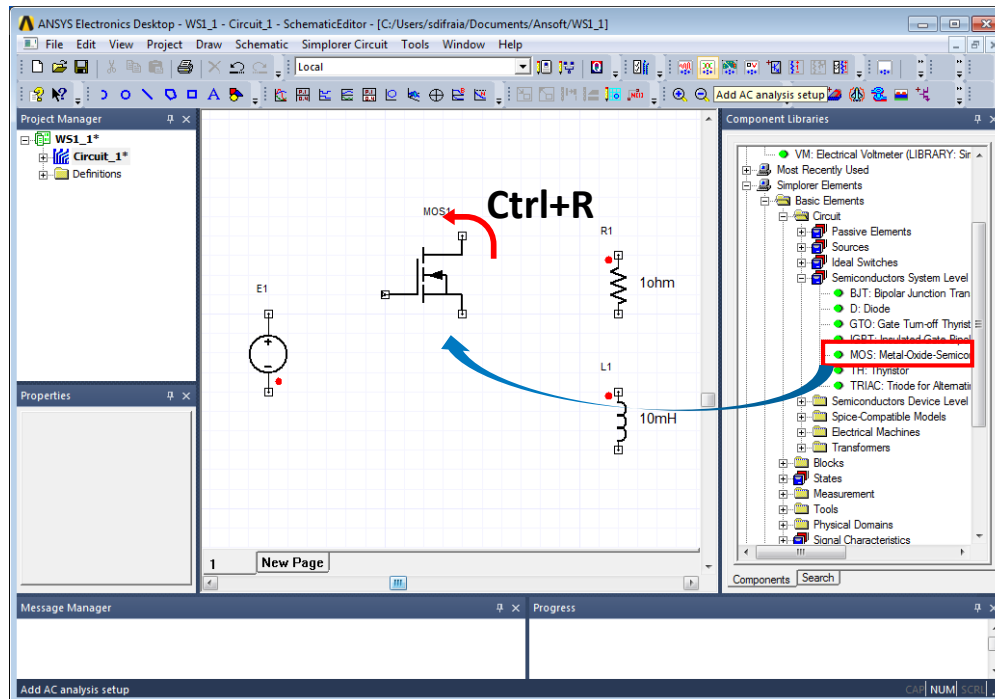


*Note: it is good practice to select “Spice compatible” when inserting voltage sources*

# Insert Components

- **Electronics Switch (Mosfet)**

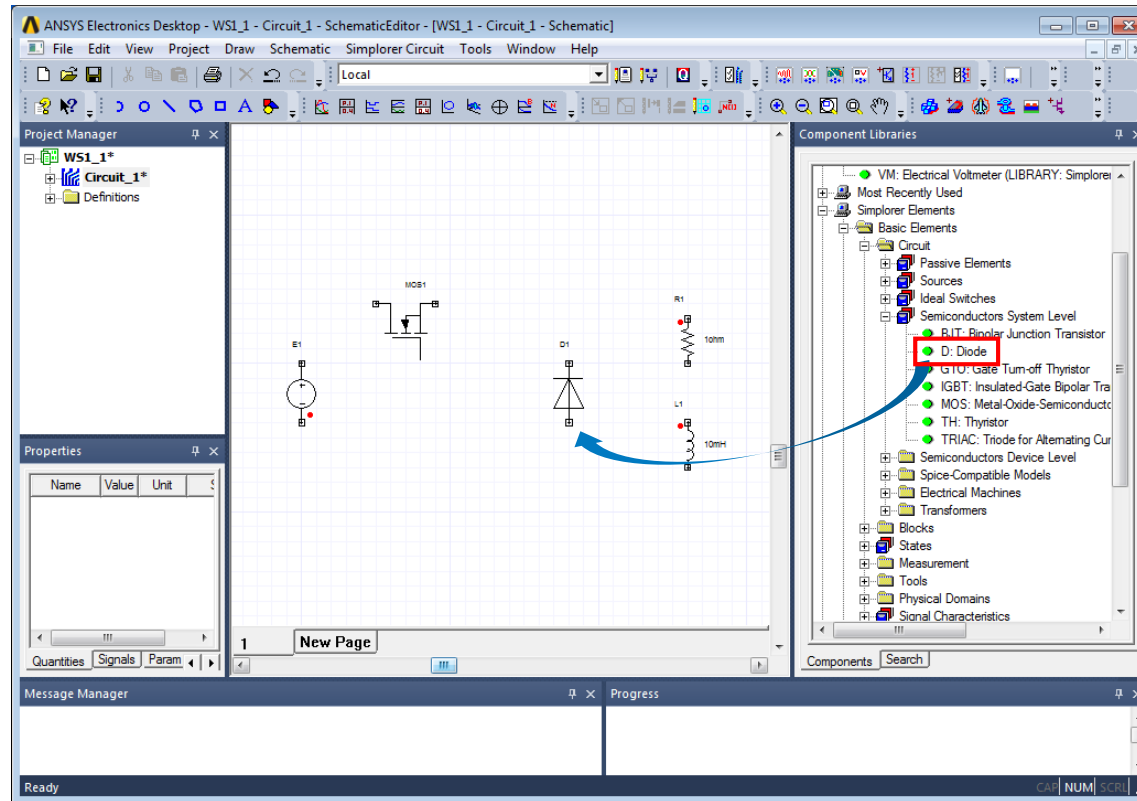
- In Comp. Libraries window *Simplorer Elements* → *Basic Elements* → *Circuit* → *Semiconductor system Level*
- Select the **MOSFET (MOS)**, drag and drop it into the Schematic
- Select the **MOS** and use the shortcut **Ctrl+R** to rotate it 90 degrees counterclockwise
- Double click on the **MOSFET**, uncheck **Use Pin** and set the **Control Signal** to **1** (the value “1” turns the switch ON, while a “0” turns the switch OFF). Note that the shape of MOSFET’s gate pin slightly changes



