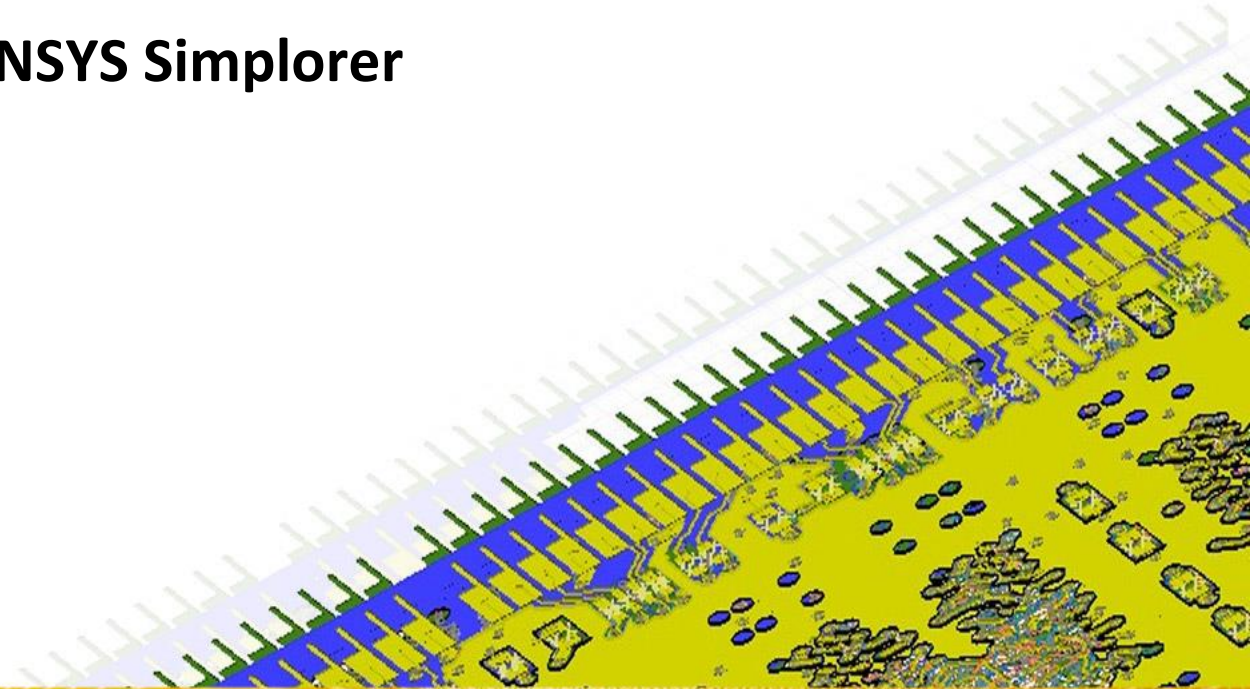




Workshop 3.2: Sub Circuits



Introduction to ANSYS Simplorer

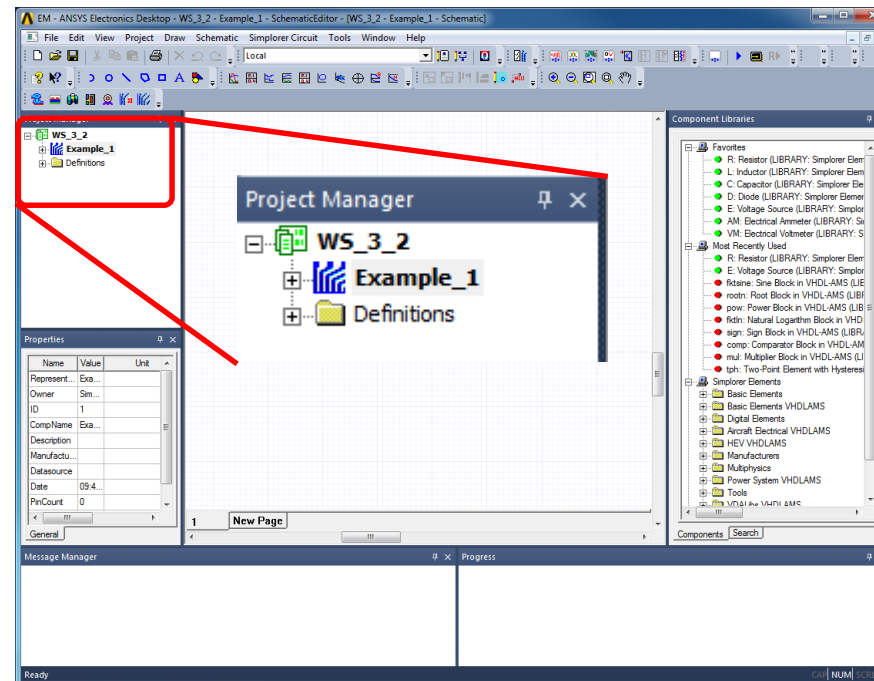
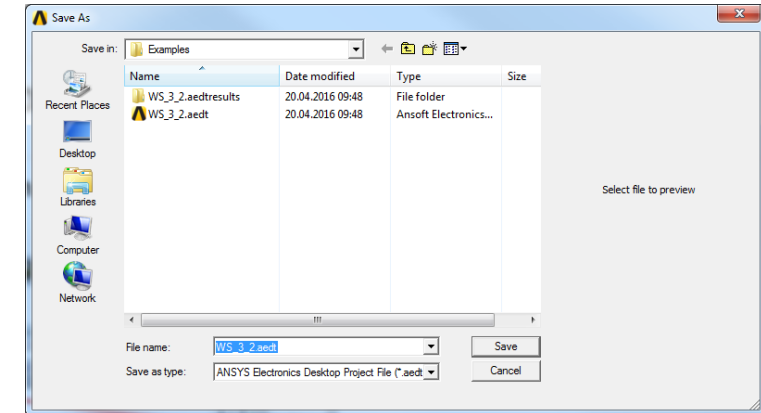


