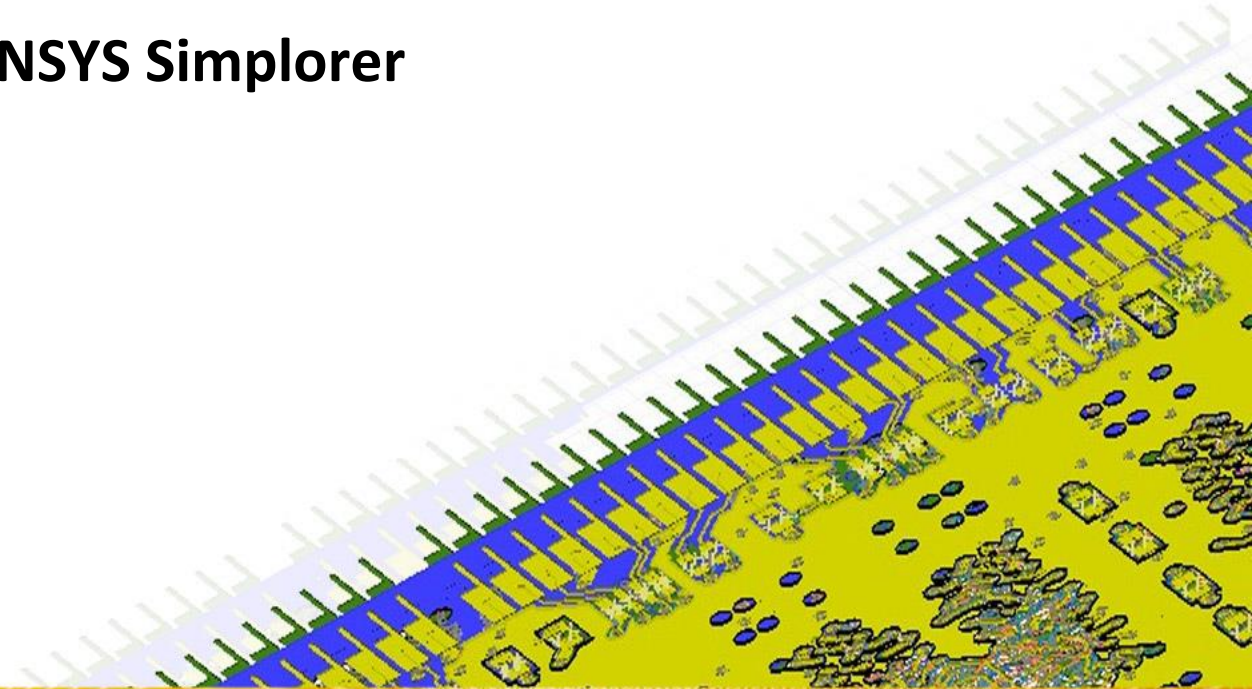




# Workshop 2.2: Single Phase Inverter



Introduction to ANSYS Simplorer

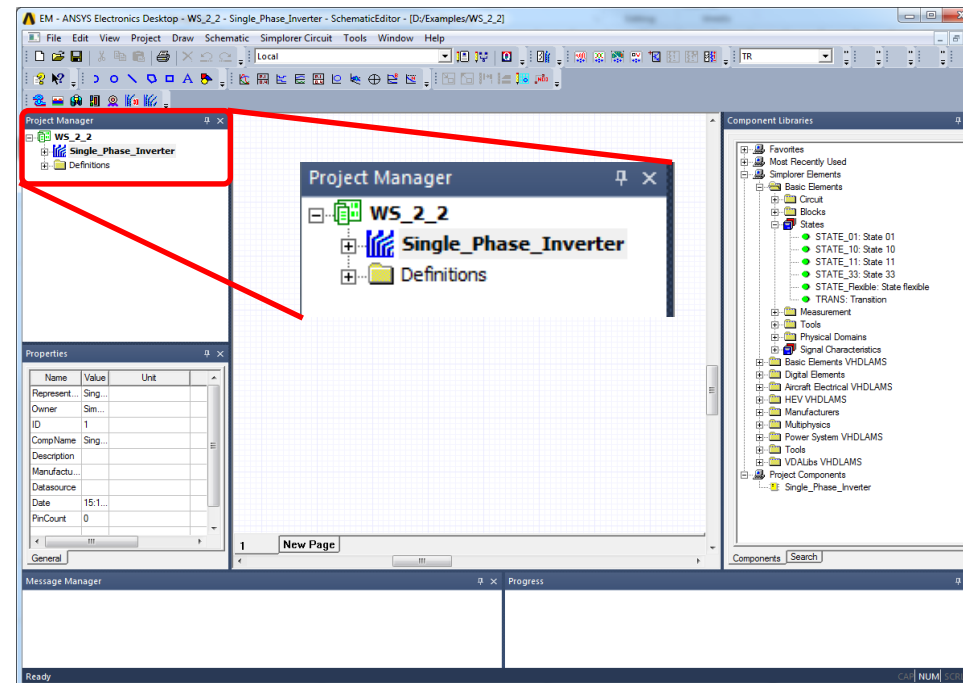
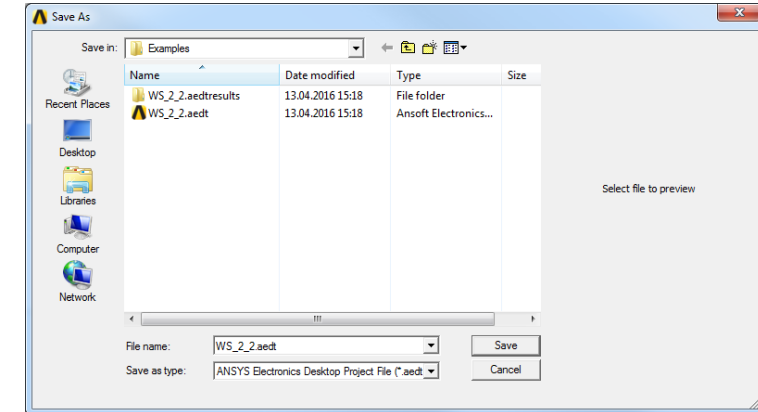


# Overview

- **Single Phase Inverter**
  - In this example we will build a simple DC/AC IGBT based single phase inverter feeding a passive R-L load. The inverter functioning is controlled by a Bang-Bang (hysteresis) logic.
  - In particular we will learn
    - How to create State Machines
    - How to use State Machines to control Electronic Switches
    - How to use Initial Values (ICA) blocks
    - How to use Functions (EQU) blocks

# Insert a Simplorer Design

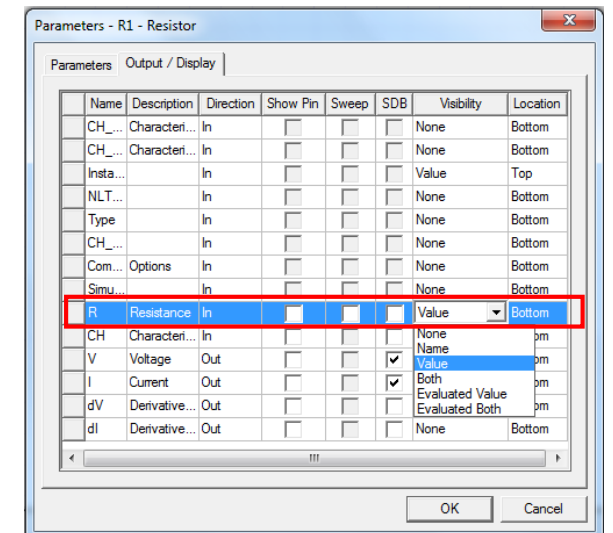
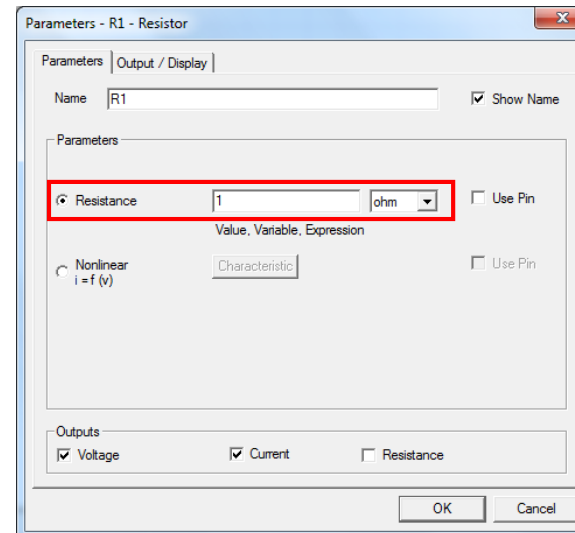
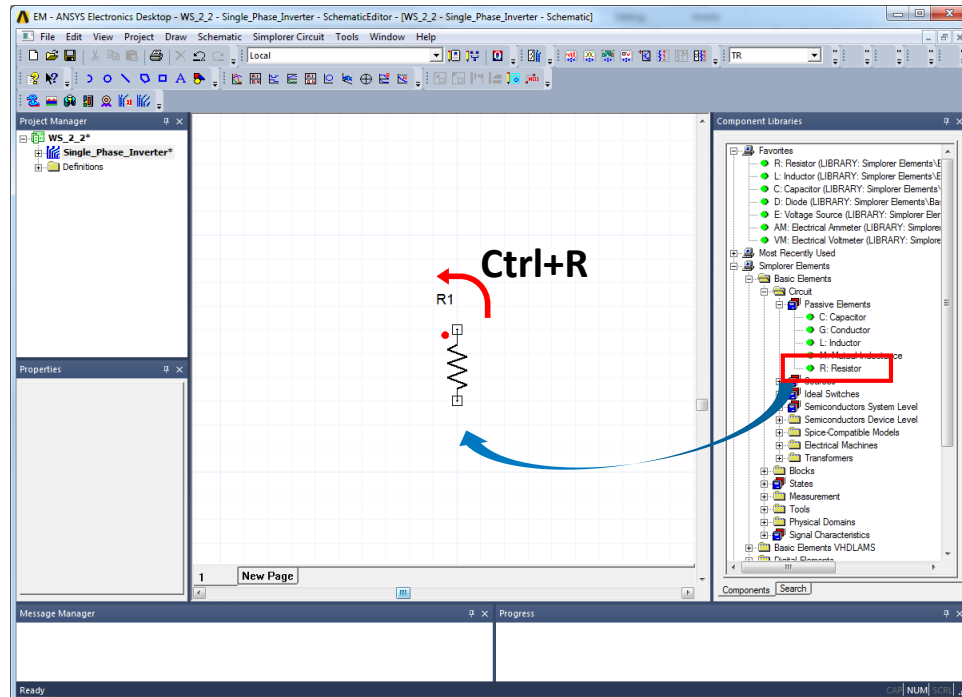
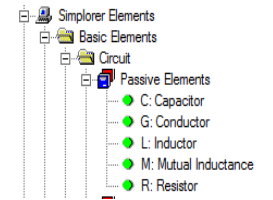
- Launch the Electronics Desktop 2016
  - Save the Project as **WS\_2\_2.aedt**
  - Insert a Simplorer Design using the icon 
  - Rename the Design as **Single\_Phase\_Inverter**
  - Save again the project using the icon 



# Insert Components

- Resistance

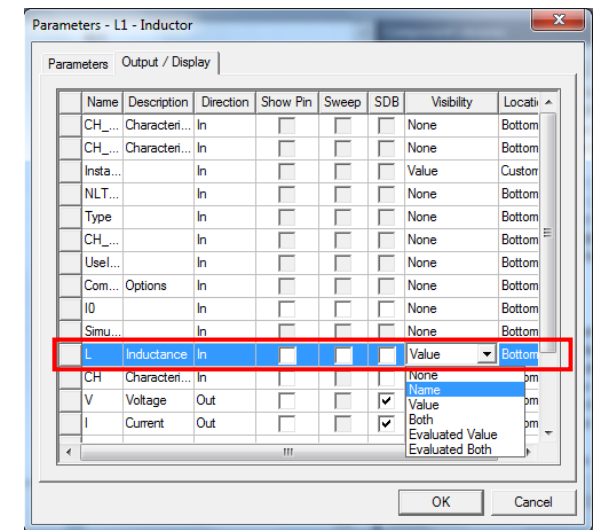
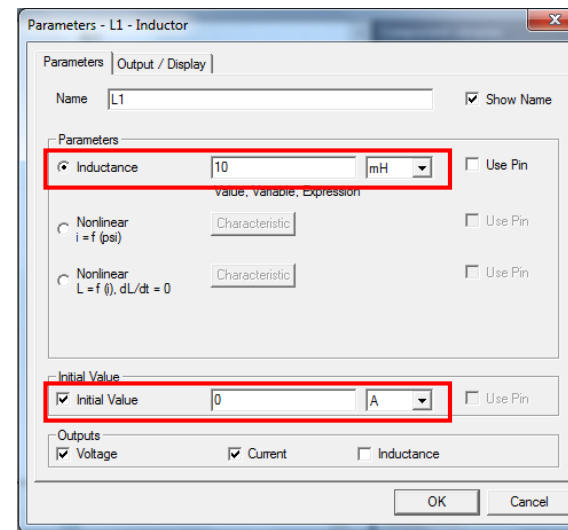
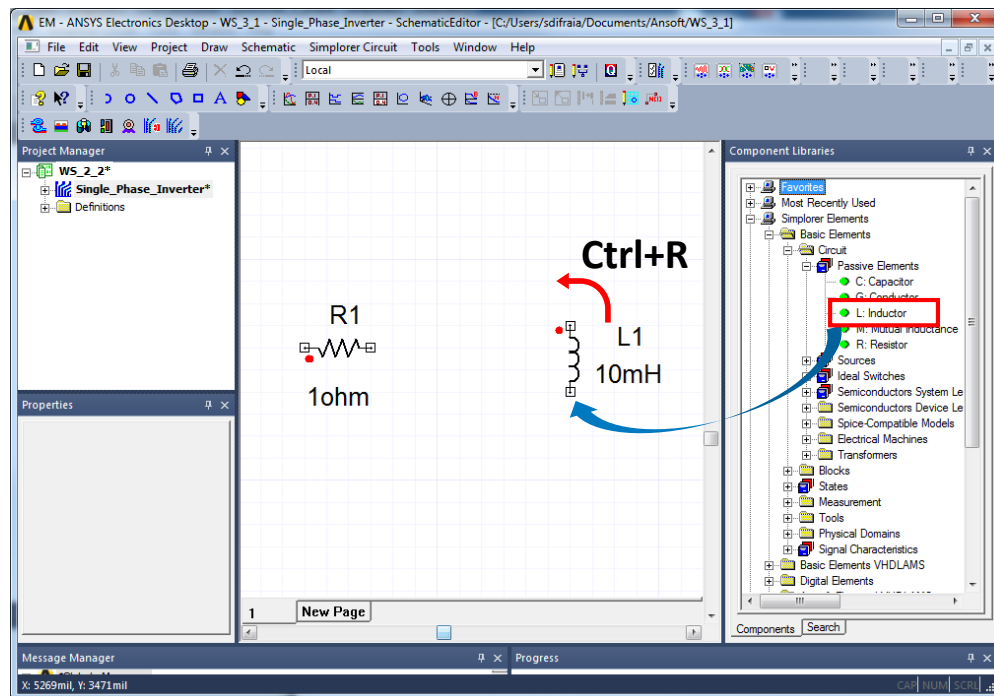
- In Component Libraries window *Simplorer Elements* → *Basic Elements* → *Circuit* → *Passive Elements*
- Select the **R: Resistor**, drag and drop it into the Schematic. Press **Esc** key to exit the insert mode
- Select the Resistor and use the shortcut **Ctrl+R** to rotate it 90 degrees counterclockwise
- Double click on the Resistor, change the value to **1  $\Omega$**
- In the **Output/Display Tab** under Visibility, select Value for Resistance