# Insert Components


- Diode

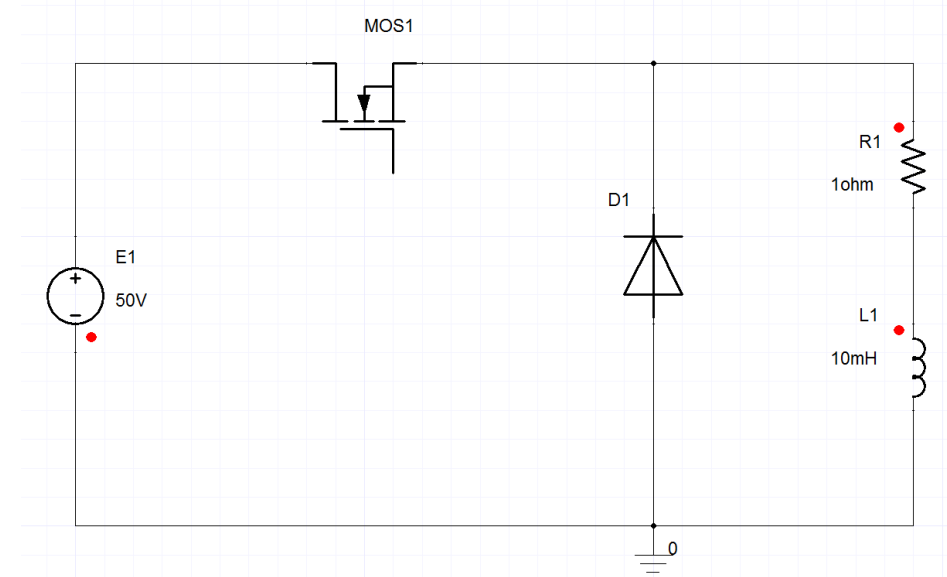
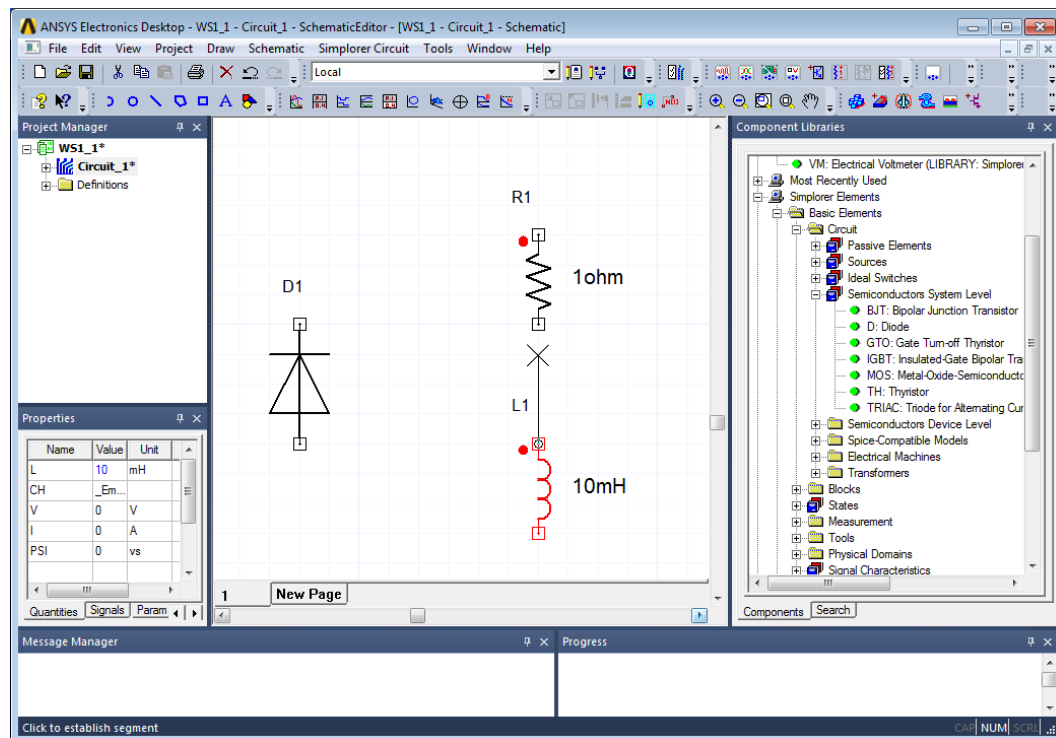
- Select the **D: Diode**, drag and drop it into the Schematic
- Press **Esc** key to exit the insert mode and use the shortcut **Ctrl+D** to fit all





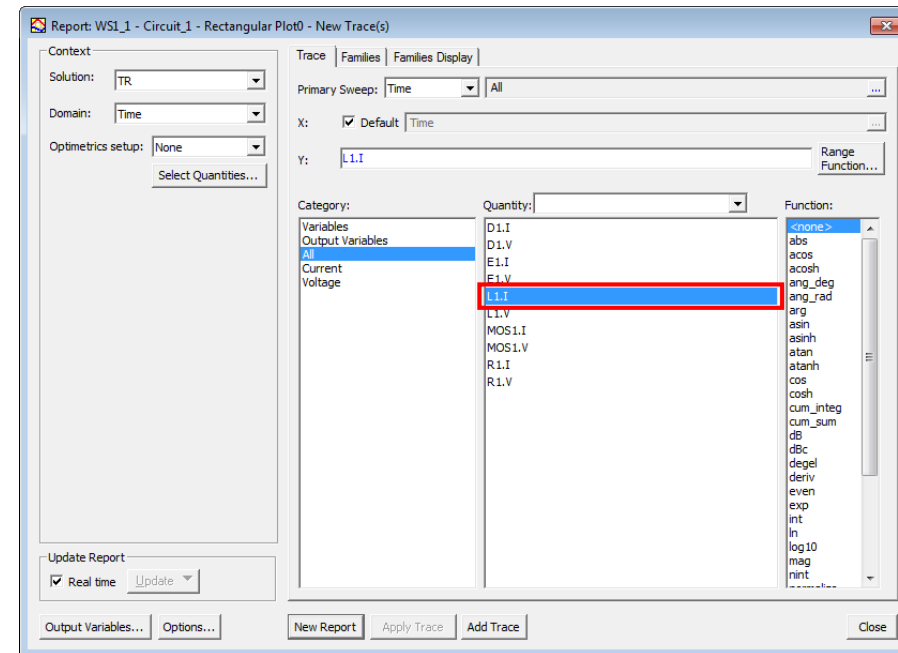
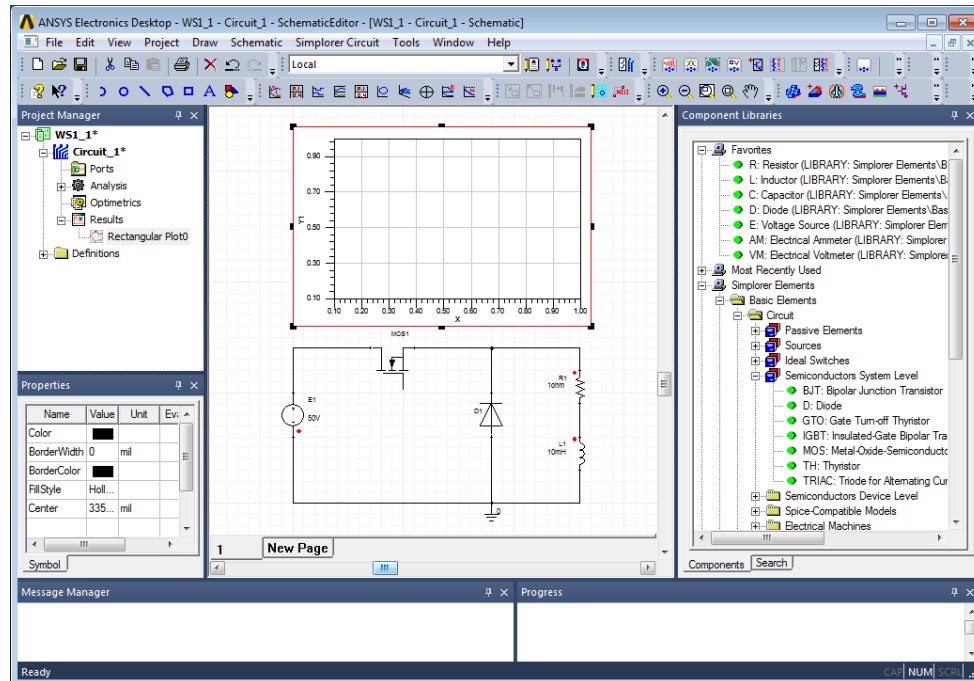
# Connect Components

- Place the mouse over one terminal of a component. The mouse pointer changes its shape becoming a cross. Press the **LMB** (Left Mouse Button) 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, for example above the Circuit
- The **New Trace** window pops-up automatically. Select the Current flowing through the Inductance L1 by selecting the quantity **L1.I**, press the **Add Trace** button and then **Close**

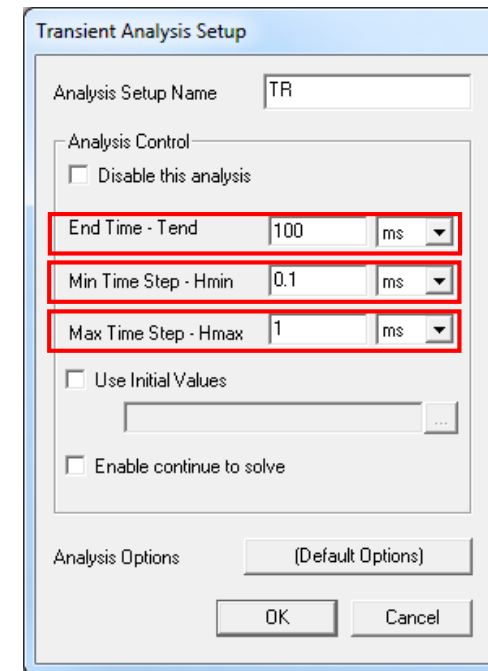


*Note: it is best to zoom out before placing the report, so you can resize it in relationship to the existing circuit*