Overview

- **Sub Circuits**
 - In this example we will investigate how to create Sub-circuit and how to transfer information between sub-levels
 - In particular we will learn
 - How to insert a Sub-circuit
 - How to interface Conservative and non-conservative quantities through the Ports
 - How to pass a top-level local variable to a component placed in a sub-circuit during a Parametric Sweep
 - How to edit the sub-circuit Symbol and change it

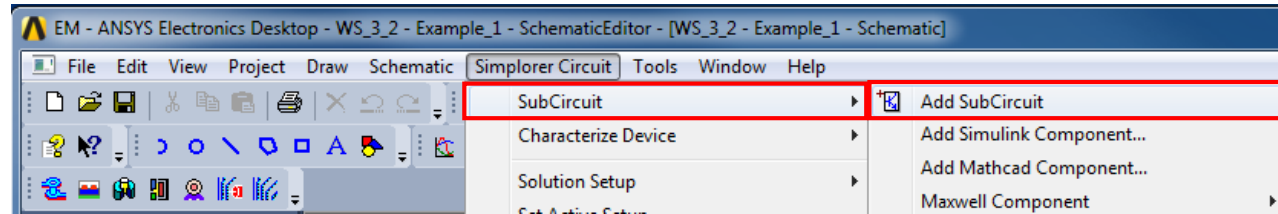
Insert a Simplorer Design

- Launch the Electronics Desktop 2016
 - Save the Project as **WS_3_2.aedt**
 - Insert a Simplorer Design using the icon 
 - Rename the Design as **Example_1**
 - Save again the project using the icon 