# Insert Components

- Inductance

- Select the **L: Inductor**, drag and drop it into the Schematic
- Press **Esc** key to exit the insert mode and use the shortcut **Ctrl+D** to fit all
- Select the Inductor and use the shortcut **Ctrl+R** to rotate it 90 degrees counterclockwise
- 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

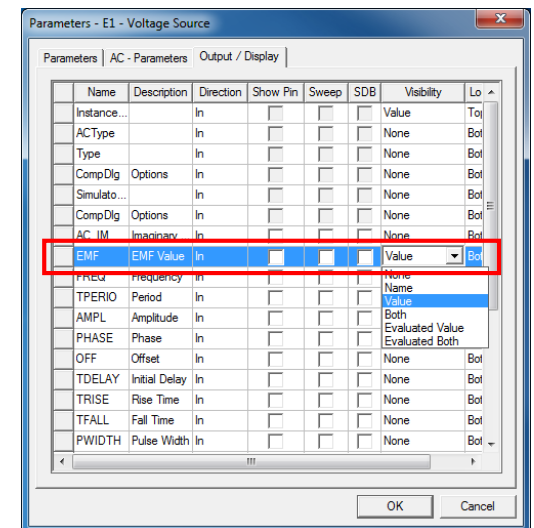
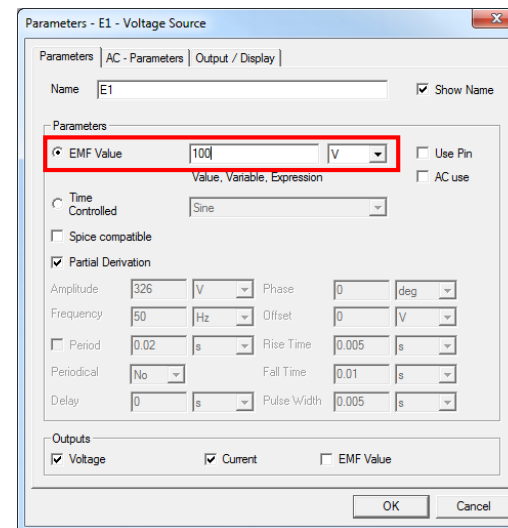
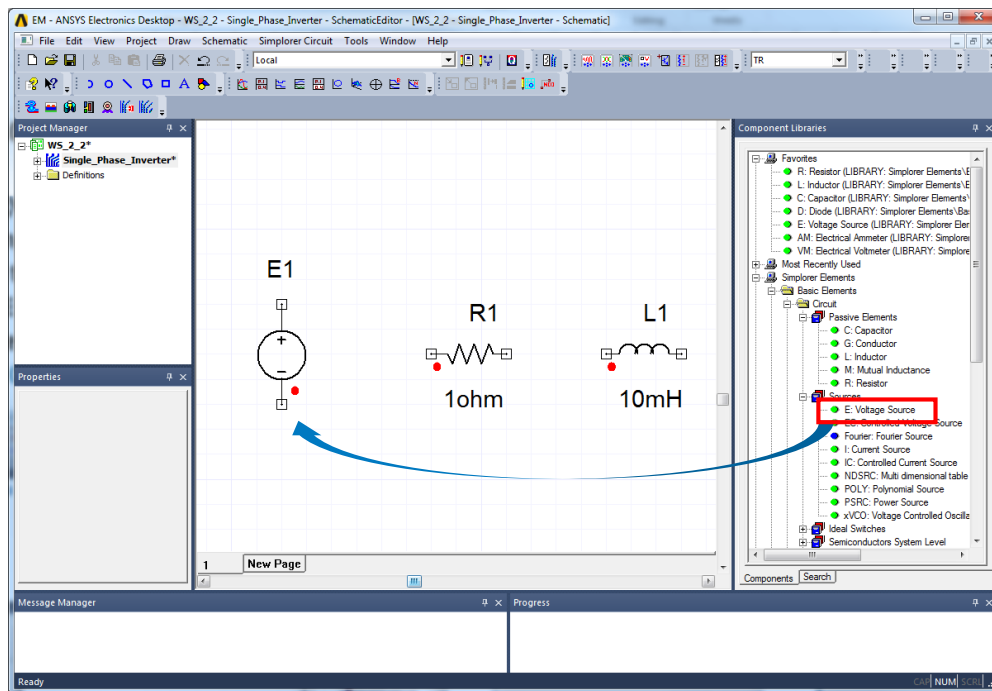




# Insert Components

- Voltage Source

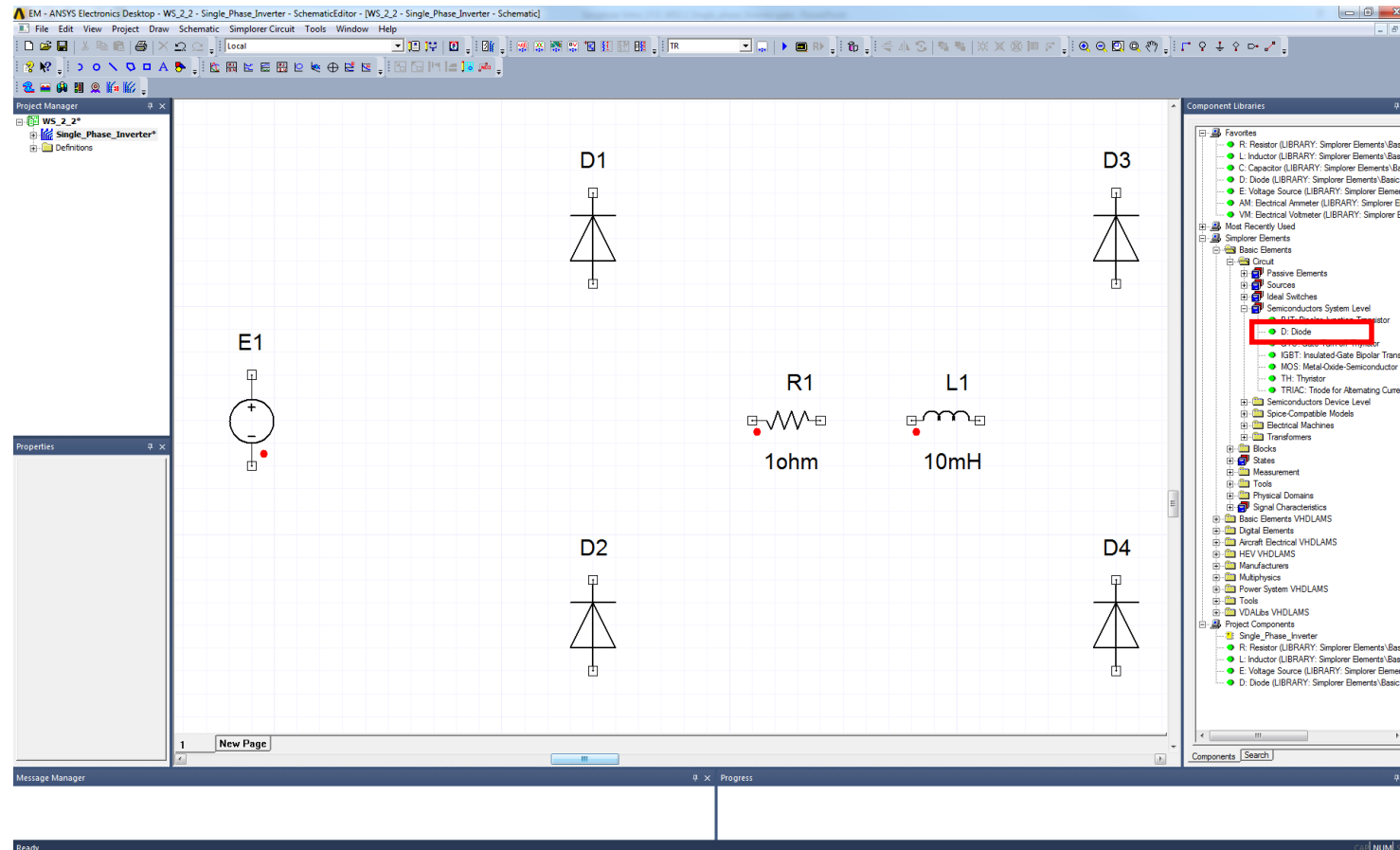
- In Component Libraries window *Simplorer Elements* → *Basic Elements* → *Circuit* → *Sources*
- Select the **E: Voltage Source**, drag and drop it into the Schematic
- Press **Esc** key to exit the insert mode and use the shortcut **Ctrl+D** to fit all
- Double click on the **Voltage Source** and change the value to **100 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 – Diodes

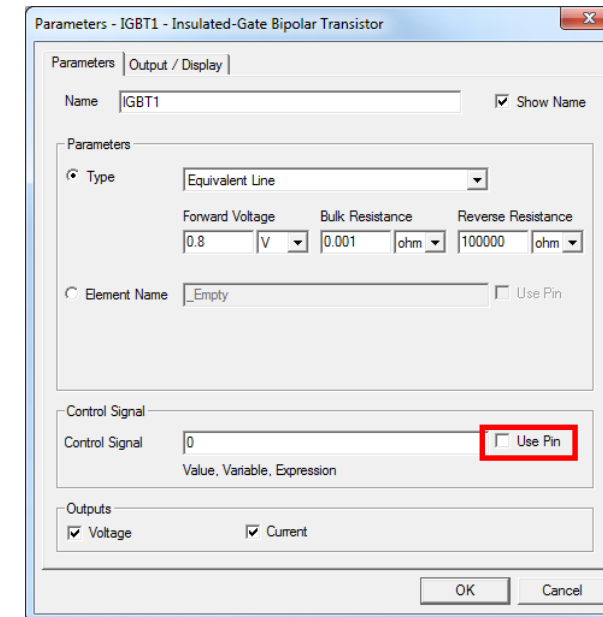
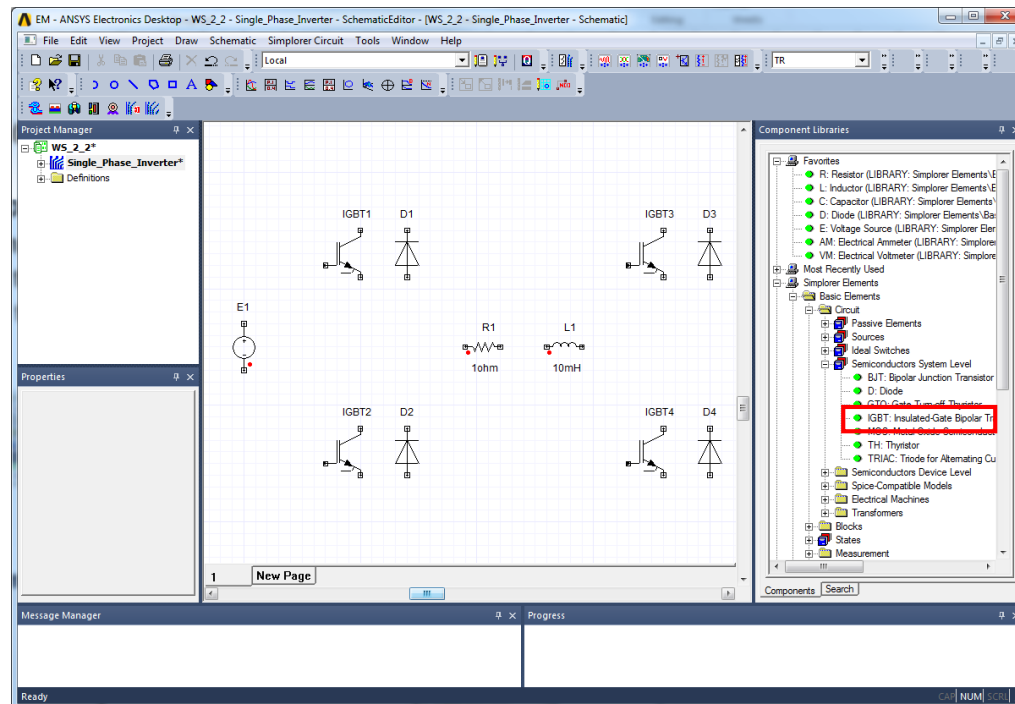
- In Comp. Libraries window *Simplorer Elements* → *Basic Elements* → *Circuit* → *Semiconductor System Level*
- Select the Diode, drag and drop it four times into the Schematic
- Press **Esc** key to exit the insert mode and use the shortcut **Ctrl+D** to fit all



# Insert Components

- Electronics Switches (IGBTs)

- In Comp. Libraries window *Simplorer Elements* → *Basic Elements* → *Circuit* → *Semiconductor system Level*
- Select the **IGBT**, drag and drop it four times into the Schematic
- For all the four IGBTs please uncheck **Use Pin**