# Setup the Simulation Analysis

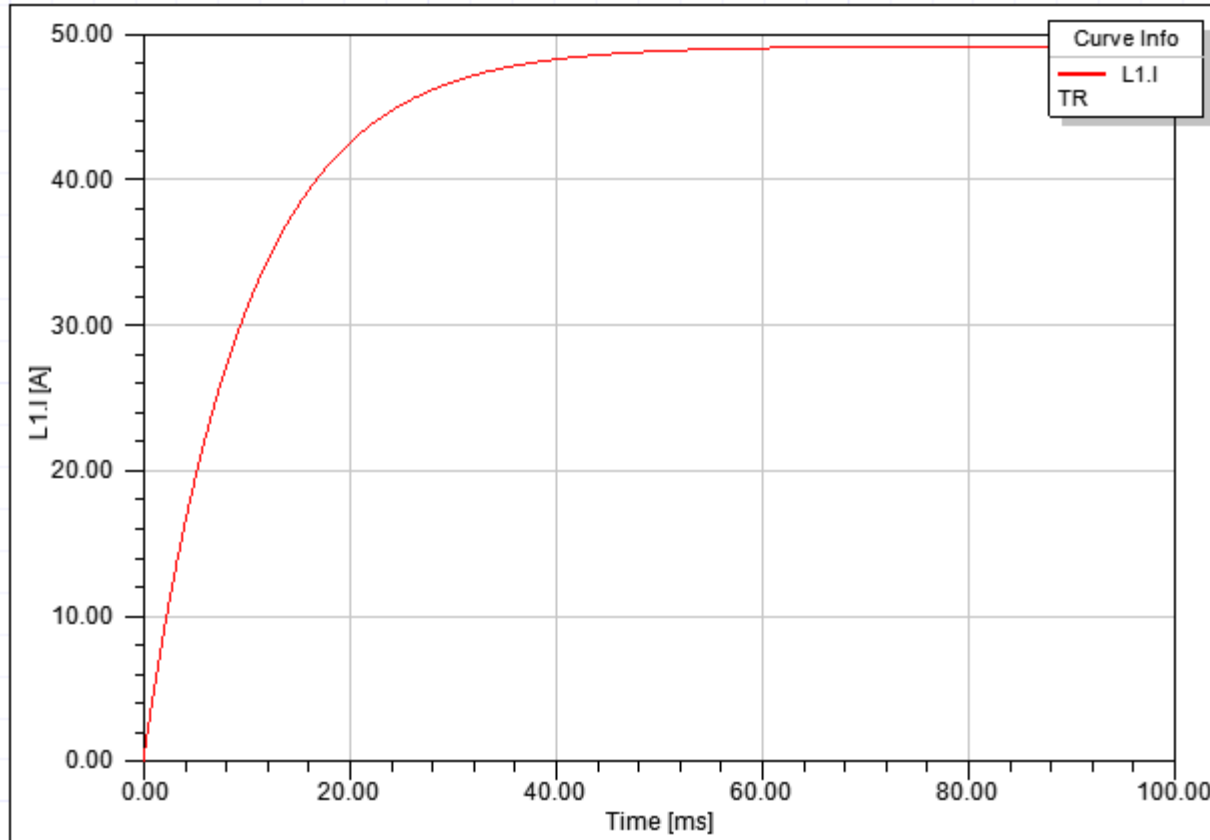
- Since the Mosfet is closed, also the diode plays no role in our first example
- We expect in this case a simple first order transient increase of the load current, reaching the final value of  $E1/R1 = 50\text{ A}$  with a time constant of  $\tau = L1/R1 = 10\text{ ms}$
- We can set a simulation time equal to **10 times  $\tau$** , with a minimum time step of  **$\tau/10$  or less**
- Expand the Analysis in the Project Manager window and double click on TR
- In the transient analysis Setup window set:
  - Tend: **100 ms**
  - Hmin: **0.1 ms**
  - Hmax: **1 ms**
  - Press **OK**

*Note: in the present case we have chosen Hmin to be  $\tau/100$  to allow the solver to catch good details during the transition from initial to final value and Hmax  $\tau/10$  to allow the solver to speed up the simulation once reached the final steady state*



# Analyze and View Results

- Select the menu item *Simplorer Circuit* → *Analyze to run the Simulation*
- The final result should look very similar to the following Figure:

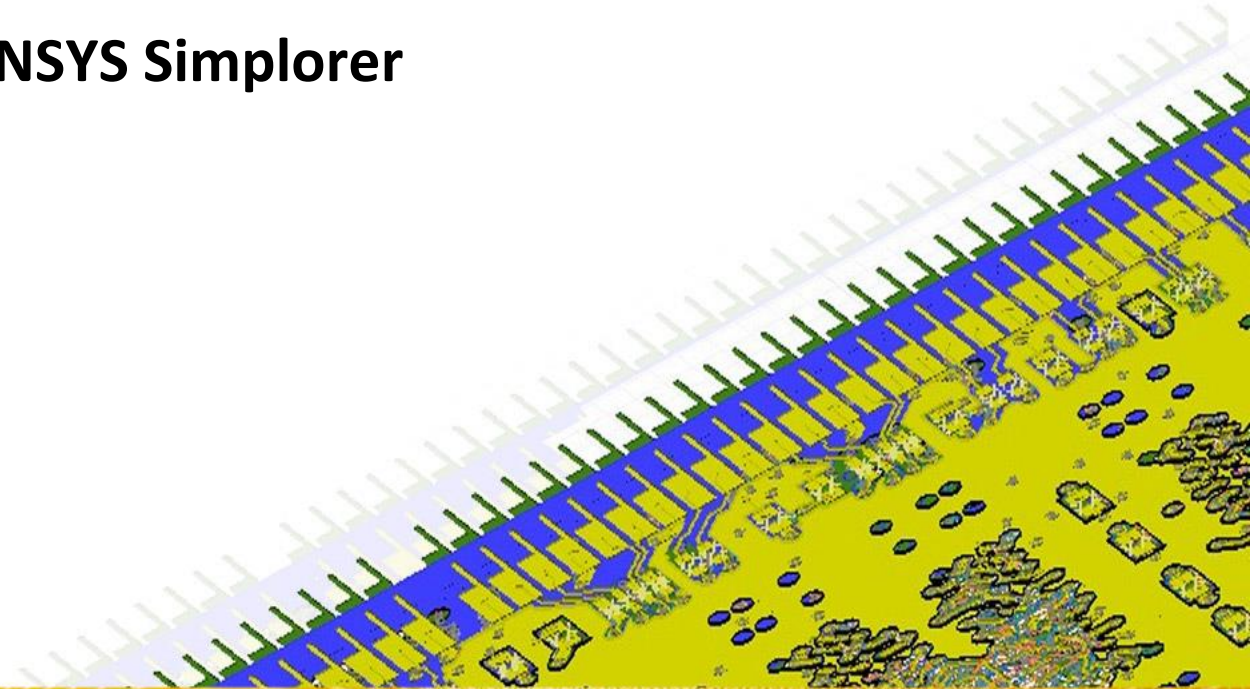


*Note that the final value is near to 50 A but not exactly 50 A. That is due to the voltage drop equal to 0.8 V present in the Mosfet model*