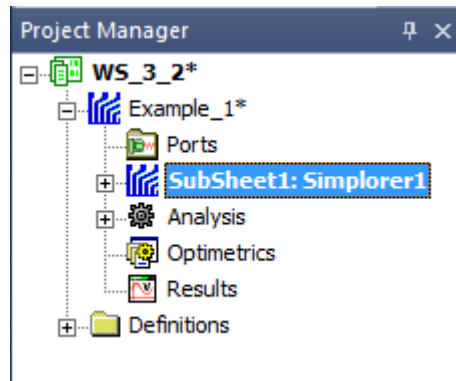


Create Sub Circuit

- Select the menu item *Simplorer Circuit* → *SubCircuit* → *Add SubCircuit*



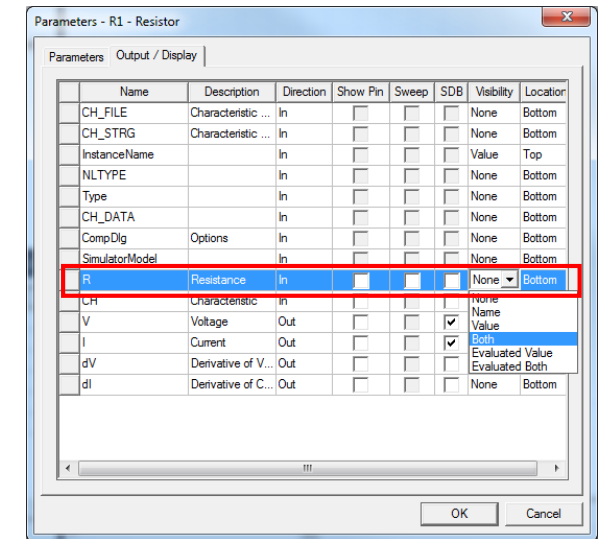
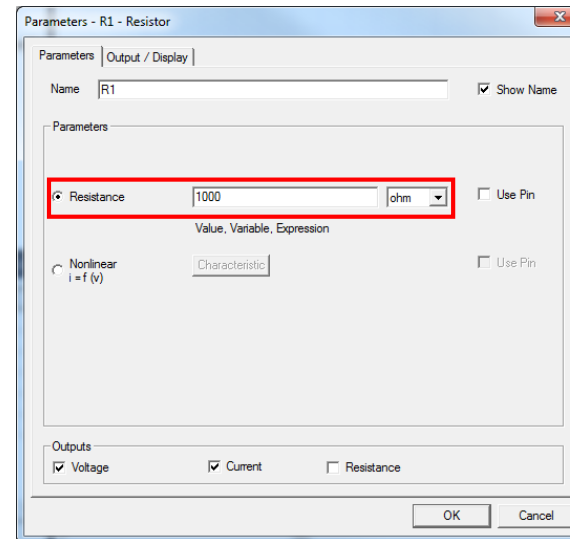
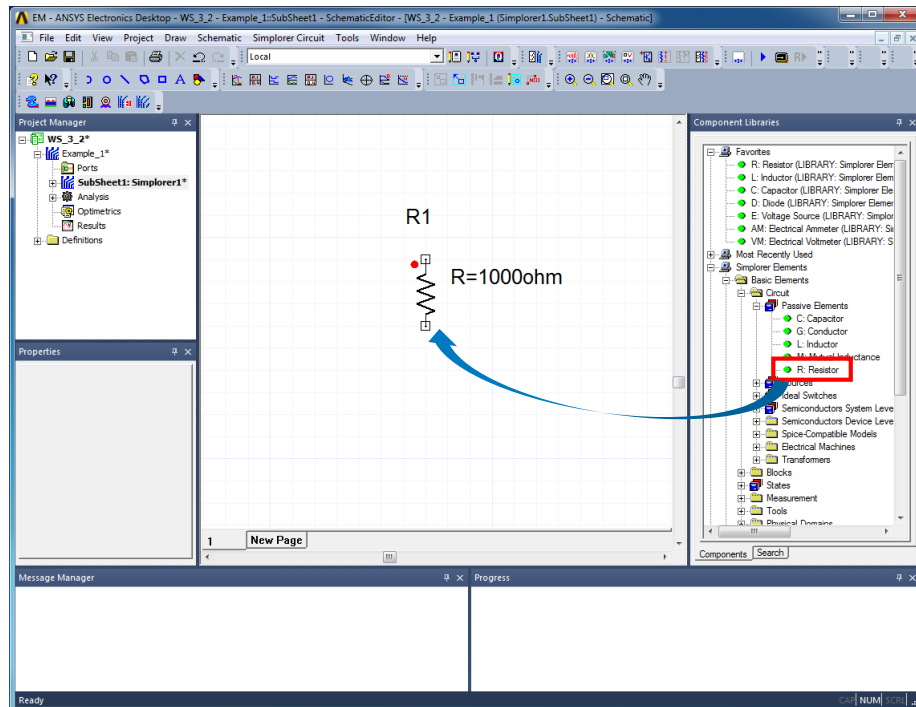
- Automatically in the Project Manager Window a new sub-sheet appears and the schematic window is already active on the new sub-sheet



Insert Components

- Resistor

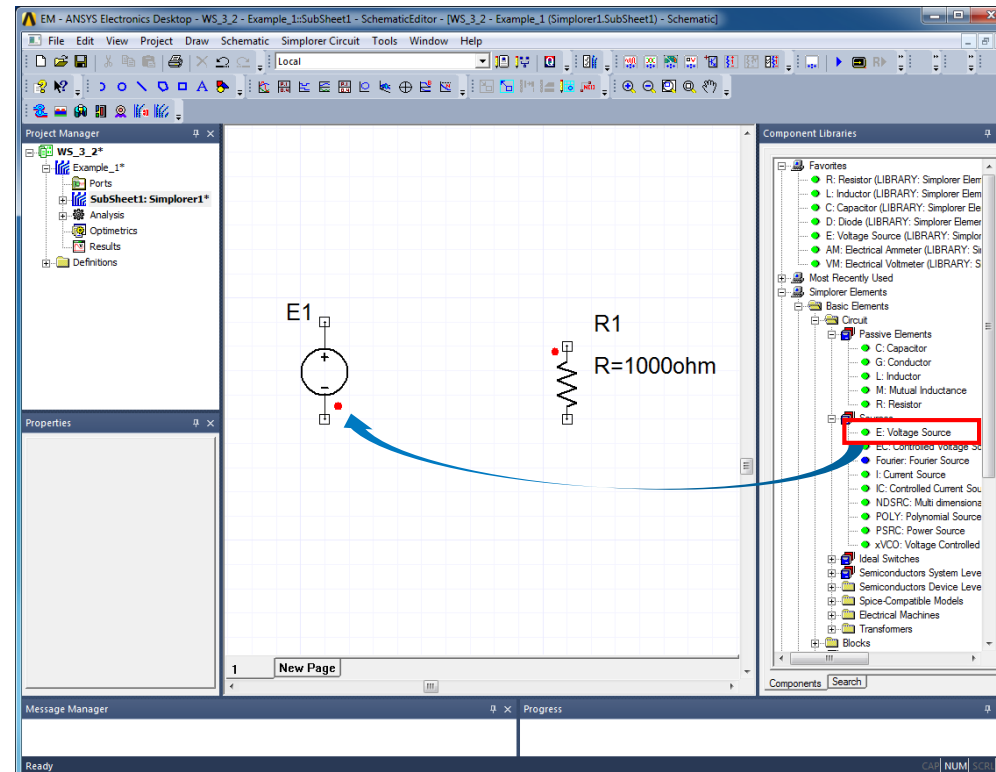
- In Component Libraries window *Simplorer Elements* → *Basic Elements* → *Circuit* → *Passive Elements*
- Select the **R: Resistor** and drag and drop it into the Schematic. Press **Esc** key to exit the insert mode
- Double click on the Resistor, leave for the moment the value as it is
- In the **Output/Display Tab** under Visibility, select **Both** for Resistance