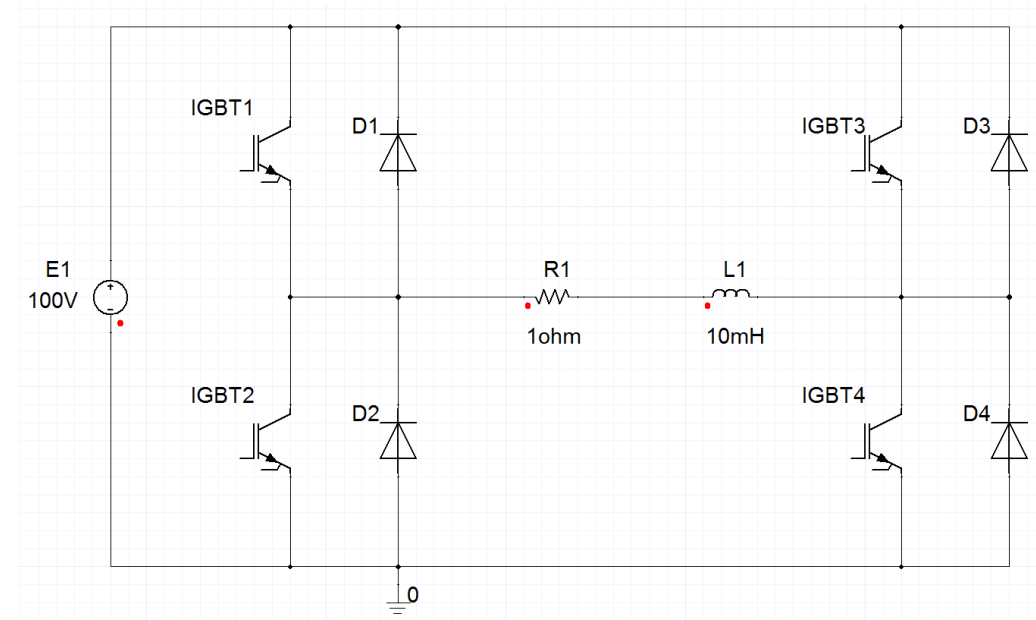
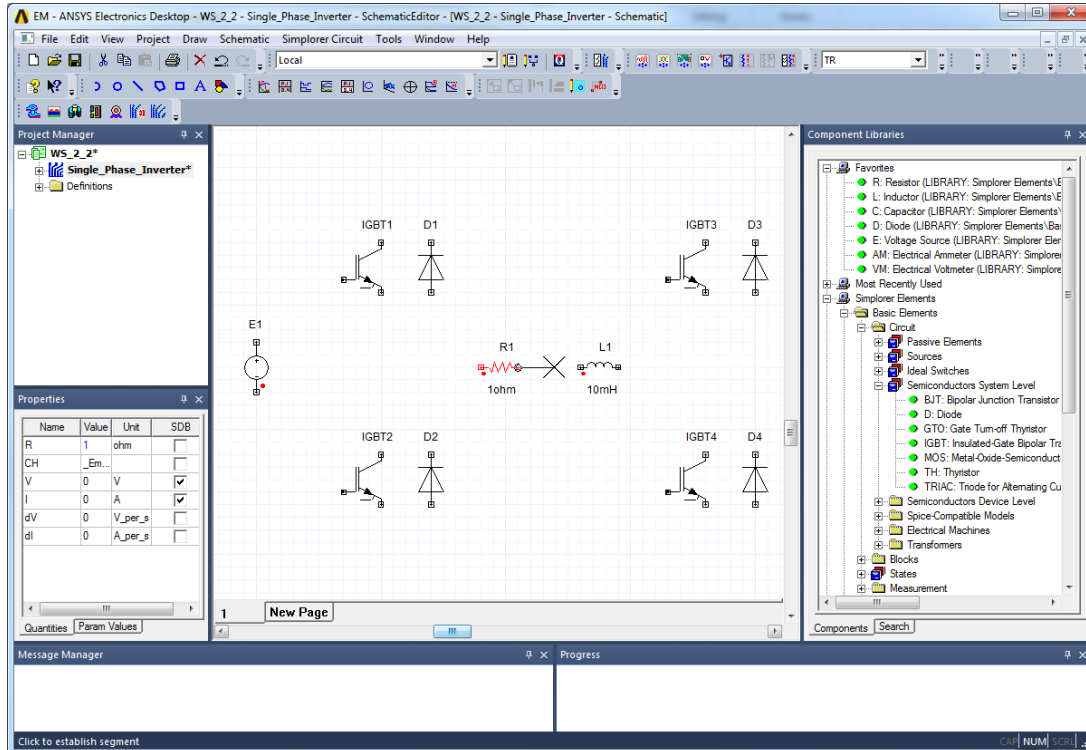


# Insert Components – Alternative method

- In order to create the circuit faster, it is also possible to copy and paste already inserted components
- In case of the 4 Diodes and IGBTs previously inserted, one could proceed the following way:
  - Insert the first Diode and the first IGBT and place them beside each other
  - Select the inserted Diode and IGBT holding the **Ctrl** key pressed
  - Use the **Ctrl+C** and **Ctrl+V** keys to copy and paste the selection 3 further times, obtaining a total of four identical Diode-IGBT couples
- The above described procedure is always applicable and very useful, especially when one has to deal with numerous identical components or components set

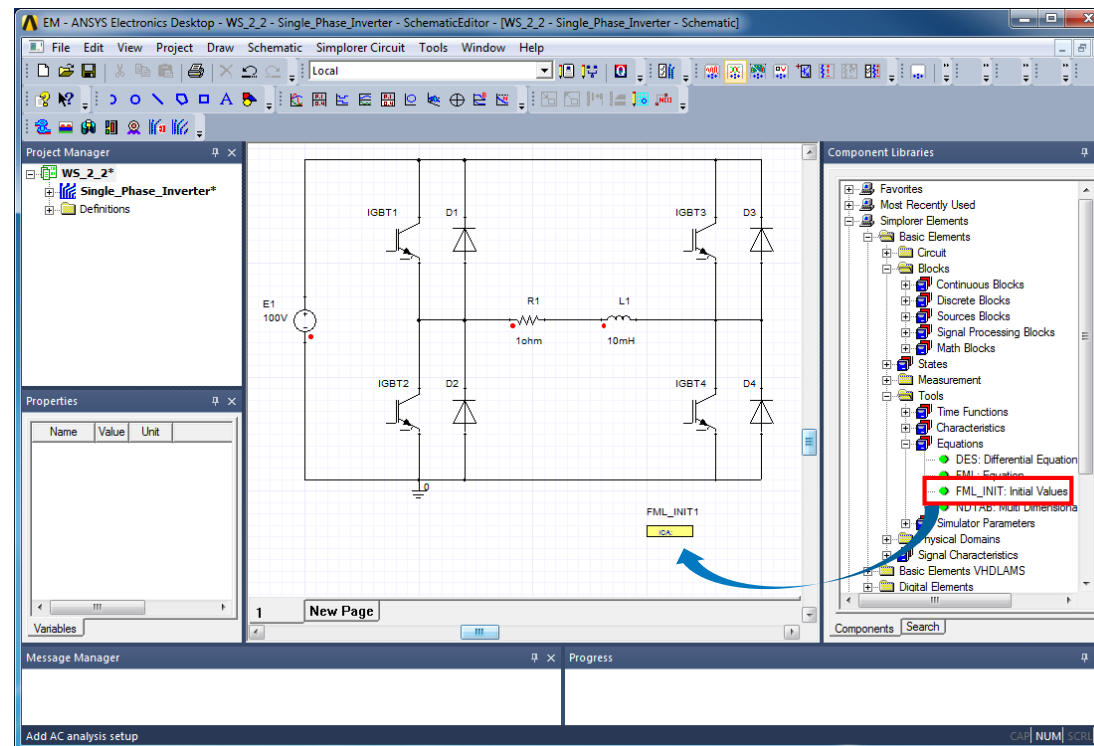
# Connect Components

- 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



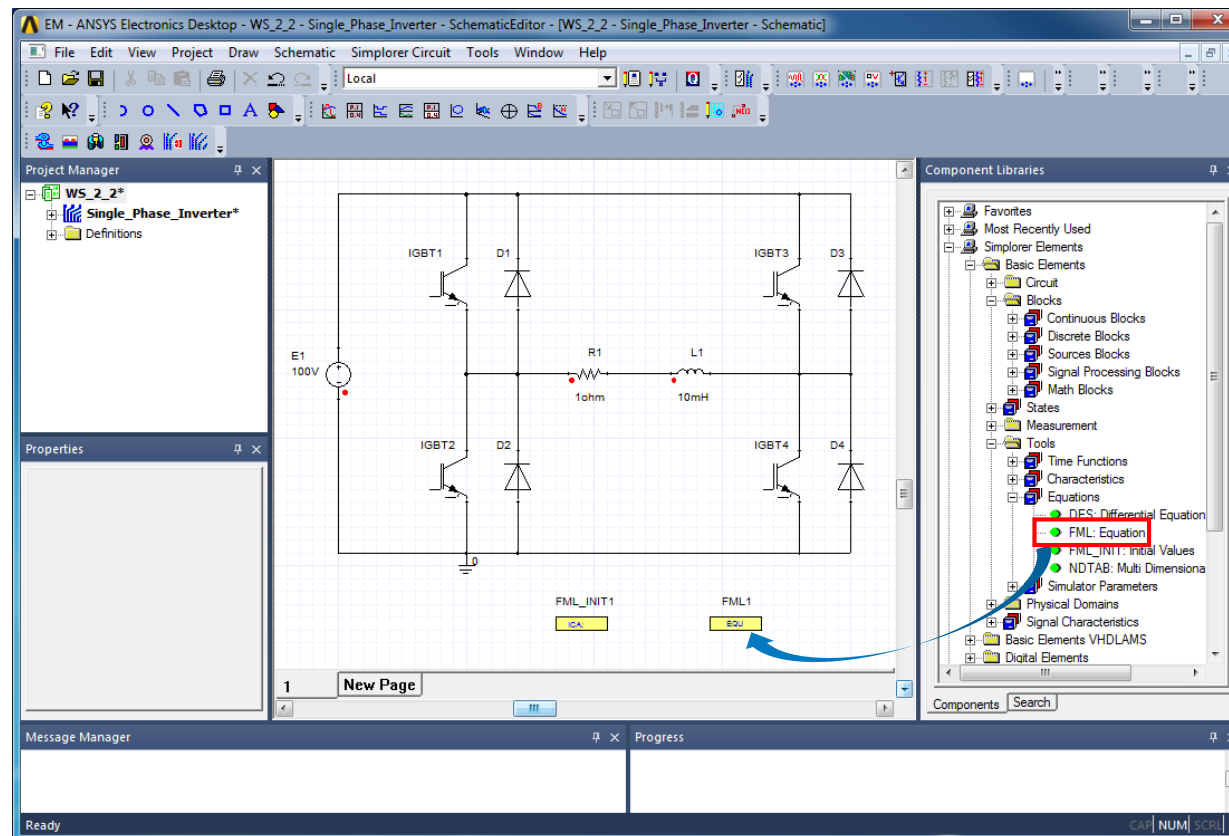
# Insert Blocks

- We need to define some initial or constant values to be used during the simulation
- In particular we need to define the output frequency of the DC/AC inverter, the amplitude of the desired (almost sinusoidal) load current and the width of the control band
- To that purpose we need to insert in the Schematic a particular block called **FML\_INIT (ICA)**
- In Component Libraries window *Simplorer Elements* → *Basic Elements* → *Tools* → *Equations*
- Select the **FML\_INIT** component, drag and drop it into the Schematic




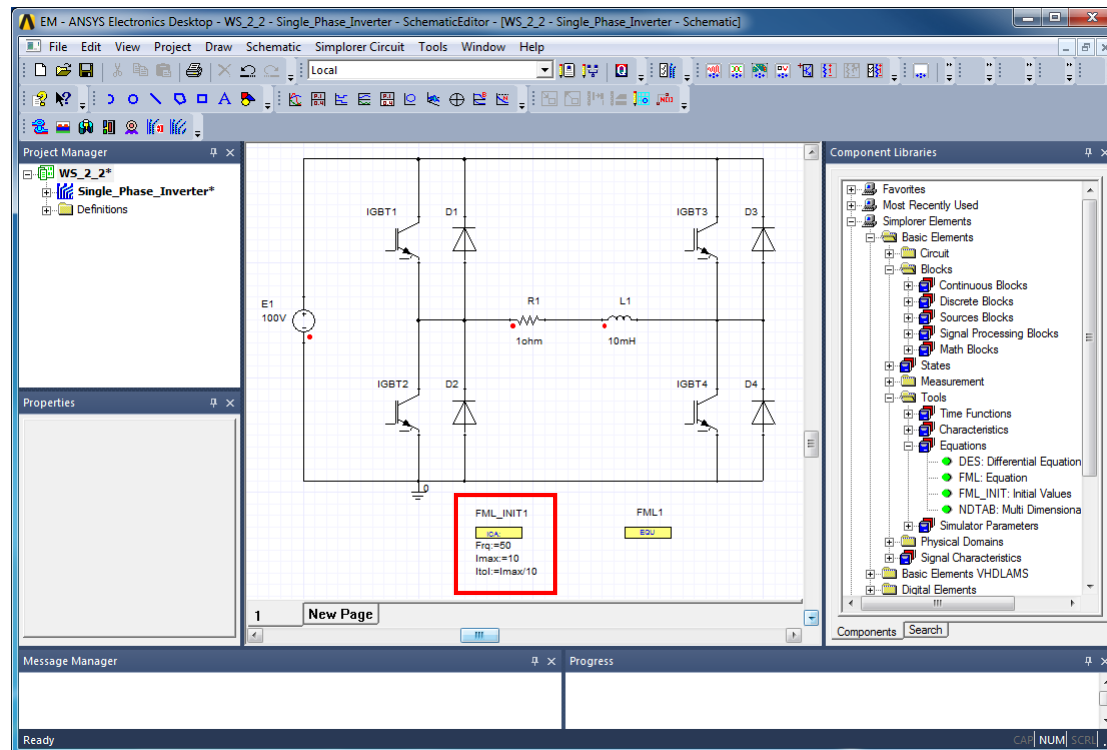
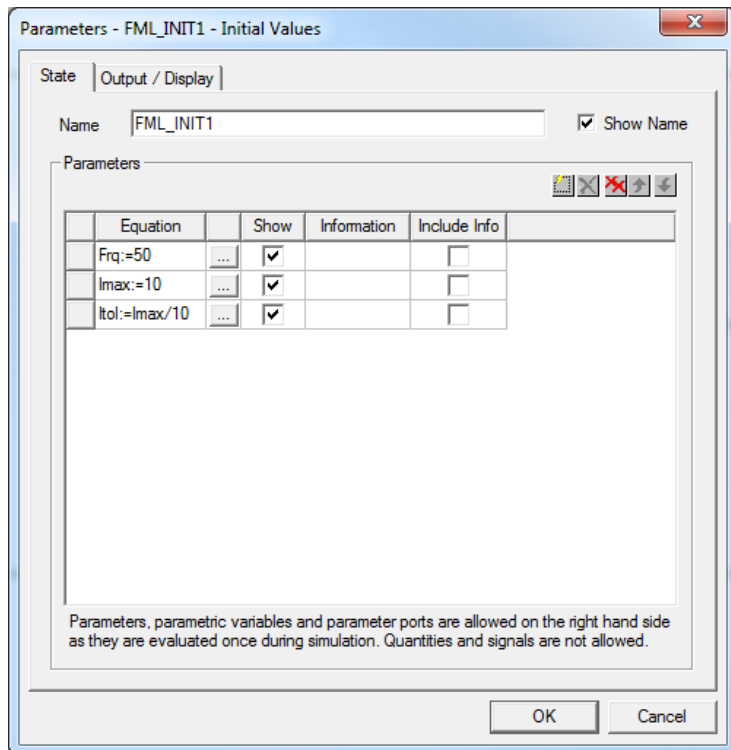
# Insert Blocks