# Mosfet Control

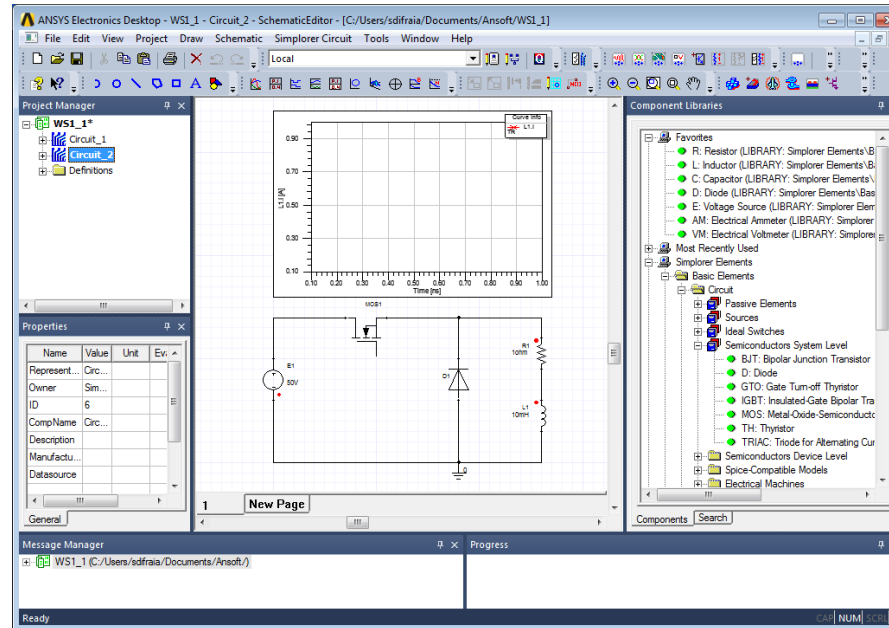
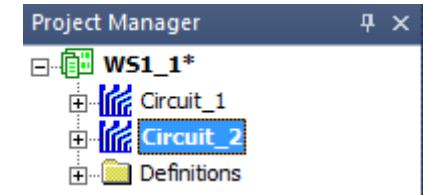
## Introduction to ANSYS Simplorer





# Copy the Simplorer Design

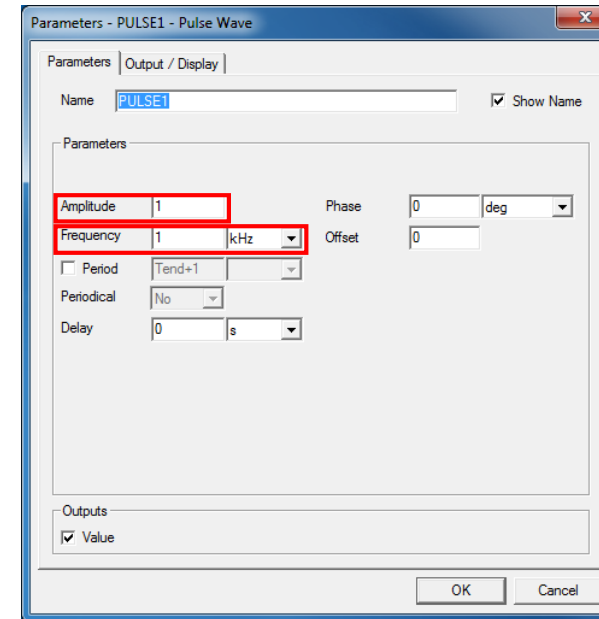
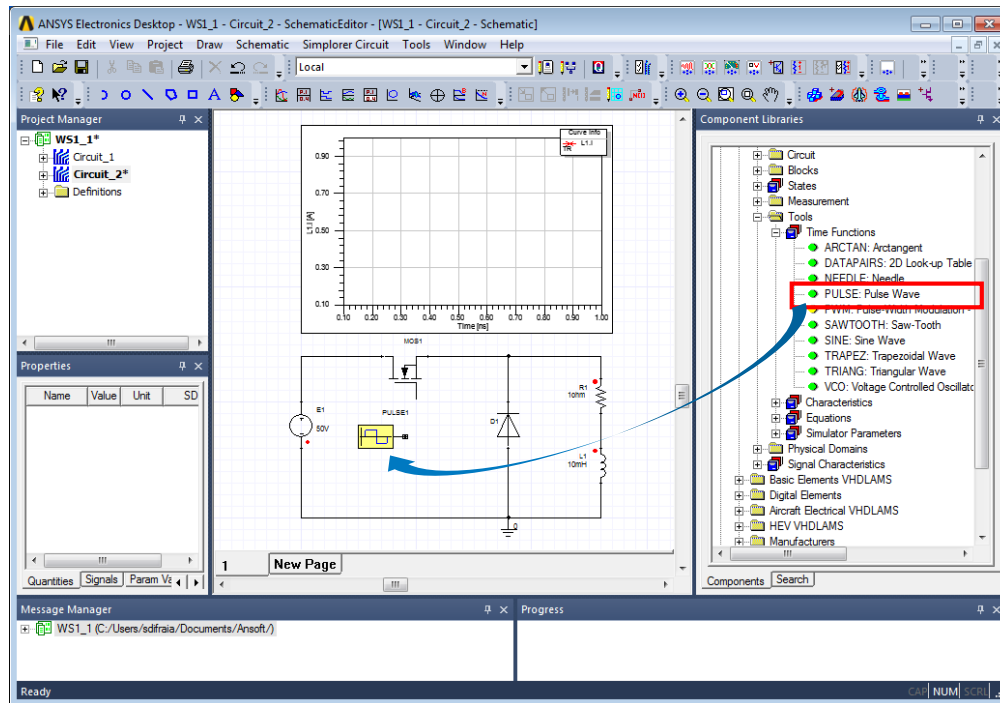
- In the Project Manager Window, select the Design **Circuit\_1**
- **Ctrl+C** to copy the Design
- Select the Project **WS1\_1**
- **Ctrl+V** to paste the Design
- A new Design named **Circuit\_2** is created
- The new Design is identical to the first one but not solved
- Click on the new Design to activate it



# Insert Components

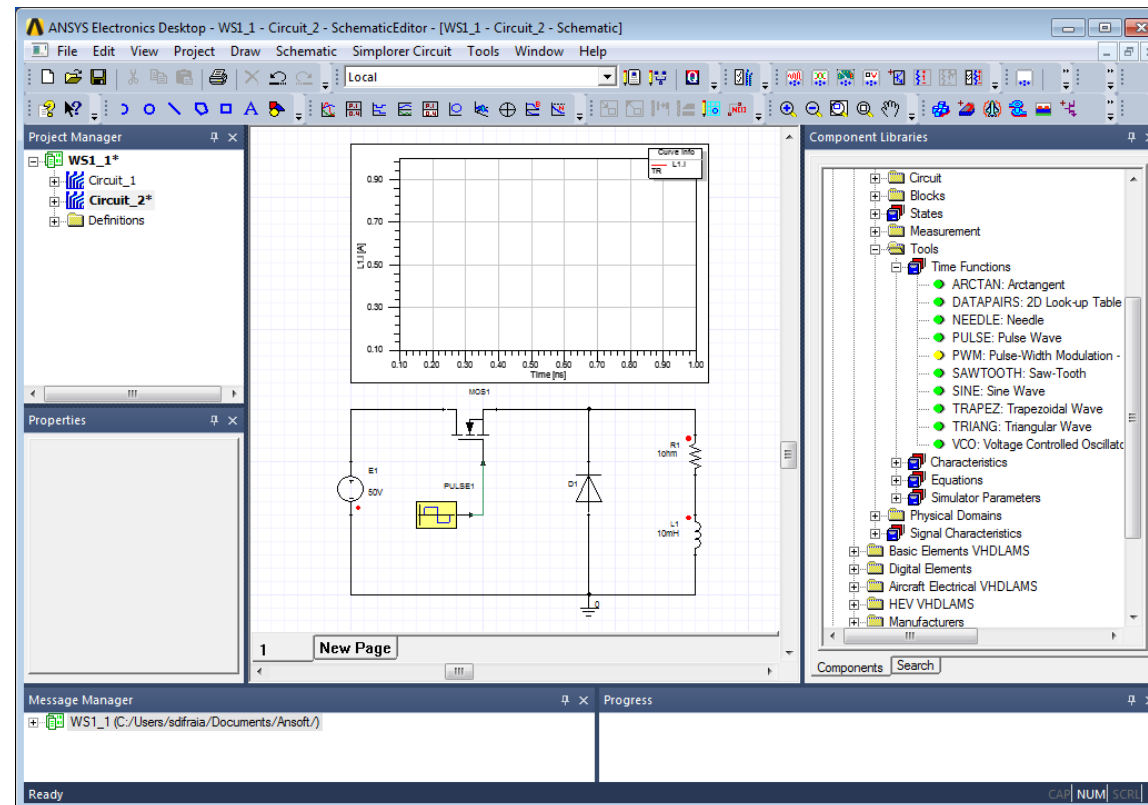
- Pulse Block

- In Comp. Libraries window *Simplorer Elements* → *Basic Elements* → *Tools* → *Time Functions*
- Select the **PULSE** component, drag and drop it into the Schematic
- Double click on the **PULSE** block and set the Amplitude to **1** and the Frequency to **1 kHz**. The default **duty cycle** is **50%**