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

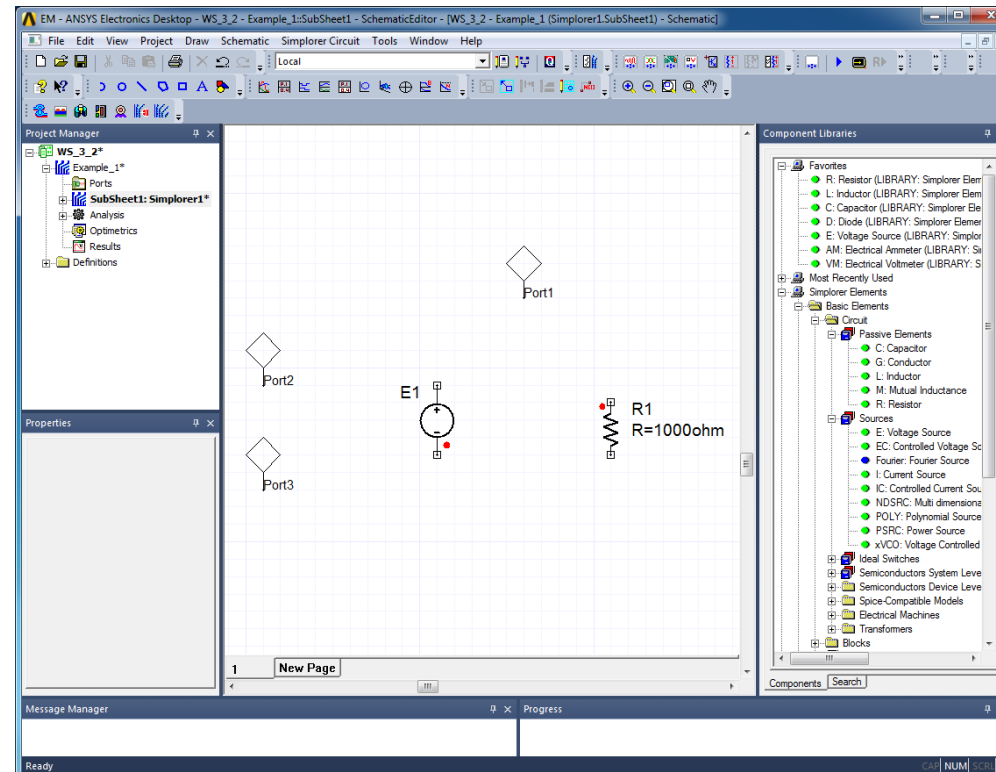
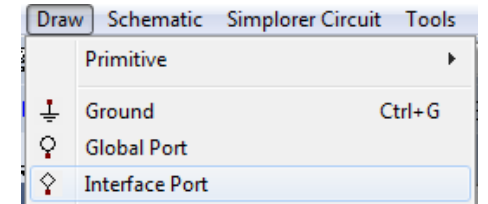


Note: it is good practice to select “spice compatible” when inserting voltage sources


Insert Components

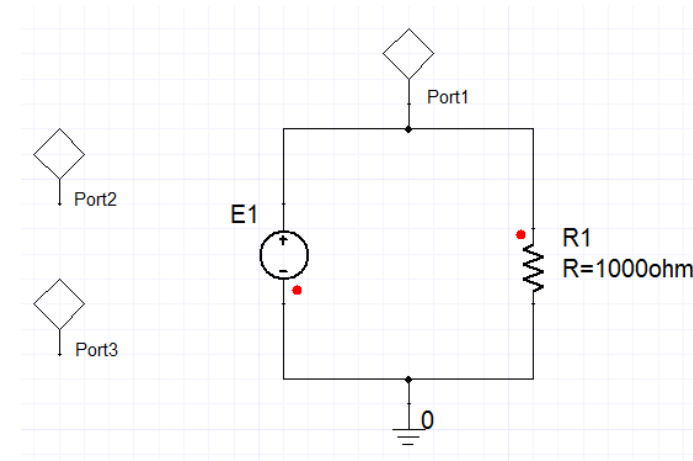
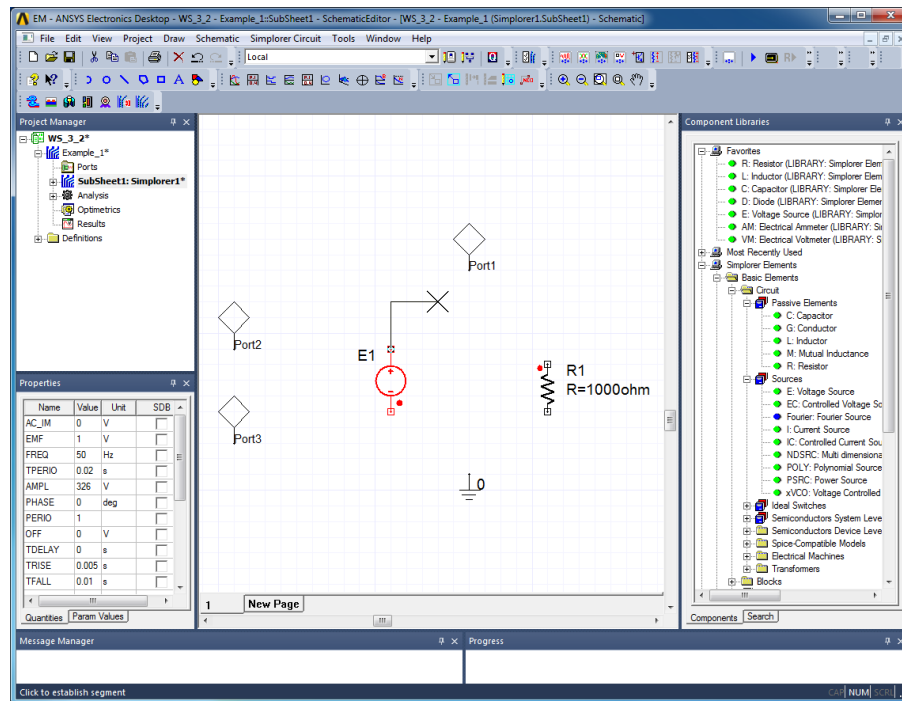
- Interface Ports