- We need as well to define the sinusoidal behavior of the desired load current
- To that purpose we need to insert in the Schematic a particular block called FML (EQU)
- In Component Libraries window *Simplorer Elements* → *Basic Elements* → *Tools* → *Equations*
- Select the **FML: Equation (EQU)** component, drag and drop it into the Schematic




# Set Properties for FML\_INIT (ICA) Block

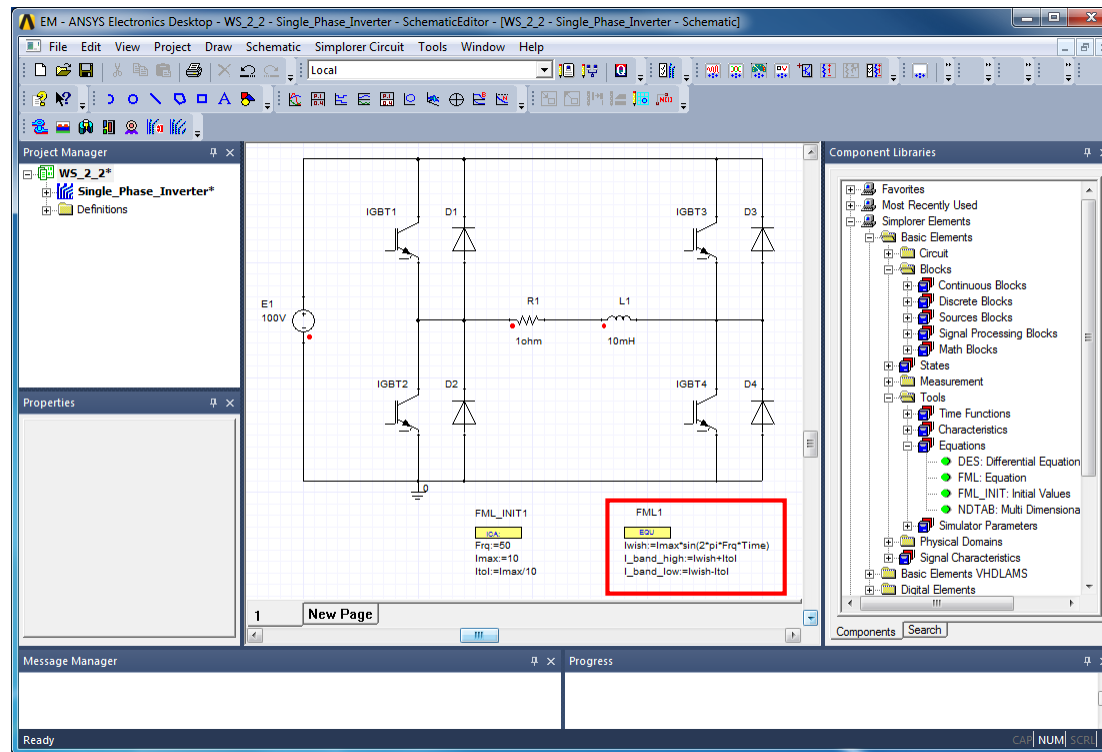
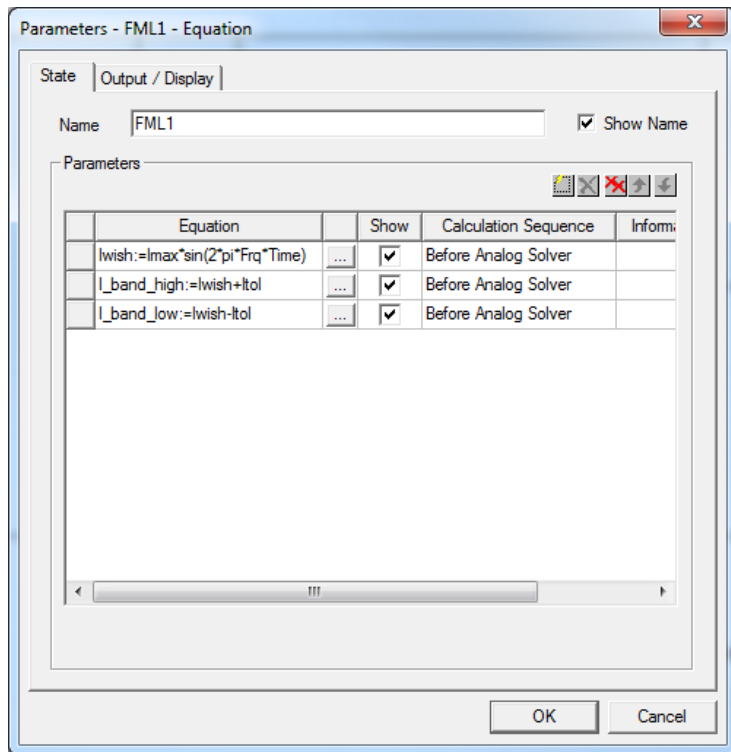
- Double click on the FML\_INIT (ICA) Block
- On the blank Parameters window, click the icon  to insert a new line
- Type  $\text{Frq}=50$  in the Equation field and check **Show**
- Insert a new line, type  $\text{Imax}=10$  in the Equation field and check **Show**
- Insert a new line, type  $\text{Itol}=\text{Imax}/10$  in the Equation field, check **Show** and Press **OK**
- Note that  $\text{Imax}$  must be defined before  $\text{Itol}$





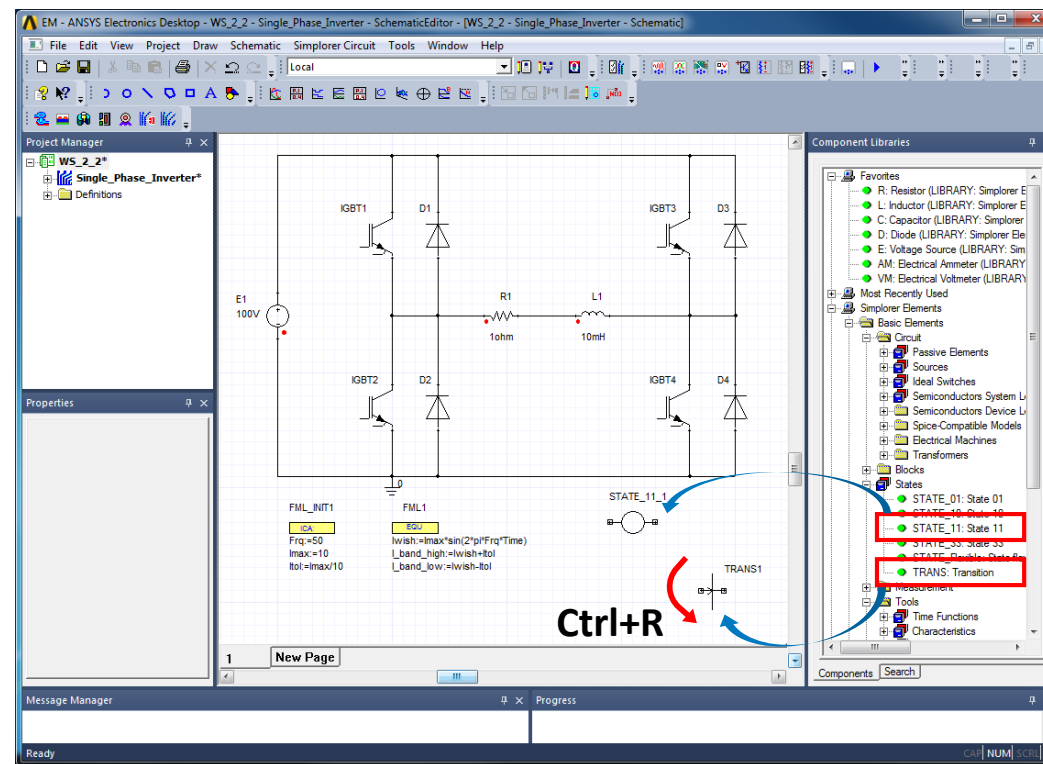
# Set Properties for FML (EQU) Block

- Double click on the FML (EQU) Block
- On the blank Parameters window, click the icon  to insert a new line
- Type  $I_{wish} = I_{max} \sin(2 \pi \text{Frq} \cdot \text{time})$  in the Equation field and check **Show**
- Insert a new line, type  $I_{band\_high} = I_{wish} + I_{tol}$  in the Equation field and check **Show**
- Insert a new line, type  $I_{band\_low} = I_{wish} - I_{tol}$  in the Equation field, check **Show** and Press **OK**
- Note that  $I_{wish}$  must be defined before  $I_{band\_high}$  and  $I_{band\_low}$ . **Time** is an internal variable




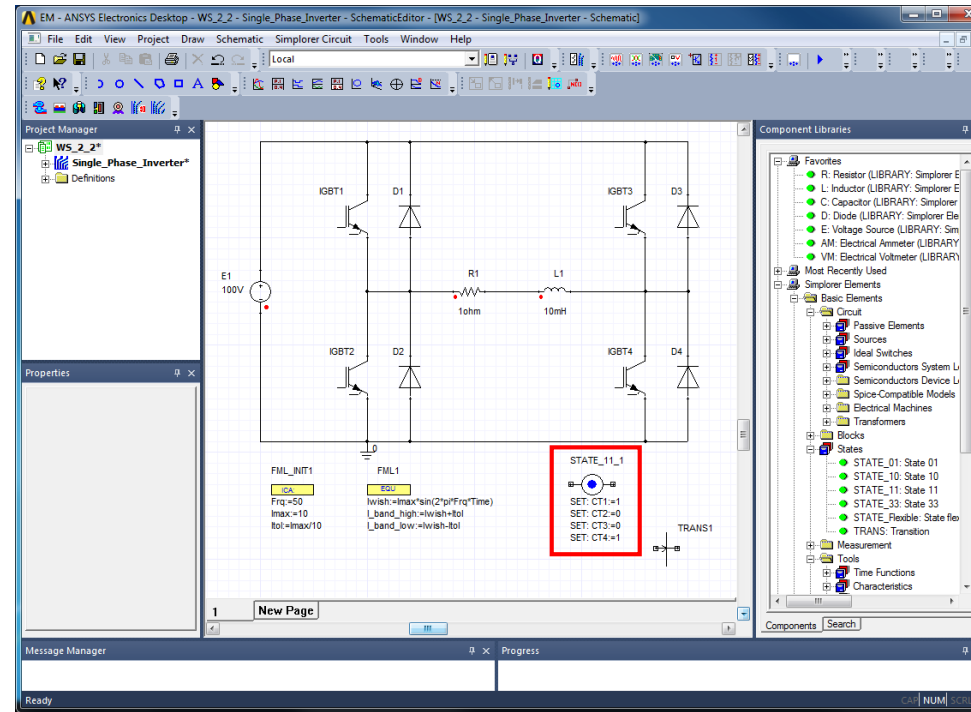
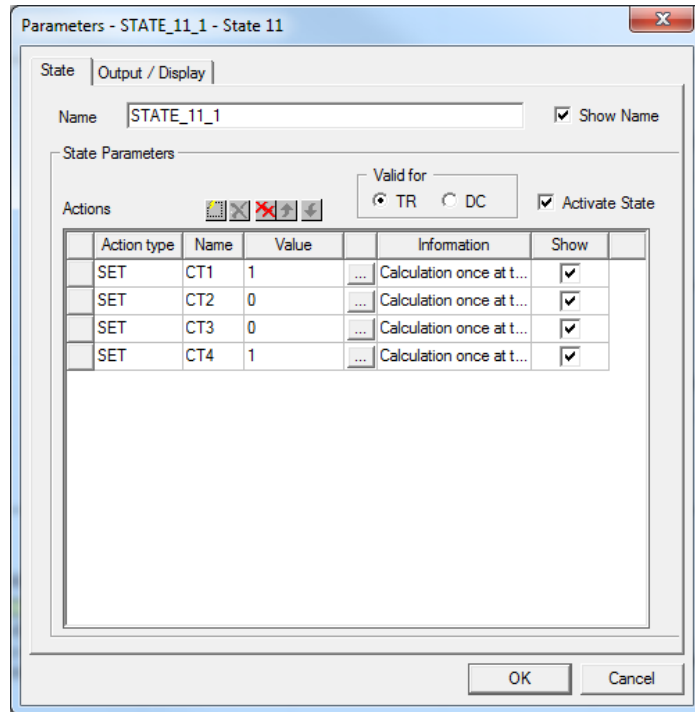
# Insert States