# Modify Mosfet Properties

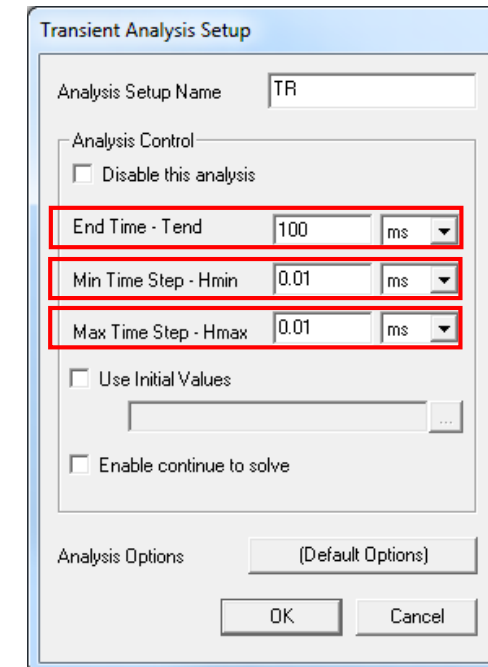
- Double click on the **Mosfet** and check again **Use Pin**
- Connect the **PULSE block** to the **Gate Pin** of the **Mosfet**
- Note the connection color: the two component exchange each other pure mathematical signals and this fact is pointed out with a **green** color connection



# Setup the Simulation Analysis

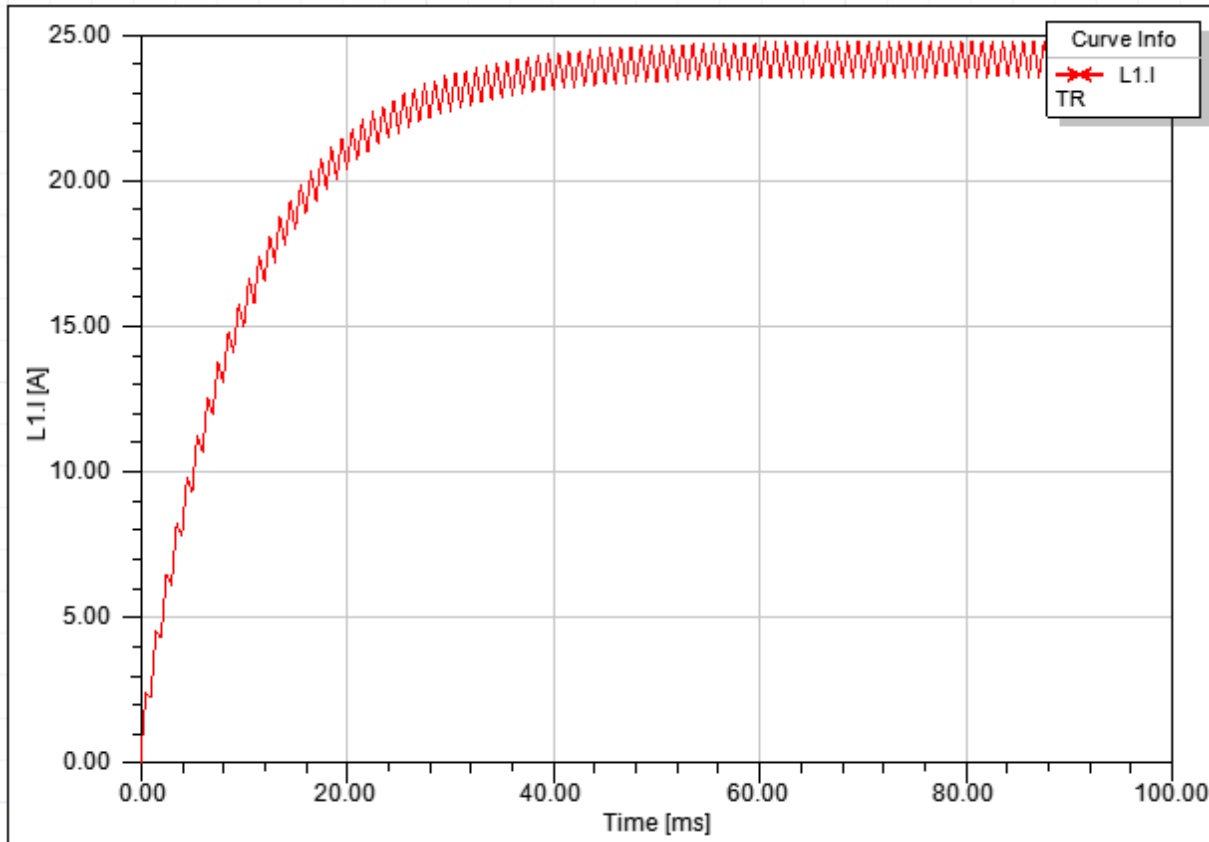
- Since the Mosfet is now driven through a control signal at 1 kHz (meaning a period  $T = 1\text{ ms}$ ) with a duty cycle of 50%, we should set Hmin and Hmax accordingly
- We expect in this case a first order transient increase of the load current, reaching the final value of  $E1/R1 * \text{duty cycle} = 25\text{ A}$  with a time constant of  $\tau = L1/R1 = 10\text{ ms}$
- We can set a simulation time equal to 10 times  $\tau$ , with a minimum time step of  $T/10$  or less
- Expand the Analysis in the Project Manager window and double click on TR
- In the transient analysis Setup window set:
  - Tend: 100 ms
  - Hmin: 0.01 ms
  - Hmax: 0.01 ms
  - Press OK

*Note: in the present case we have chosen  $Hmin = Hmax = T/100$  to allow the solver to catch good details during all the simulation time*



# Analyze and View Results

- Select the menu item *Simplorer Circuit* → *Analyze to run the Simulation*
- The final result should look very similar to the following Figure:

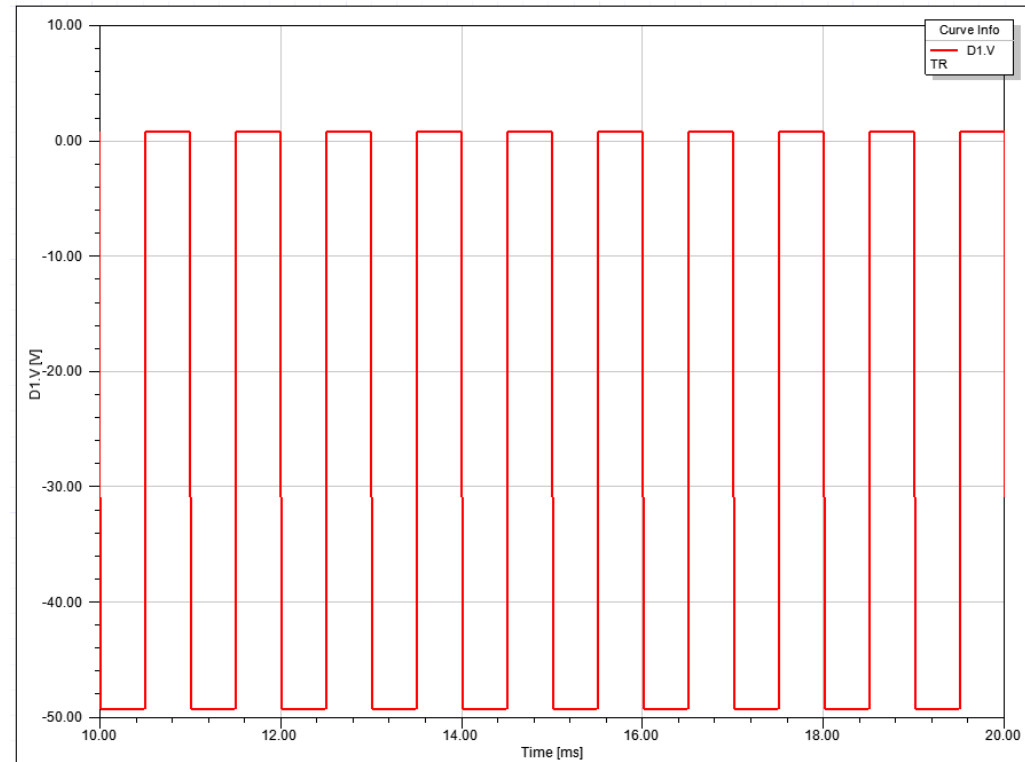
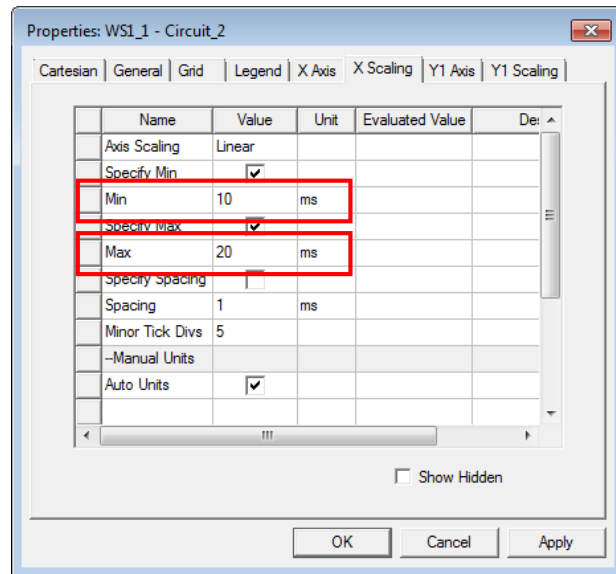


*Note that the final value is near to 25 A but not exactly 25 A.  
That is due to the voltage drop equal to 0.8 V present in the Mosfet model*



# Further Results

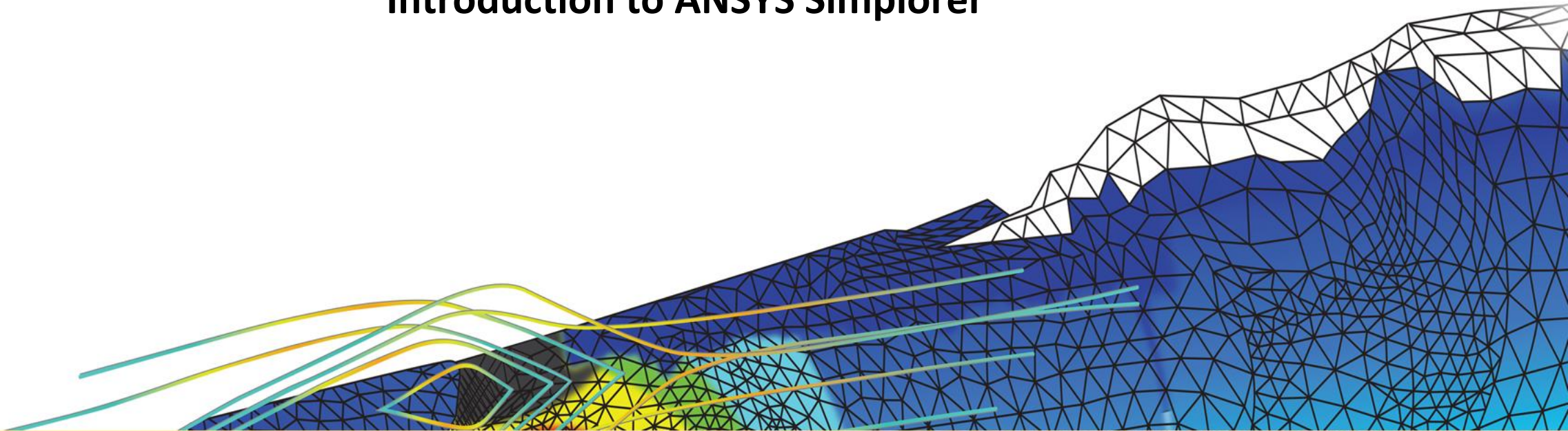
- Insert a new Report using the menu item **Draw** → **Report** → **Rectangular Plot**
- In the **New Trace** window choose **D1.V** (Voltage drop on the Diode) , press **Add Trace** and **Close**
- **RMB** on the Plot Area and select **Edit in Place**. Double click on the **x-axis**, select the **X Scaling Tab** and specify the **Min** and **Max** values as in figure. Press **Apply** and then OK





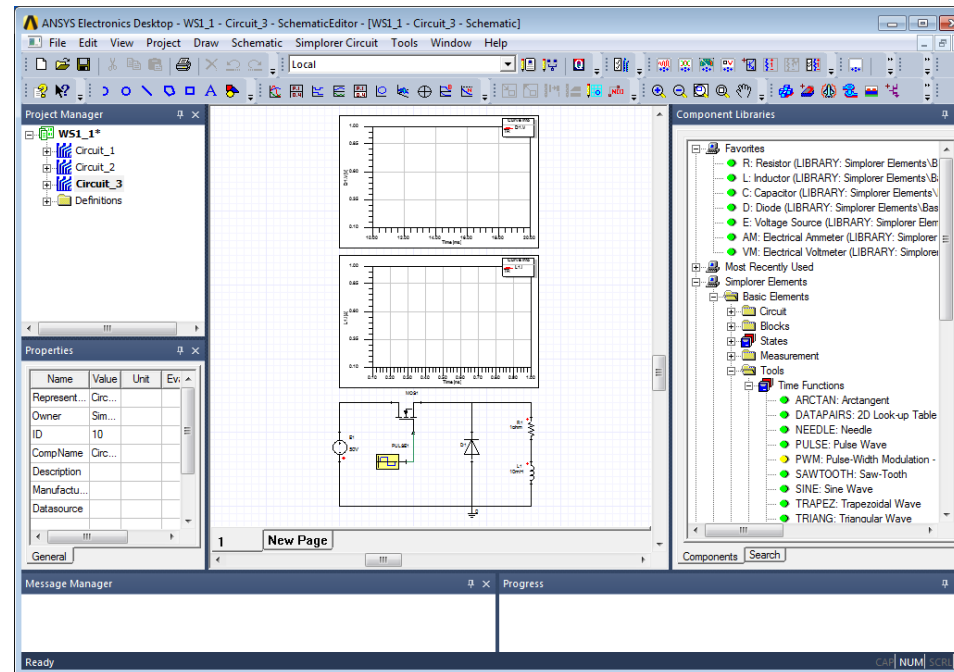
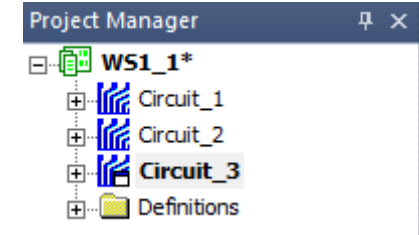
# Parametric Analysis