- Select the menu item **Draw** → **Interface Port**
- Place the port 3 times into the Schematic. Press **Esc** key to exit the insert mode
- Use the shortcut **Ctrl+D** to fit the view



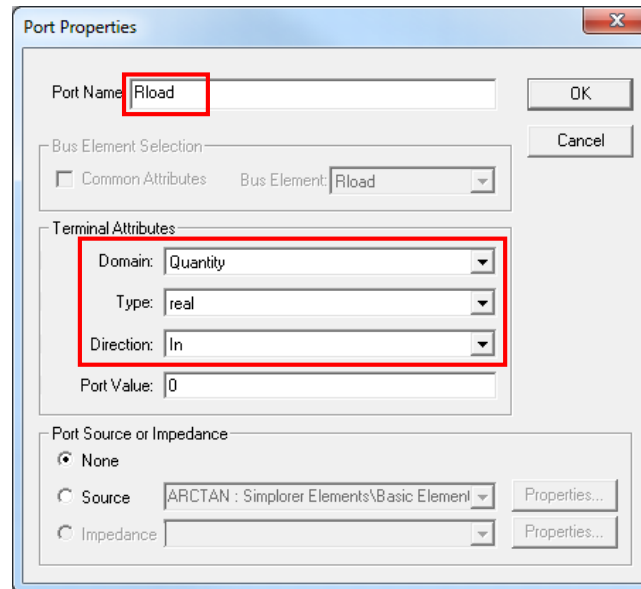
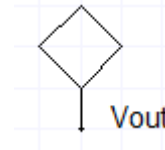
Connect Components

- 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
- Relax the grid setup for more versatile placements of values on schematic through menu item **Schematic → Grid setup...** and un-check **Snap Text and Graphics to Grid**



Set Port Properties

- **Port1**
 - Double click on **Port1** and change the name to **Vout**
- **Port2**
 - Double click on **Port2**
 - change the name to **Rload**
 - Domain: **Quantity**
 - Type: **real**
 - Direction: **In**
 - Press **OK**
- **Port3**
 - Double click on **Port3**
 - change the name to **lload**
 - Domain: **Quantity**
 - Type: **real**
 - Direction: **Out**
 - Value: **R1.I**
 - Press **OK**



Port Properties

Port Name: **Rload**

Bus Element Selection: ☐ Common Attributes Bus Element: **Rload**

Terminal Attributes:

Domain: **Quantity**

Type: **real**

Direction: **In**

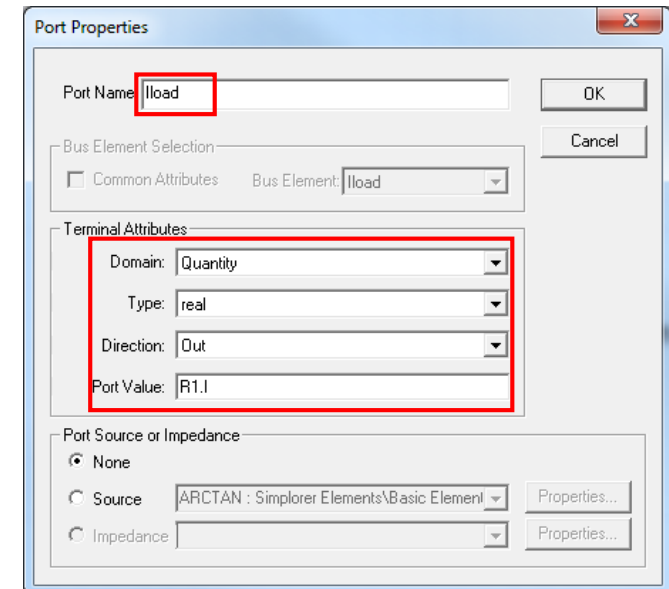
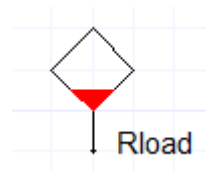
Port Value: **0**

Port Source or Impedance:

☒ None

☐ Source: **ARCTAN : Simplorer Elements\Basic Element** Properties...