- We need to define the control logic for the four IGBTs
- In particular we need to define four control signals to be sent to the four IGBT Gates (Control Pins)
- To do that it is helpful to use the so called States Machines or States Diagrams
- In Component Libraries window *Simplorer Elements* → *Basic Elements* → *States*
- Select the **STATE\_11** component, drag and drop it into the Schematic
- Select the **TRANS** (Transition) component, drag and drop it into the Schematic and rotate it



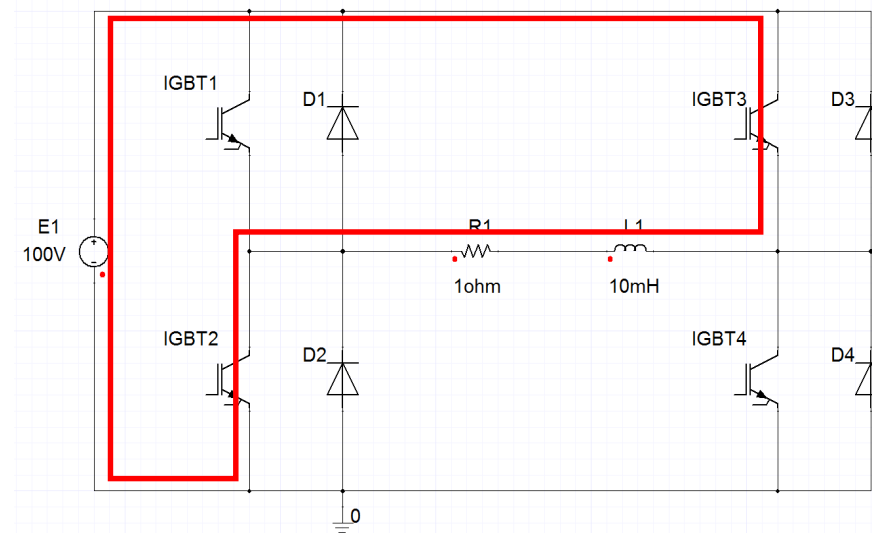
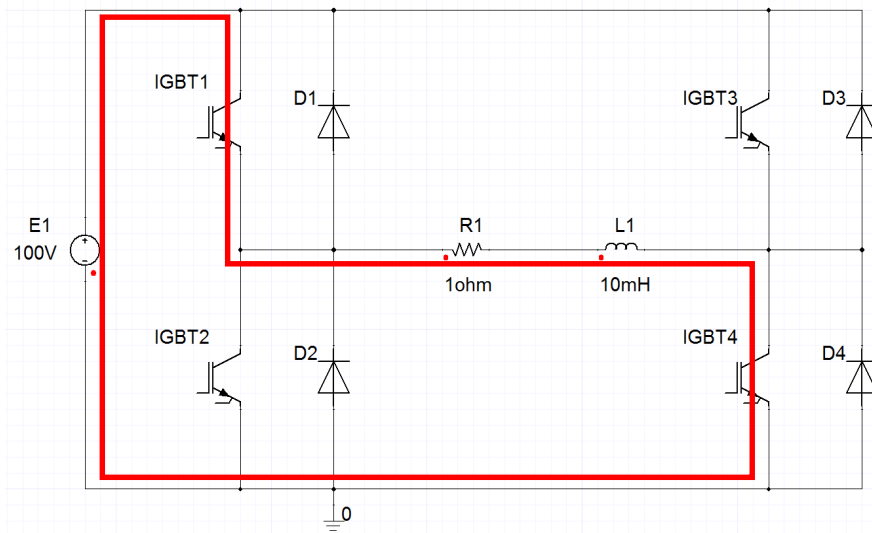
# Set Properties for STATE\_11\_1 Component

- Double click on the STATE\_11\_1 Component
- On the blank Parameters window, click the icon  to insert a new line
- Action Type: **SET**, then **CT1** in the Name field and **1** in the Value Field
- Insert a new line, type **CT2** in the Name field and **0** in the Value Field
- Insert a new line, type **CT3** in the Name field and **0** in the Value Field
- Insert a new line, type **CT4** in the Name field and **1** in the Value Field
- Check **Show** for all the four entries and check also **Activate State**. Press **OK**



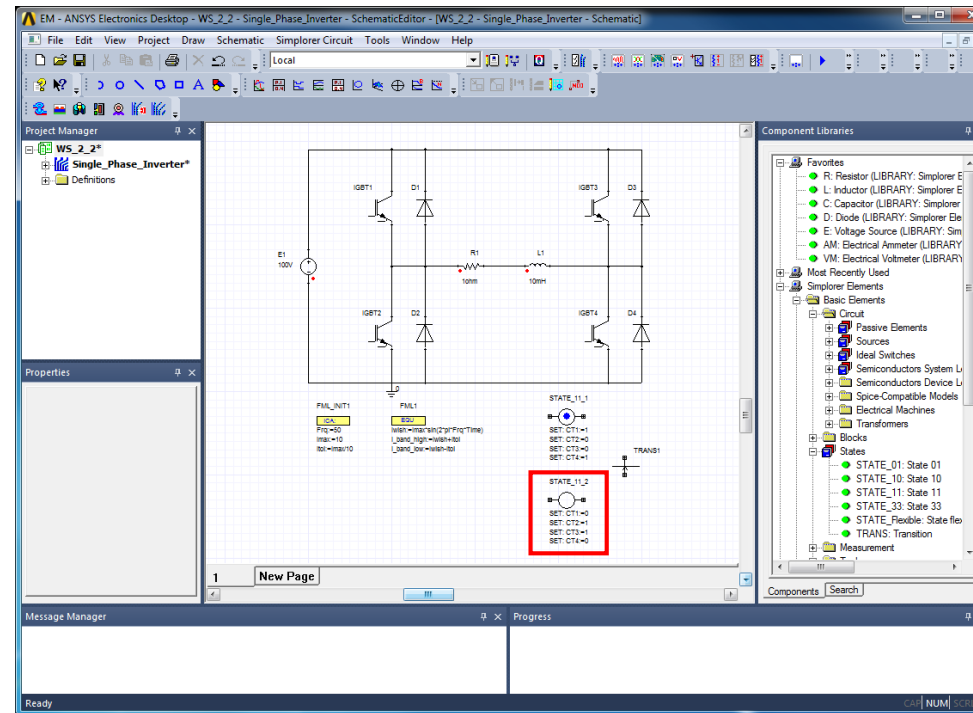
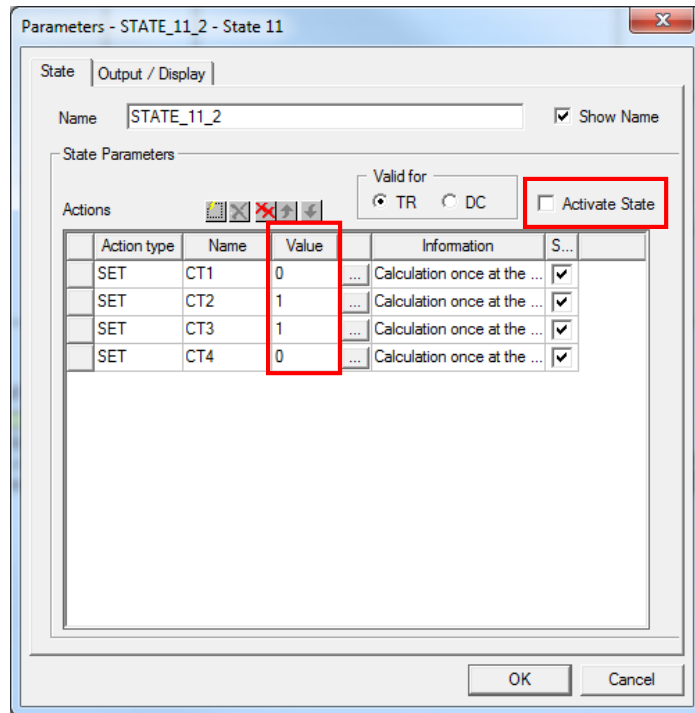
# Comments on the Control Strategy and Signals

- We just created four new signals CT1, CT2, CT3, CT4. Those signals are intended to control the four IGBTs, respectively CT1-IGBT1, CT2-IGBT2, CT3-IGBT3, CT4-IGBT4
- It is mandatory that IGBT1 and IGBT2 are not closed at the same time, otherwise a short-circuit takes place. The same can be said for IGBT3/IGBT4
- In that perspective, when IGBT1 is ON (CT1 = 1), IGBT2 must be OFF (CT2 = 0) and on the other way around. The same can be said for IGBT3/IGBT4 and CT3/CT4
- The signals values set in STATE\_11\_1 describe the condition of E1 feeding the load as described in the figure on the left
- To obtain the opposite feeding condition, it is necessary to create a further STATE block, enabling the load to be fed as described in the figure on the right



# Copy the STATE\_11 Component and Change Properties

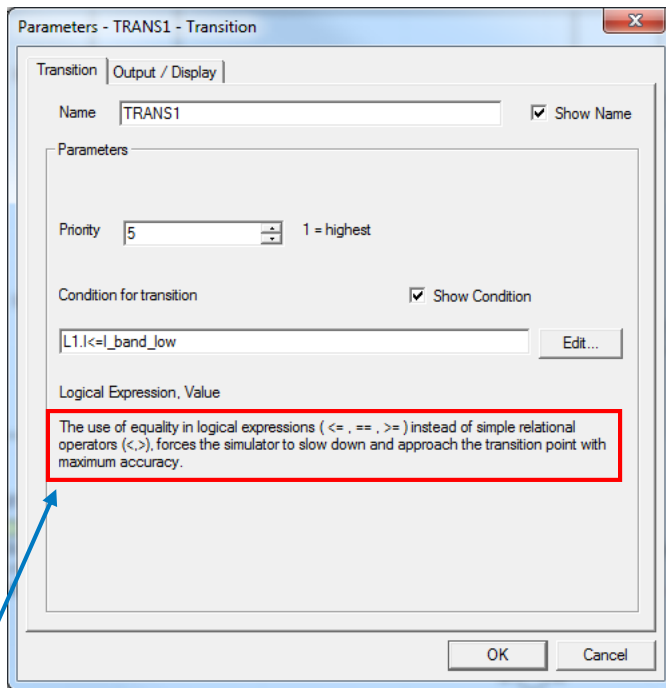
- Select the STATE\_11\_1 Component, **Ctrl+C** → **Ctrl+V**
- Double click on the just created STATE\_11\_2 component and change the Values as follows
- **CT1 = 0**
- **CT2 = 1**
- **CT3 = 1**
- **CT4 = 0**
- uncheck **Activate State**, Press **OK**



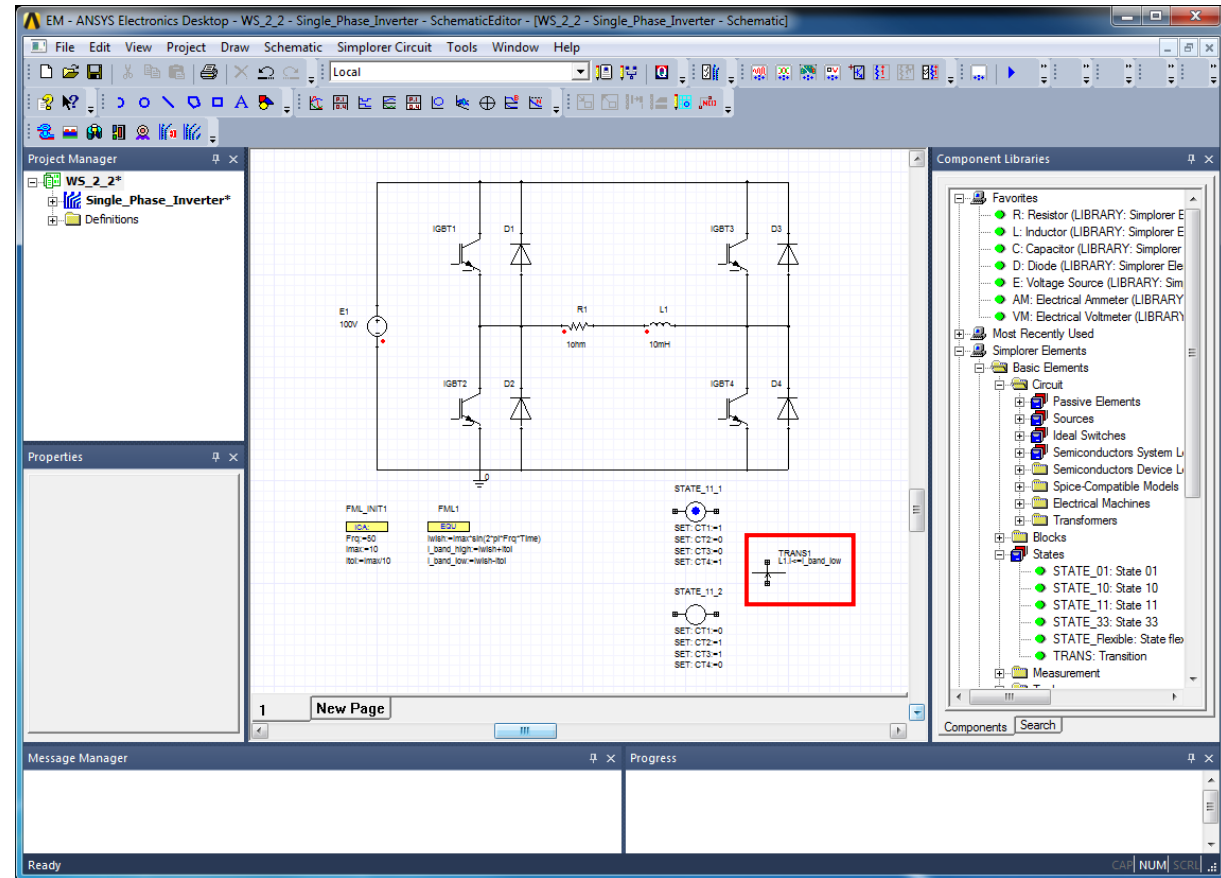


# Set Properties for TRANS1 Component

- Select the **TRANS1** Component,
- In the Condition for transition entry field, type **L1.I <= I\_band\_low**
- Check **Show Condition**
- Press **OK**