## Introduction to ANSYS Simplorer



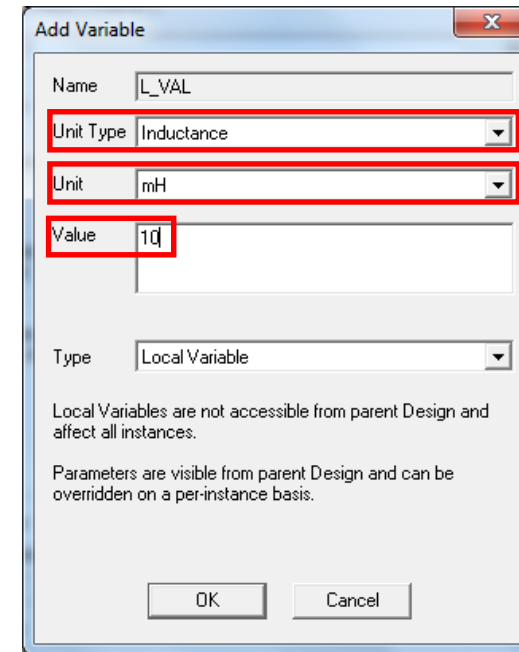
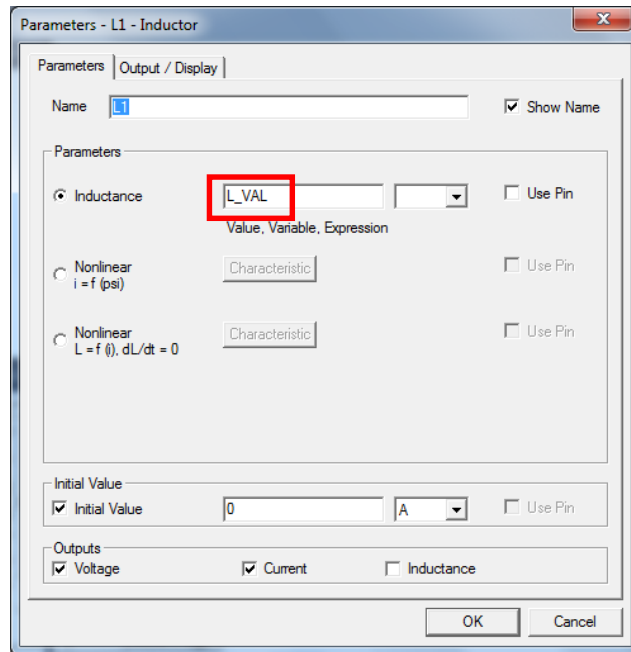
# Copy the Simplorer Design

- In the Project Manager Window, select the Design **Circuit\_2**
- Ctrl+C to copy the Design
- Select the Project **WS1\_1**
- Ctrl+V to paste the Design
- A new Design named **Circuit\_3** is created
- The new Design is identical to the second one but not solved



# Add a Variable

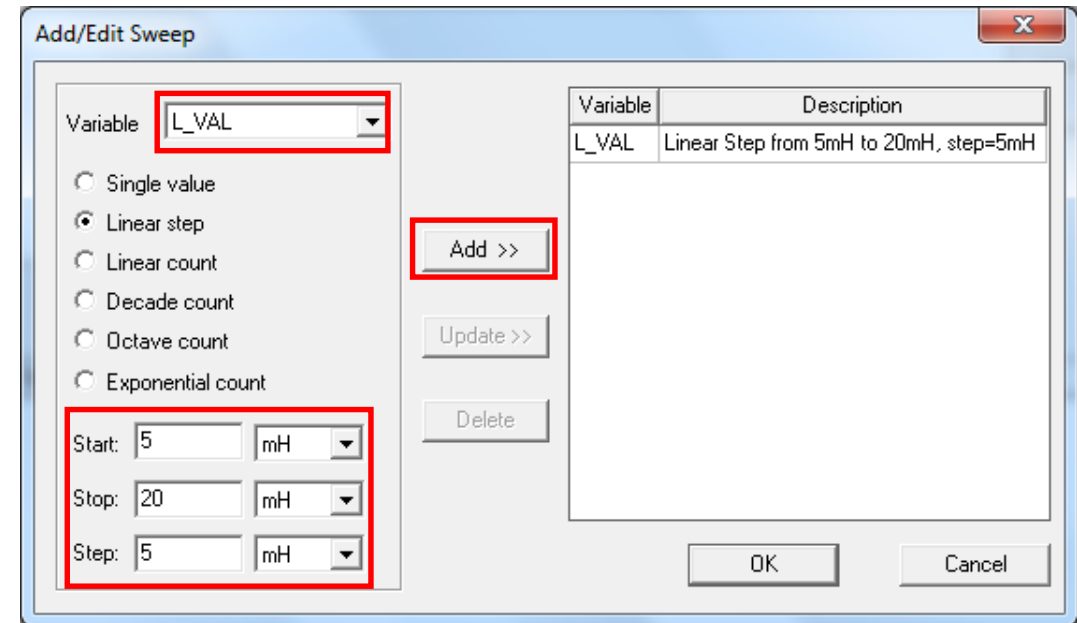
- Parameterize Inductance value
  - Double click on the Inductance L1
  - On the **Parameter** window insert **L\_VAL** as inductance value. Press **OK**
  - On the pop-up **Add Variable** window, set **Unit Type**, **Unit** and **Value** as in figure



**Note:** By defining a variable name (**L\_VAL**) it becomes a design variable. All design variables are accessible by selecting menu item **Simplorer Circuit** → **Design Properties**. If variable name is appended by symbol \$, it is defined as a project variable and can be accessed across all the designs

# Parametric Sweep Setup

- Launch Setup Sweep Analysis window,
  - Select the menu item *Simplorer Circuit* → *Optimetrics Analysis* → *Add Parametric*
- Add Parameter Sweep for **L\_VAL**
  - In Setup Sweep Analysis window, select Add
  - In Add/Edit Sweep window,
    - Variable: **L\_VAL**
    - Linear Step: ☒ **Checked**
    - Start: **5 mH**
    - Stop: **20 mH**
    - Step: **5 mH**
    - Select **Add>>**
    - Press **OK**