☐ Impedance: Properties...



Port Properties

Port Name: **lload**

Bus Element Selection: ☐ Common Attributes Bus Element: **lload**

Terminal Attributes:

Domain: **Quantity**

Type: **real**

Direction: **Out**

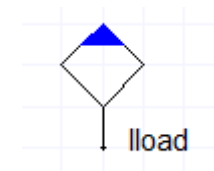
Port Value: **R1.I**

Port Source or Impedance:

☒ None

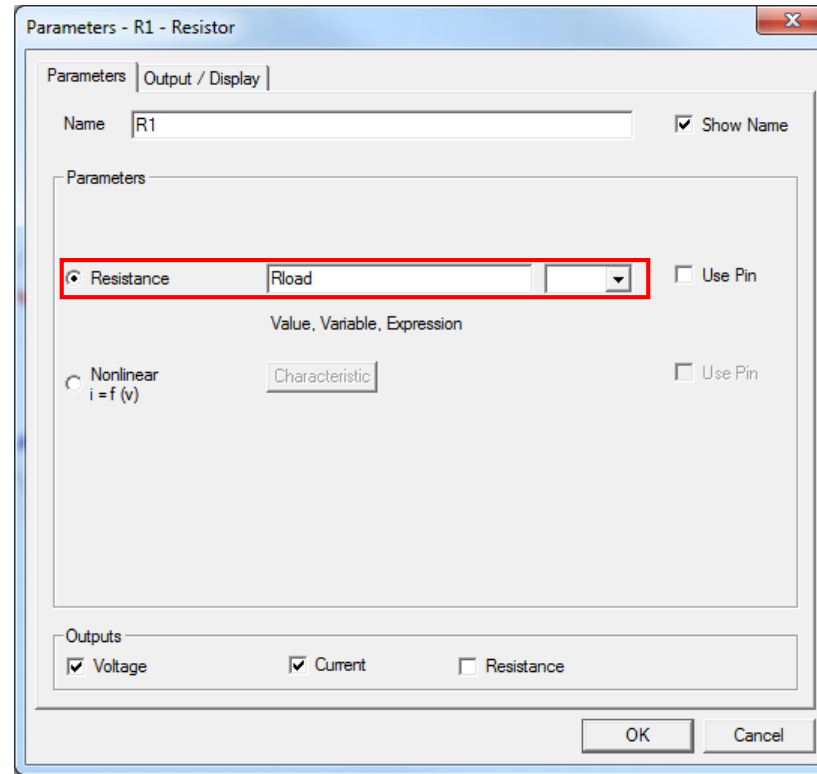
☐ Source: **ARCTAN : Simplorer Elements\Basic Element** Properties...

☐ Impedance: Properties...



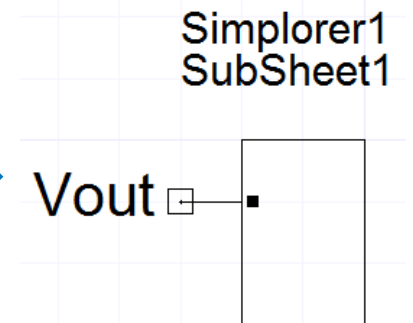
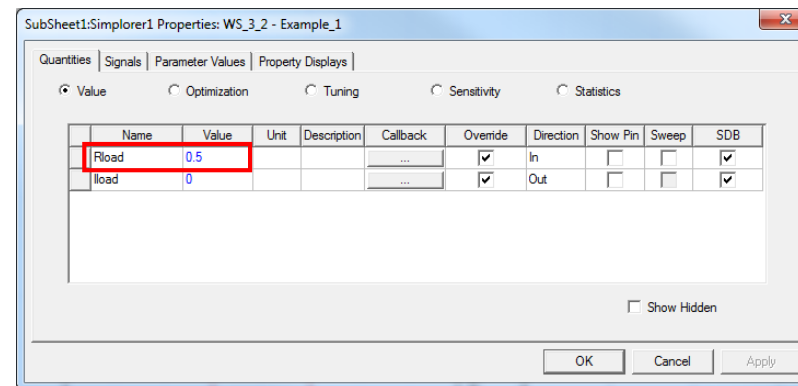
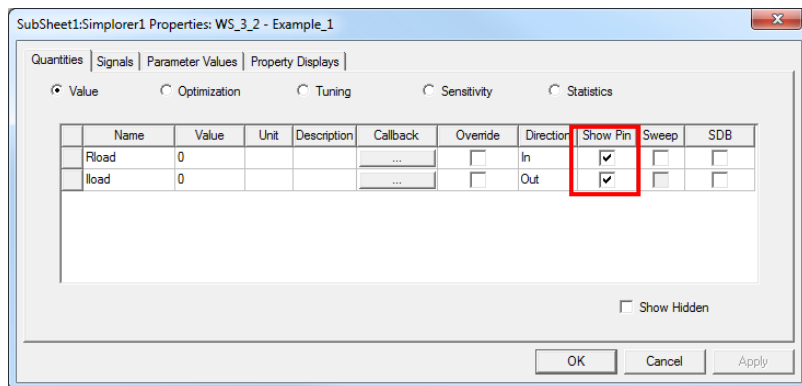
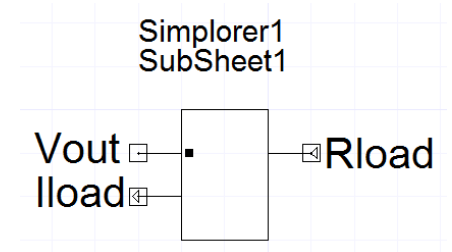
Set Component Properties