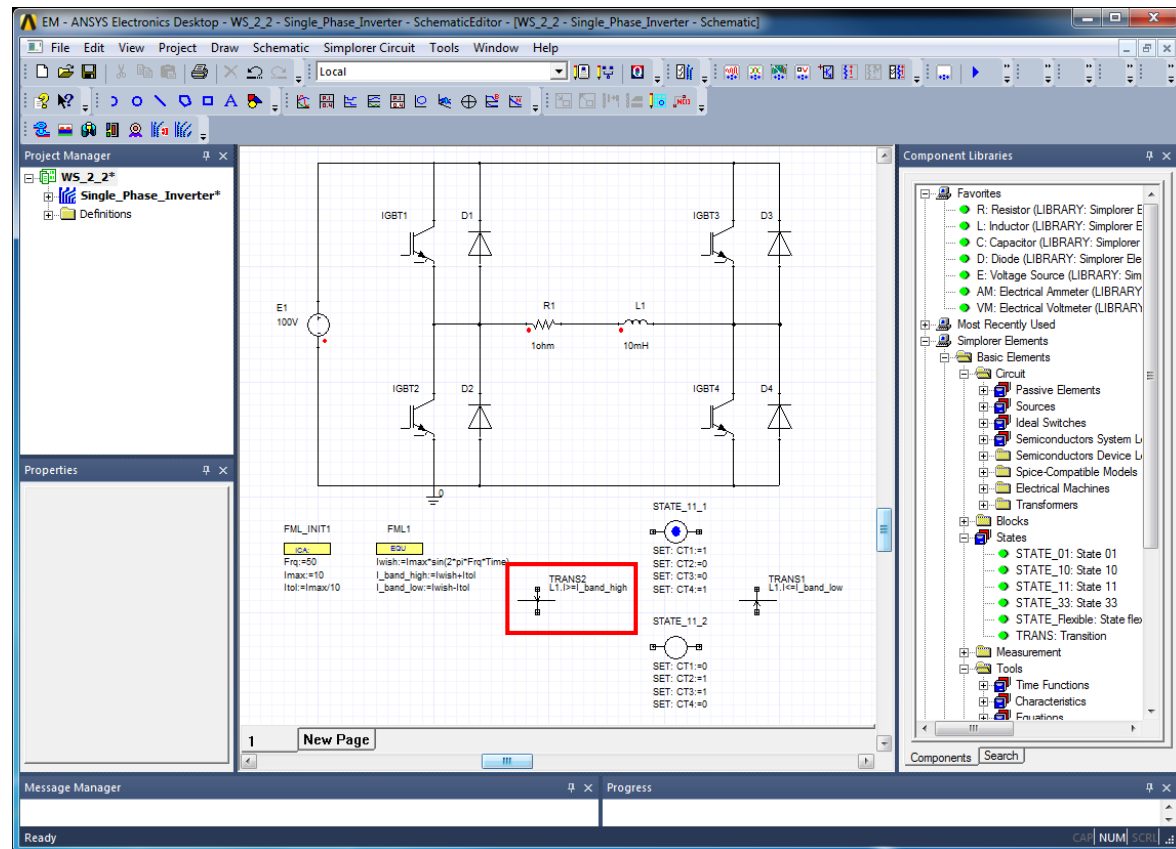
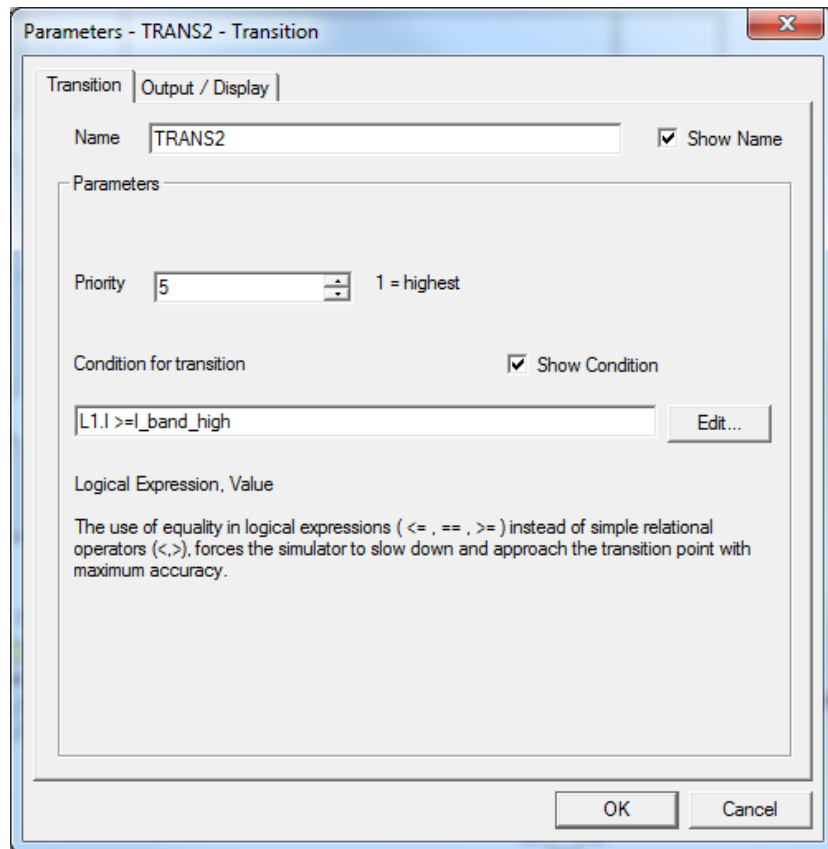


*Note the importance of using  $\geq$ ,  $\leq$  or  $=$  to achieve higher accuracy*



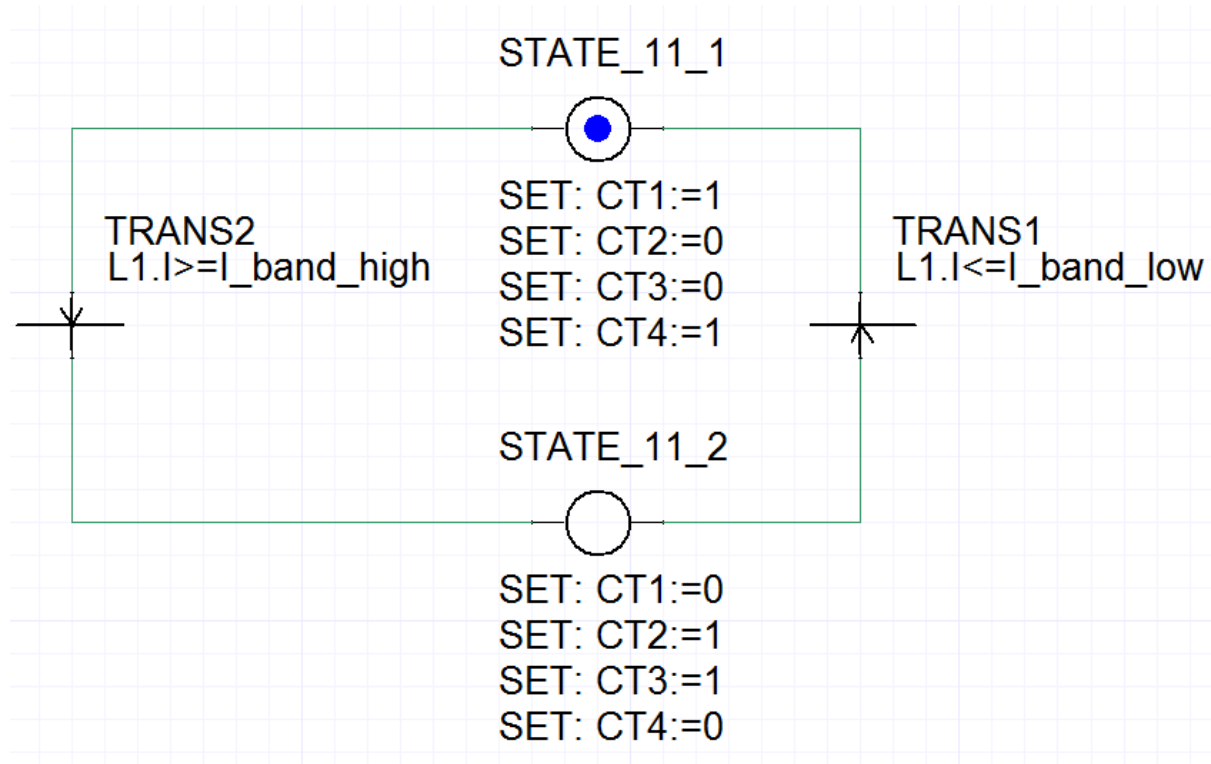
# Create a Second Transition Component and Change Properties

- Click on the TRANS1 Component, **Ctrl+C** → **Ctrl+V**
- **Rotate it twice** using CTRL+R or flip it Vertical
- In the Condition for transition entry field, type **L1.I >= I\_band\_high**
- Press **OK**



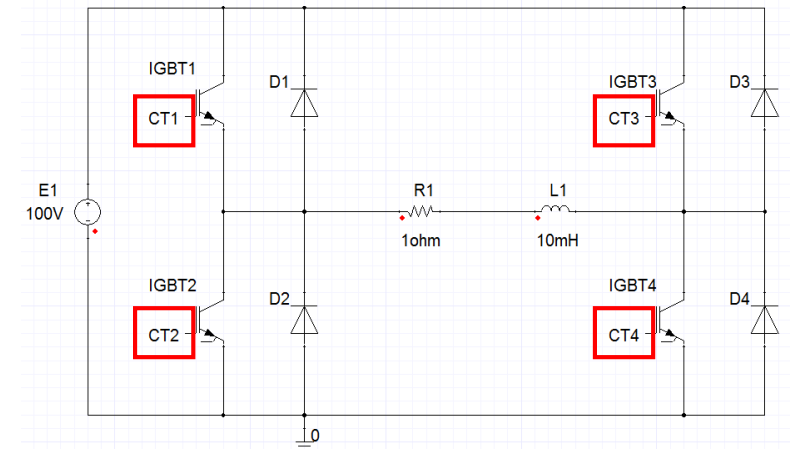
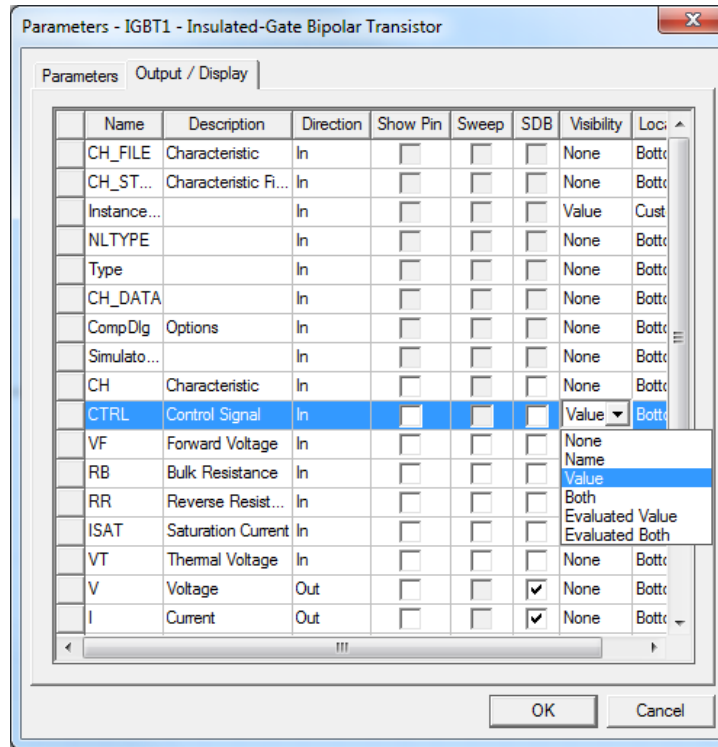
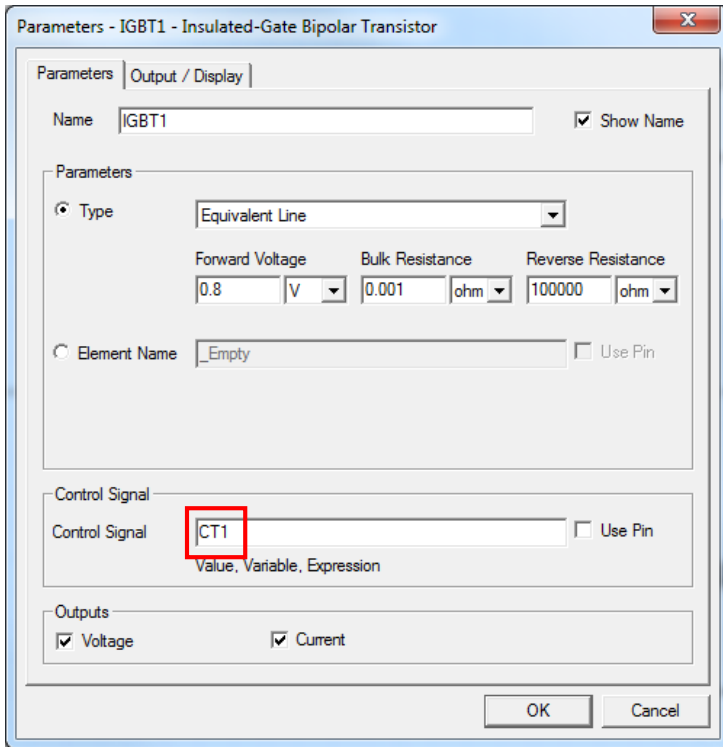
# Connect Components

- Connect the two **STATE\_11** and the two **TRANS** components so that they form a closed loop
- Note the color of connections is green, meaning that the information exchanged are only mathematical signals with no energy content