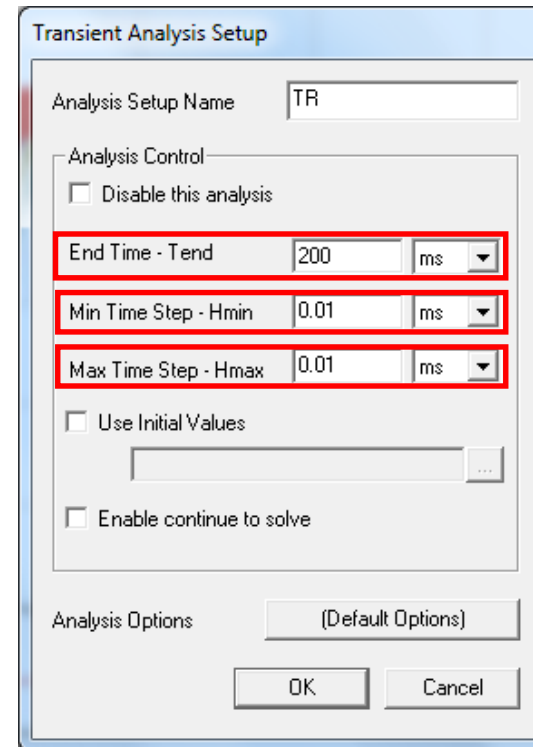


*Note: this will add 4 sweeps for L\_VAL: 5 mH, 10 mH, 15 mH and 20 mH*




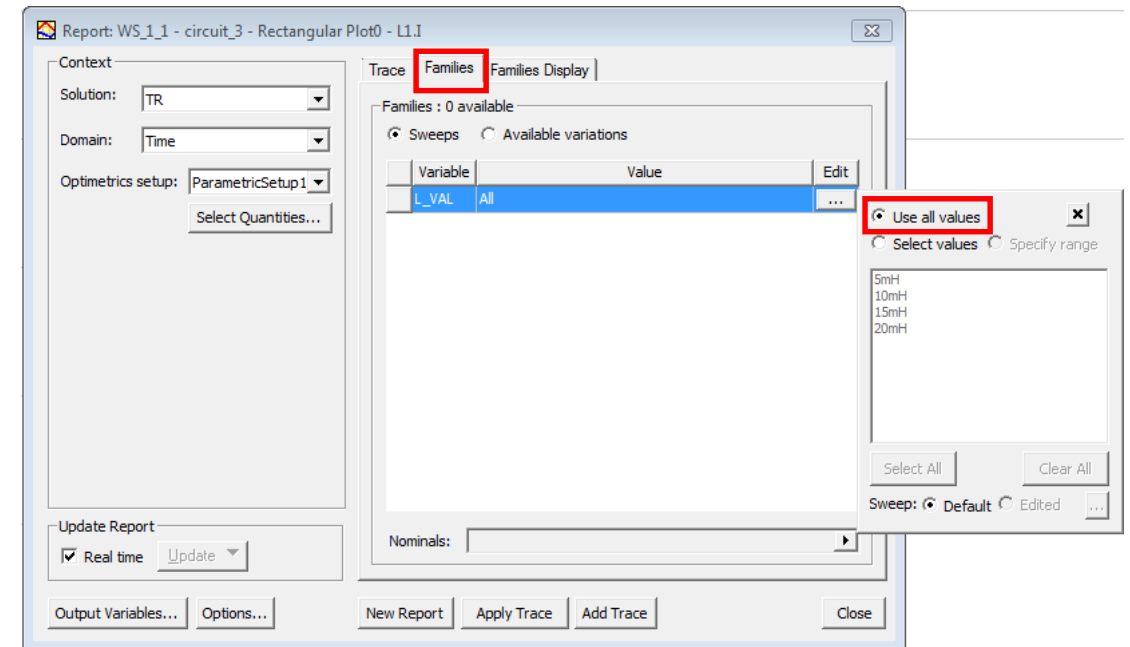
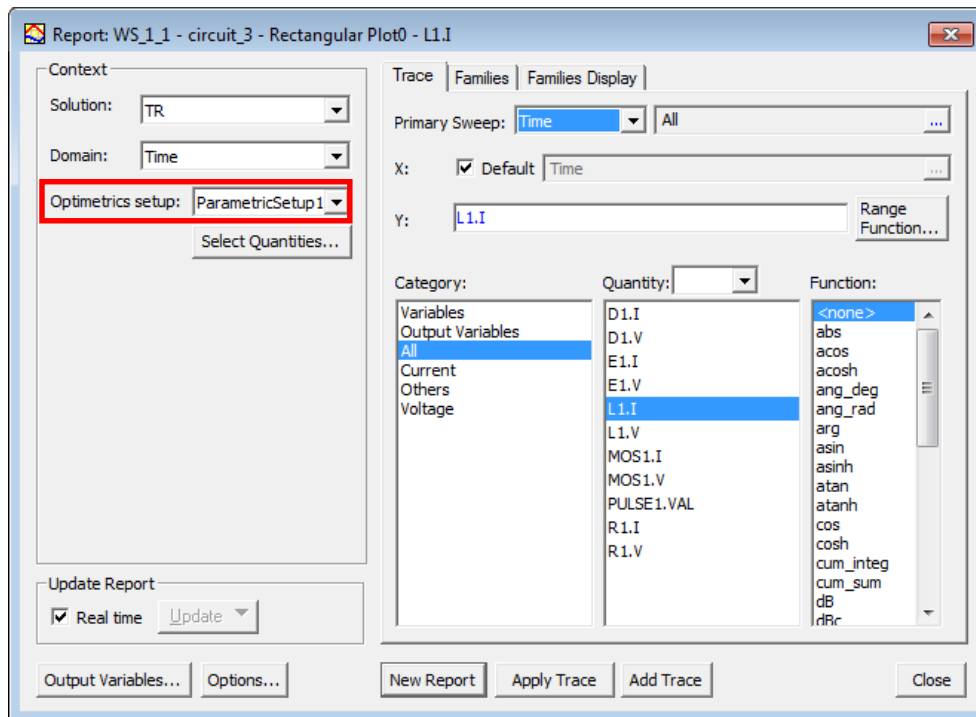
# Setup the Simulation Analysis

- Since the Mosfet control signal frequency did not change, we can leave Hmin and Hmax as they are
- The maximum value for L\_VAL is 20 mH, thus the maximum value for  $\tau$  is 20 ms
- We can set as before a simulation time equal to 10 times  $\tau$
- Expand the Analysis in the Project Manager window and double click on TR
- In the transient analysis Setup window set:
  - Tend: 200 ms
  - Hmin: 0.01 ms
  - Hmax: 0.01 ms
  - Press OK



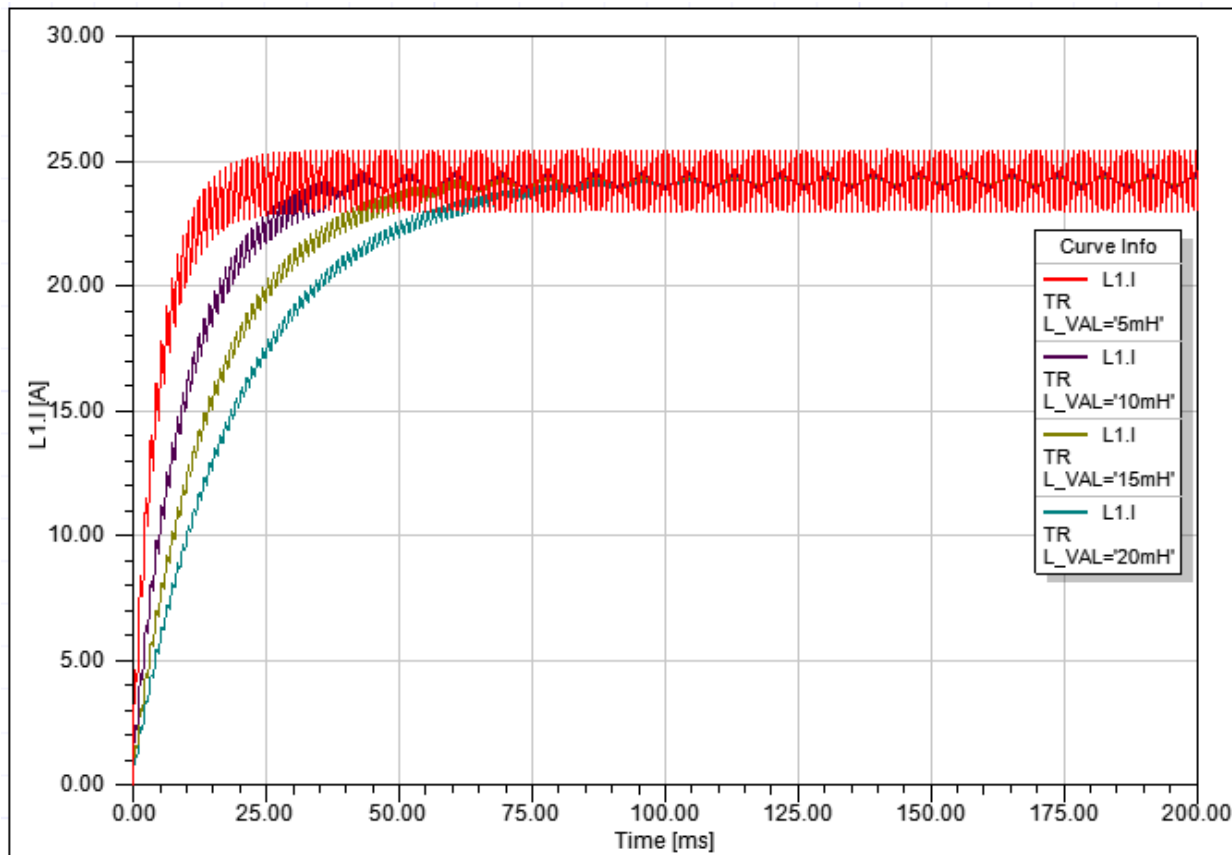
# Prepare the Postprocessing

- Select the plot displaying the load current L1.I
- *RMB on plot area* → *Modify Report*
- Optimetrics Setup: **ParametricSetup1**
- **Families Tab**
- Click the icon  and check **Use all values**. Press **Apply Trace** and **Close**



# Analyze and View Results

- From the Project Manager window, select the *ParametricSetup1* → *RMB* → *Analyze* to run the sweep Simulation
- The final result should look very similar to the following Figure:



*Note that the final average value is the same for all the 4 sweeps, since the values of  $E1$  and  $R1$  do not change. The current rising time differs due to the different values of time constant  $\tau$ , as well as the ripple amplitude*

# Saving the Project

- This completes the workshop
- Save the file with the name **WS\_1\_1** in the working folder