- Resistor
 - Double click on the **Resistor**
 - Change the value to **Rload**
 - Press **OK**



Set Sub-Circuit Properties

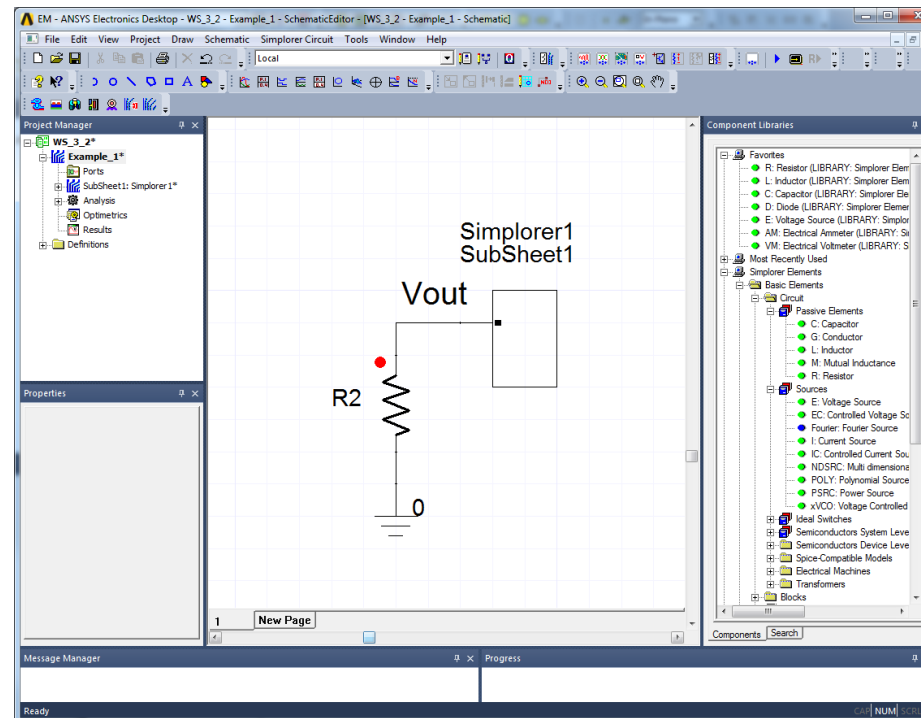
- Go back to the Top-level schematic double clicking on the Design in the Project Manager window or using the command **RMB → Pop-up**
- On the Top-level Schematic, double click and the Sub-Circuit block which has been automatically created (note the directions of Pins for **Rload** and **Iload**)
- Uncheck **Show Pin** for both **Rload** and **Iload**
- The two **Value** fields are now active (the color turned to blue)
 - Change the **Rload** value to **0.5**
 - Press **OK**



Insert Components

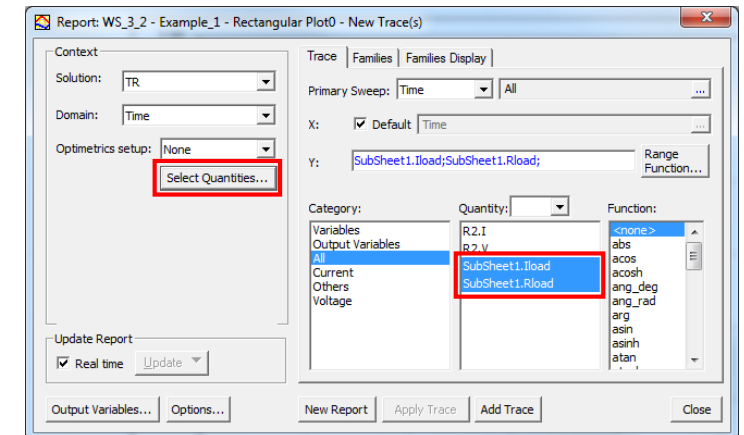
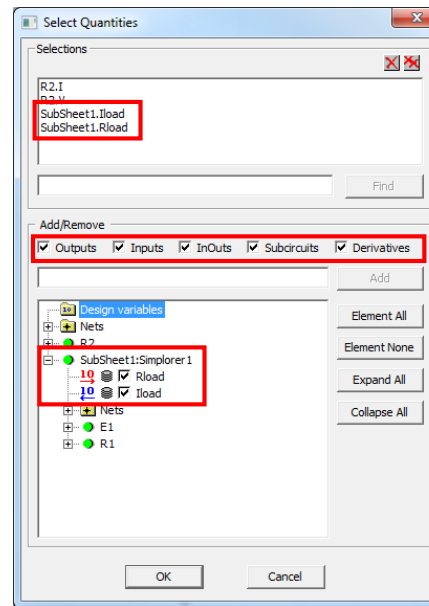
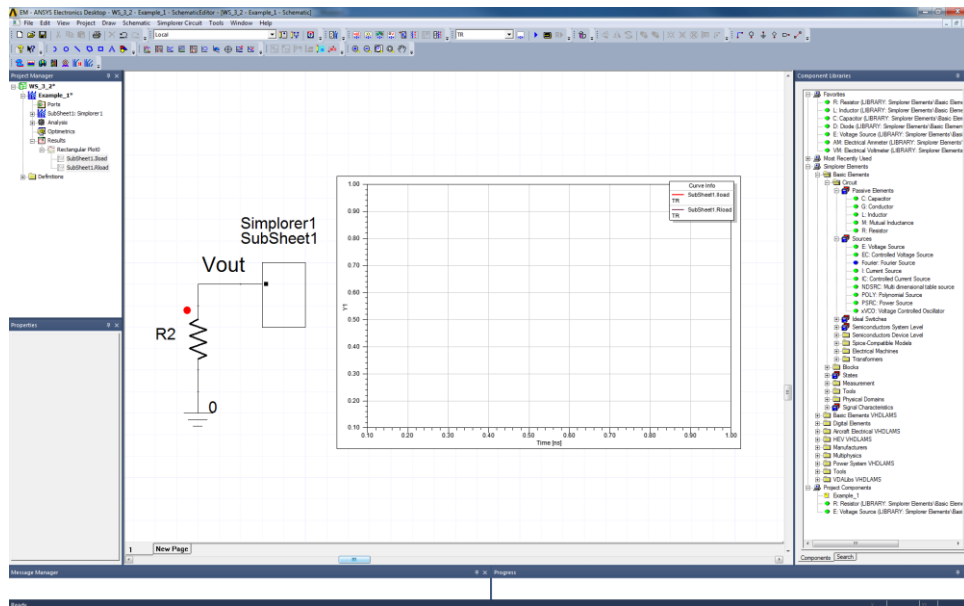
- Resistor