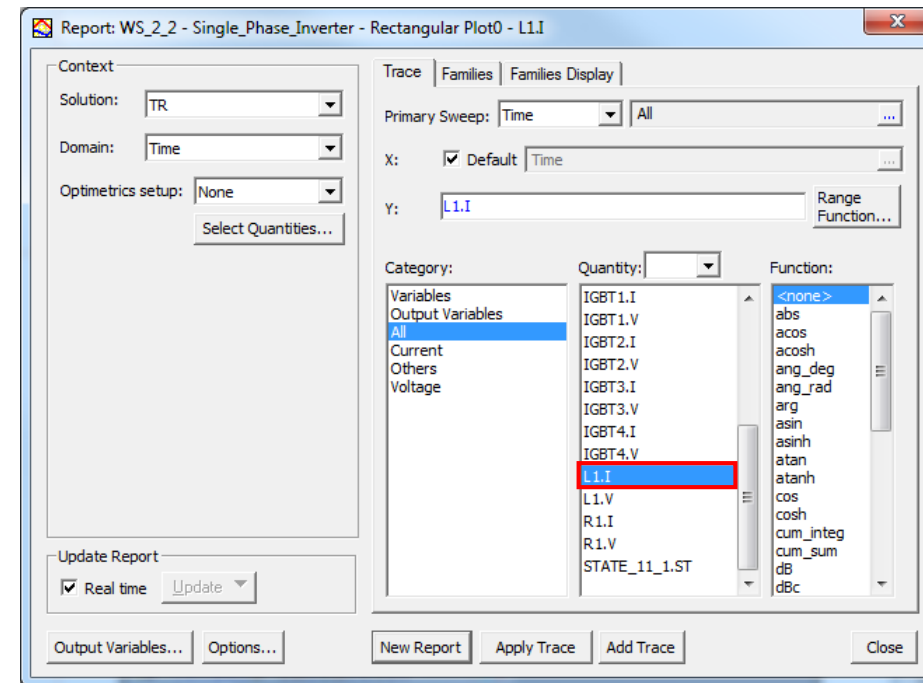
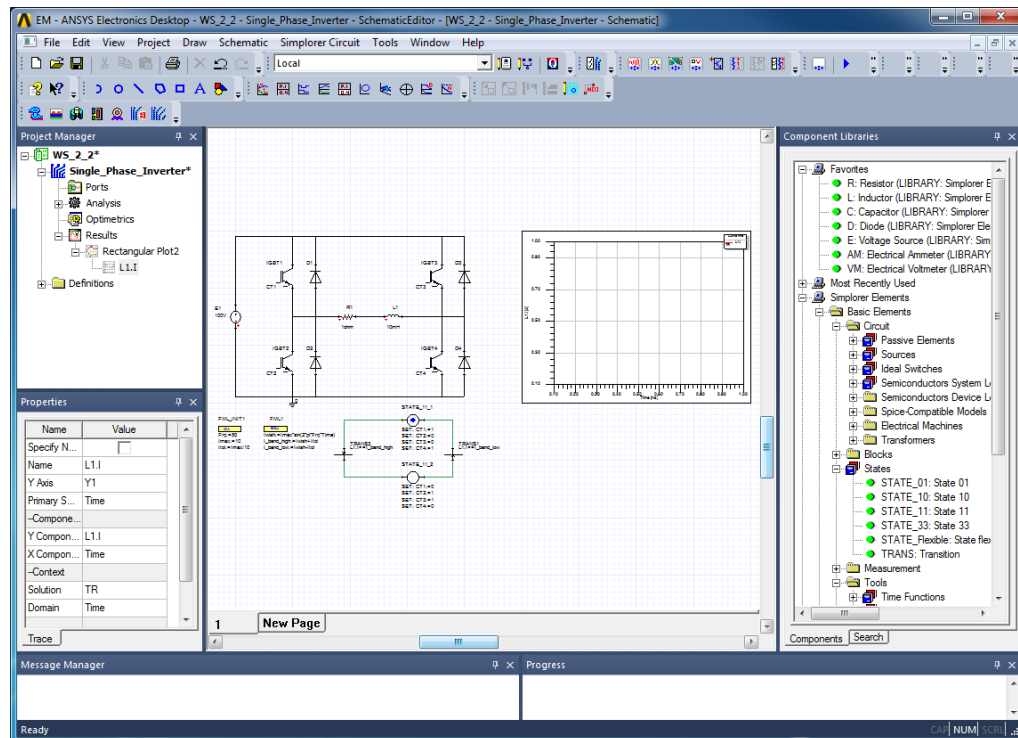
# Link Control Signals to IGBTs

- Double click on IGBT1
- Type **CT1** in the Control Signal entry field
- In the **Output/Display Tab** under Visibility, select Value for **CTRL** and press **OK**
- Repeat the procedure for IGBT2 (CT2), IGBT3 (CT3) and IGBT4 (CT4)



# Prepare the Postprocessing

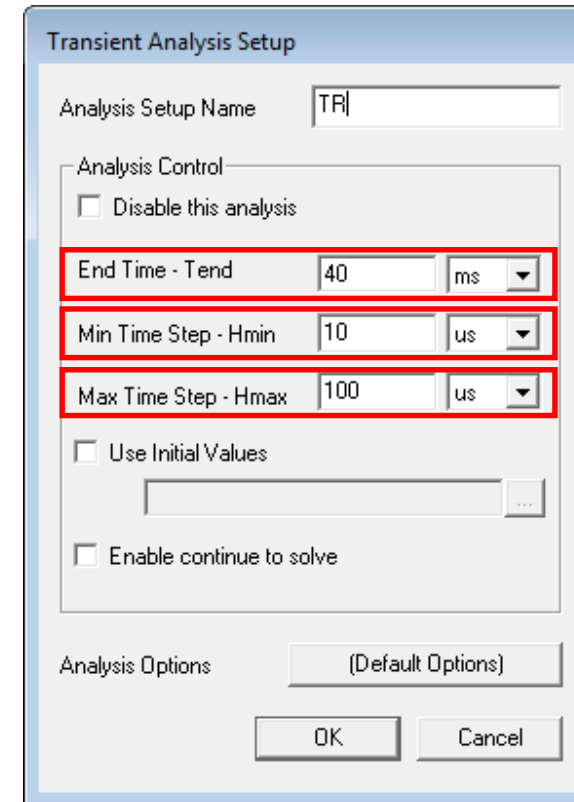
- Select the menu item **Draw** → **Report** → **Rectangular Plot** and place the plot in the Schematic, for example on the right with respect to the Circuit
- Automatically the **New Trace** window pops-up
- Select the Current flowing through the load inductance by selecting the quantity **L1.I**
- Click on the **Add Trace** button and then **Close**





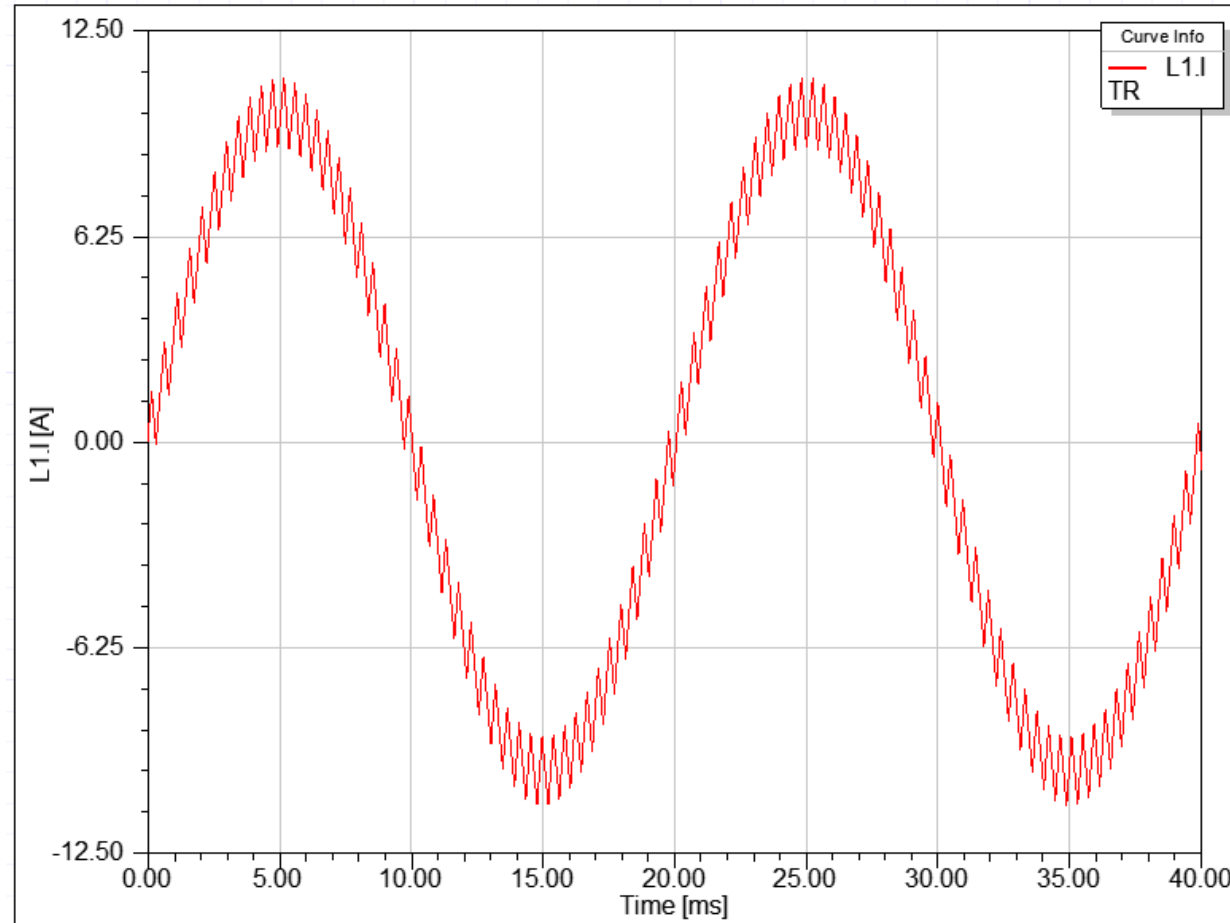
# Setup the Simulation Analysis

- In the present example is not possible to easily predict the IGBT switching frequency due to the used control technique, but since the band is not very thin, we can expect already good results with time steps in the order of tens of micro-seconds
- To reproduce 2 periods at 50 Hz, we can run the simulation for 40 ms
- In the Transient Analysis Setup window:
  - Tend: 40 ms
  - Hmin: 10  $\mu$ s
  - Hmax: 100  $\mu$ s
  - Press OK



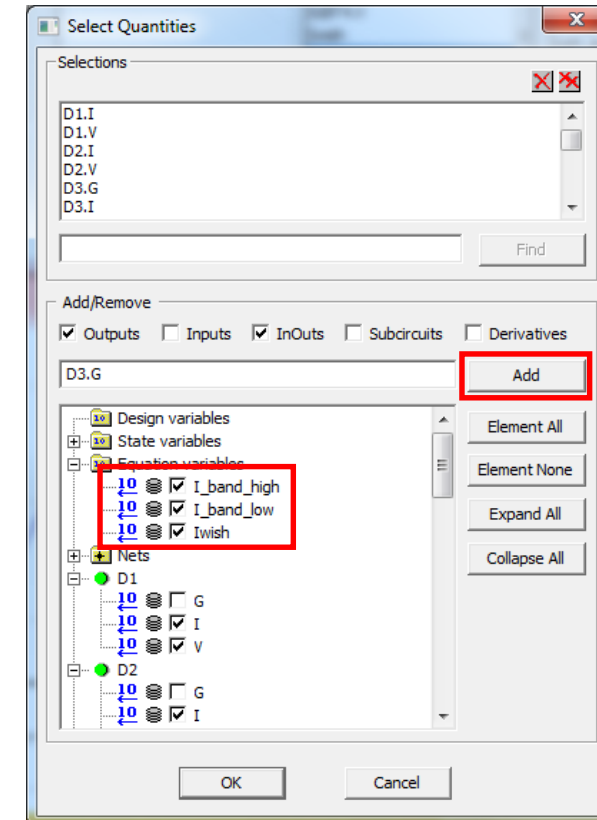
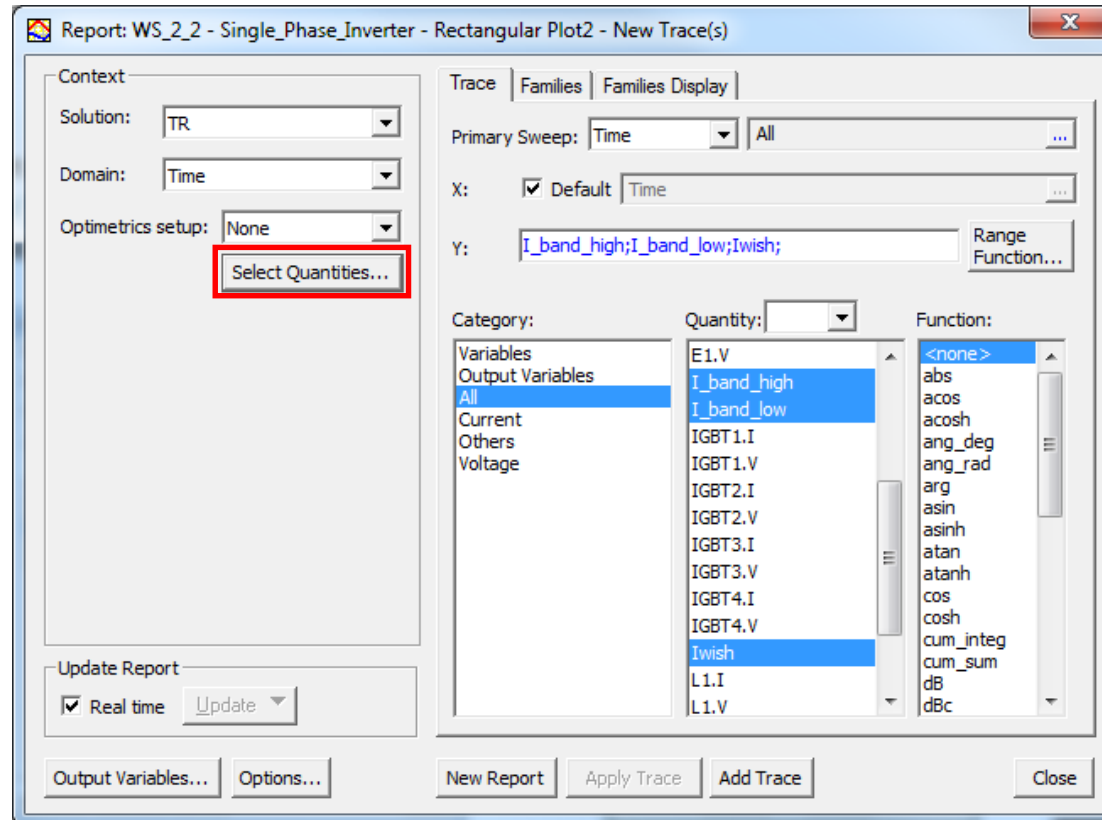
# Analyze and View Results

- Select the menu item *Simplorer Circuit* → *Analyze to run the Simulation*
- The final result for the current flowing through the R-L load should look very similar to the following Figure:



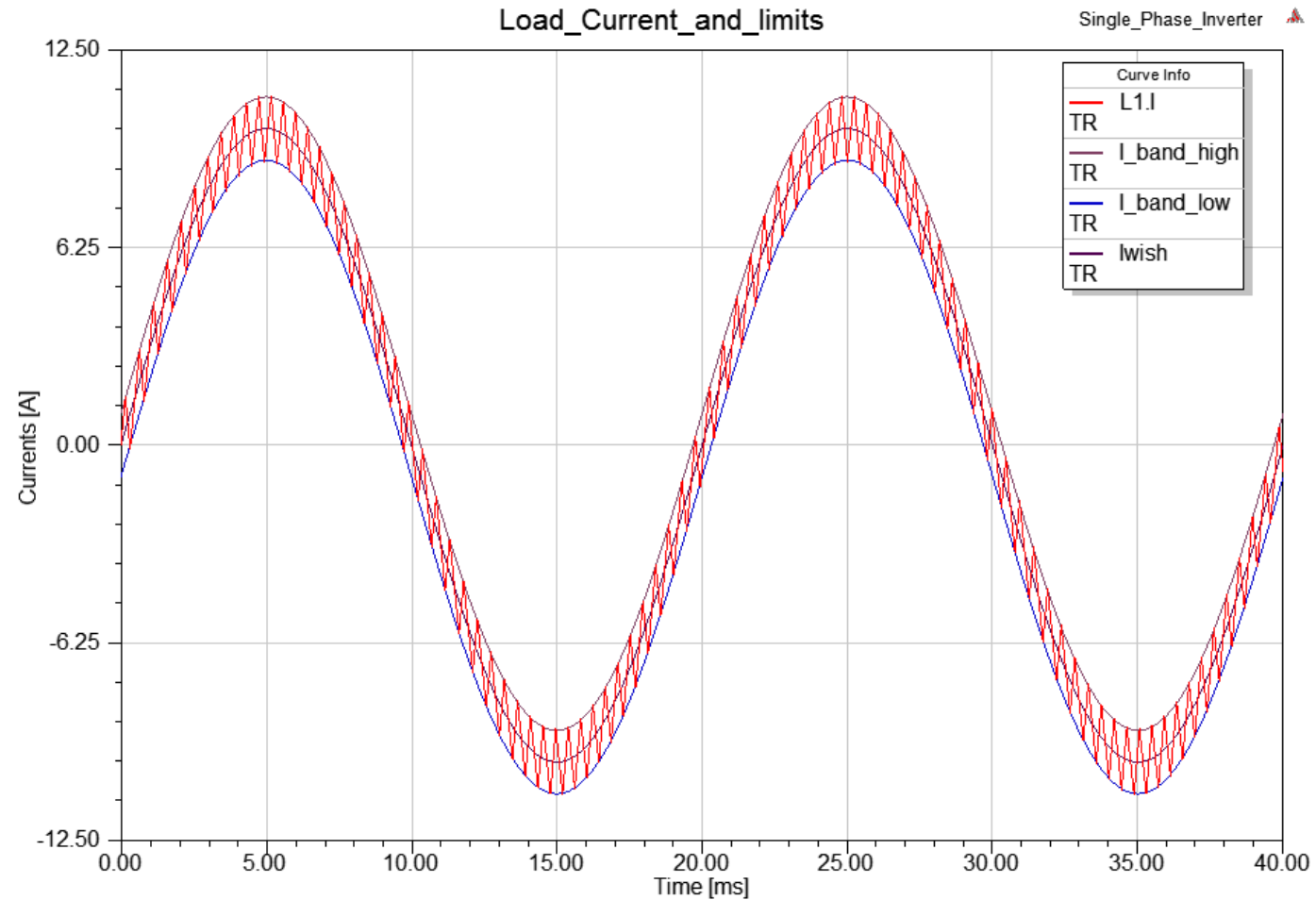
# Further Results

- Select the Plot Report **RMB** → **Modify Report**, click on **Select Quantities** button. In the Select Quantities window check ***I\_band\_high***, ***I\_band\_low*** and ***Iwish***, press **Add** and then **Add Trace**
- Run again the Simulation





# Further Results

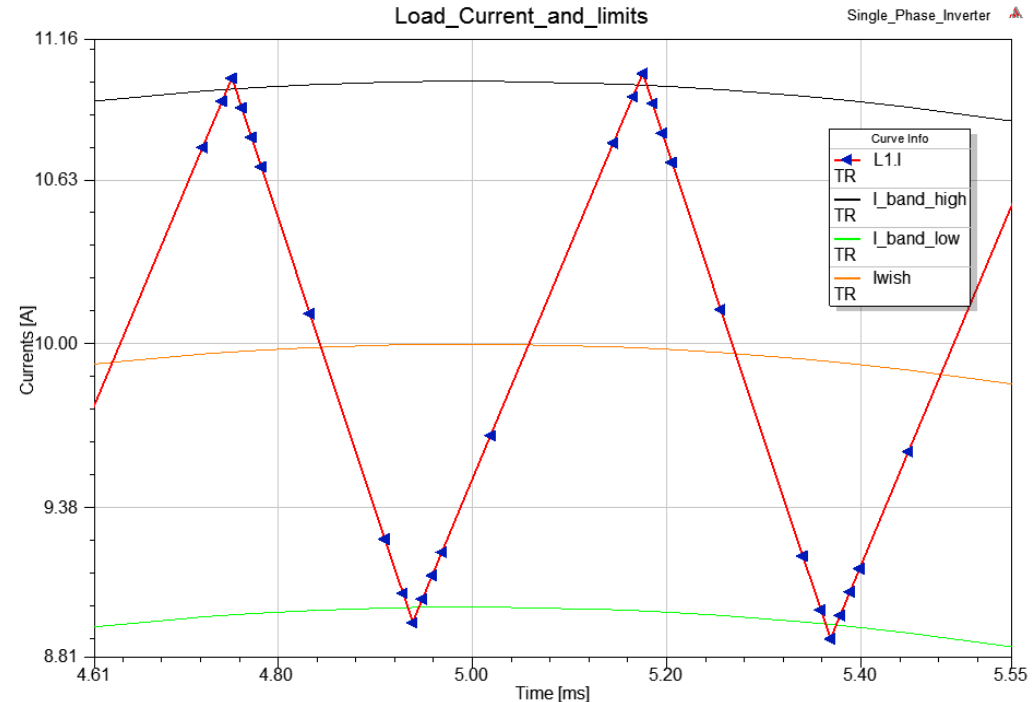
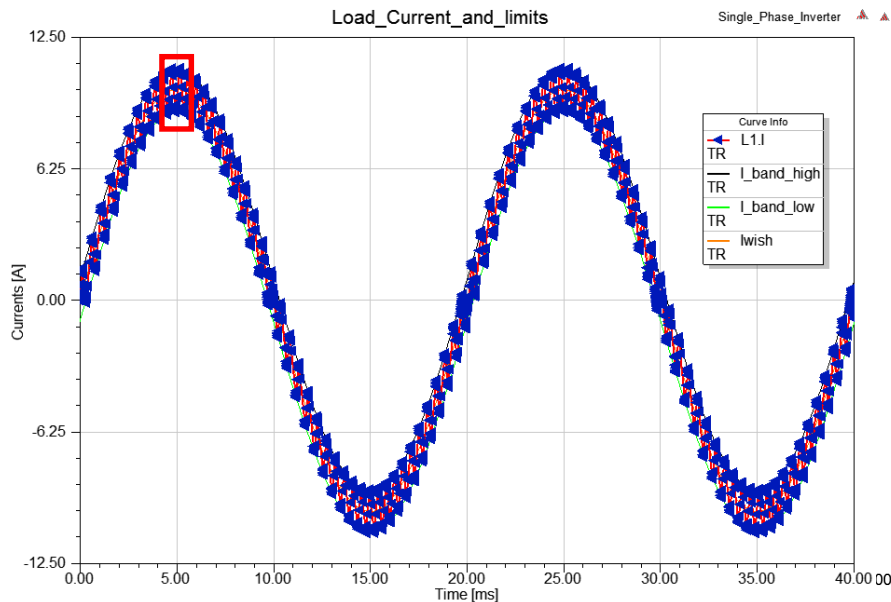
- The resulting plot should look similar to the following one:



# Further Results

- Open the previous Plot from the Report folder in Project Manager window
- Double click on the **L1.I** signal directly in the Plot
- Check **Show Symbol** and set **Symbol Frequency** to **1**. Press **OK**
- Use the icon  to zoom near one peak of the currents
- It is now possible to appreciate the reducing time step when approaching to defined limits, as set when using  $\geq$  and  $\leq$  in the transition conditions

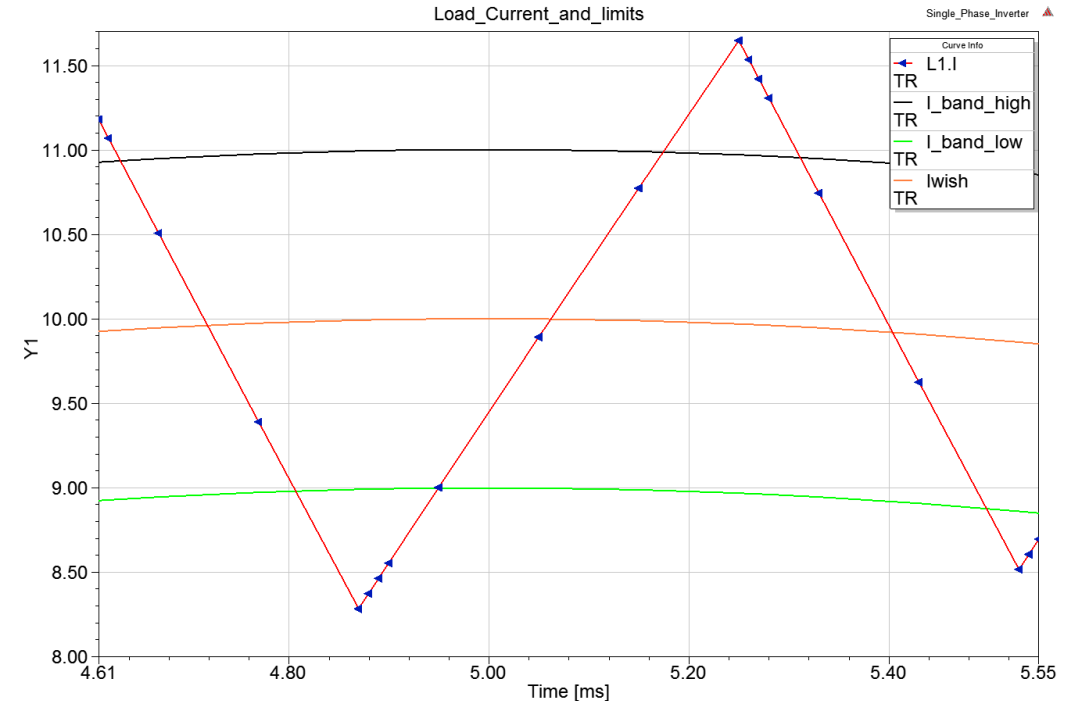
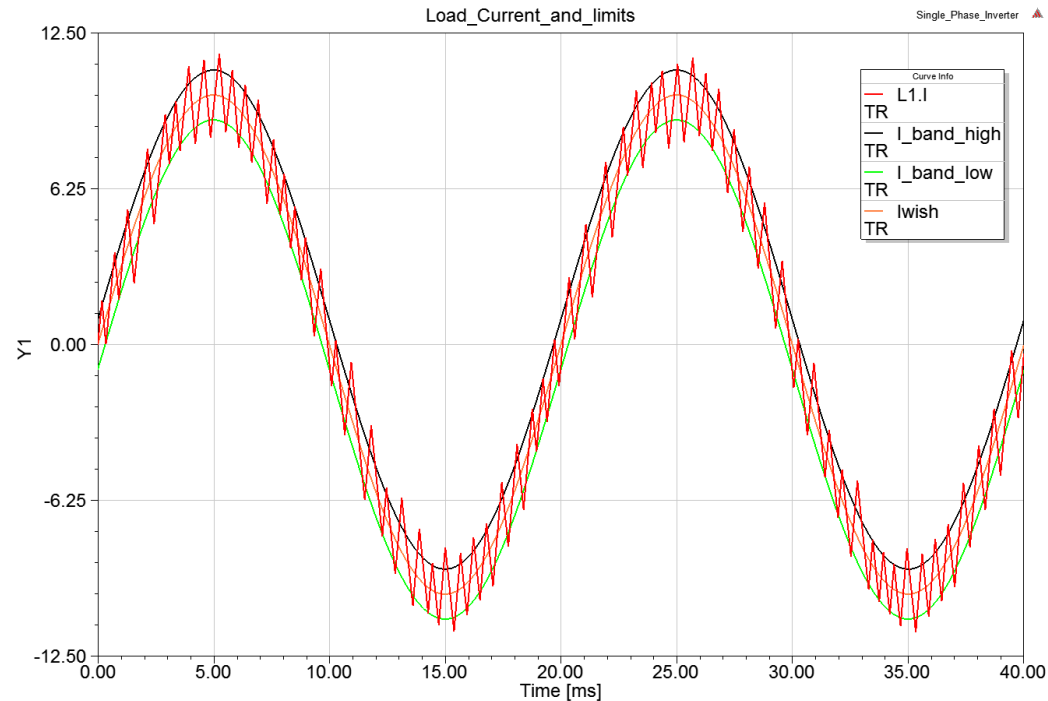
Name	Value
Name	L1.I:Curve1
Color	
Line Style	Solid
Line Width	1
Trace Type	Continuous
Show Symbol	<input checked="" type="checkbox"/>
Symbol Frequency	1
Symbol Style	HorizontalLeft Triangle





# Further Results

- Go back to the schematic and change the Transition conditions from  $\geq$  and  $\leq$  to  $>$  and  $<$
- Run again the simulation
- Compare the obtained results with the previous ones: it is clear how the transition conditions settings influence the solution accuracy



# Saving the Project

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