- In Component Libraries window *Simplorer Elements* → *Basic Elements* → *Circuit* → *Passive Elements*
- Select the **Resistor** and drag and drop it into the top level Schematic
- Press **Esc** key to exit the insert mode
- Insert the Ground and connect the components as shown in figure below



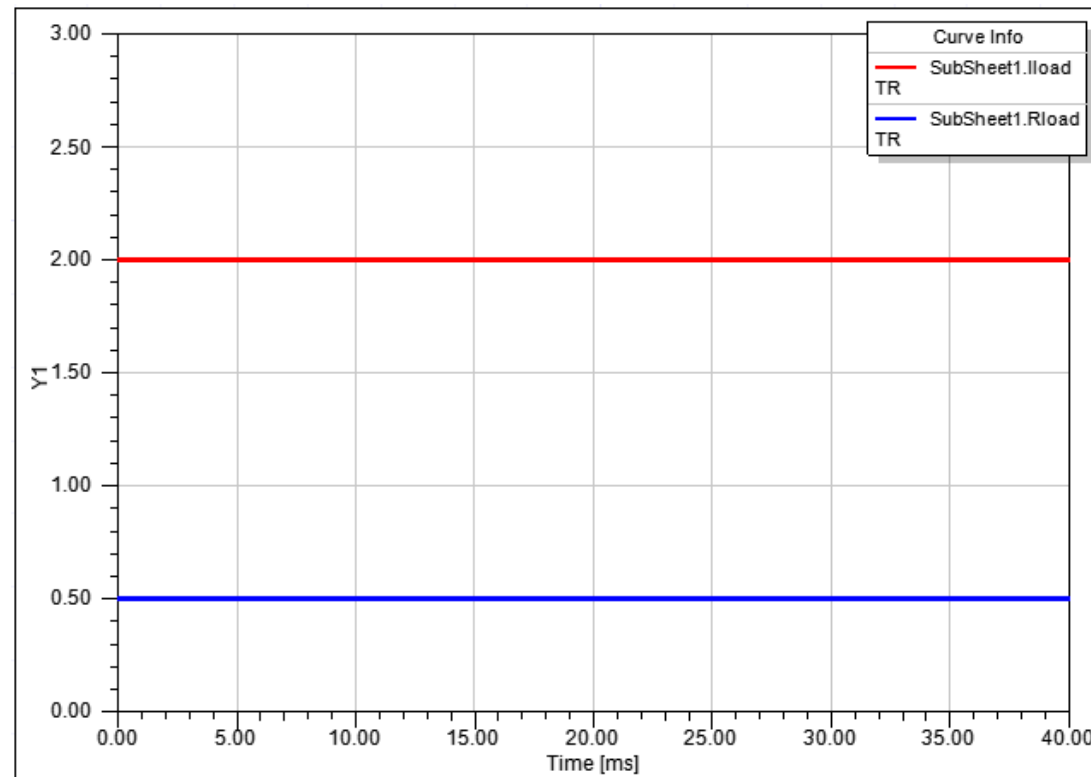
Prepare the Postprocessing

- 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. 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
- Click on **Select Quantities**. In the output dialog window, be sure all the **Add/Remove** checkboxes are selected. Check **Rload** and **Iload**. Press **OK**
- Click on the **Add Trace** button and then **Close**



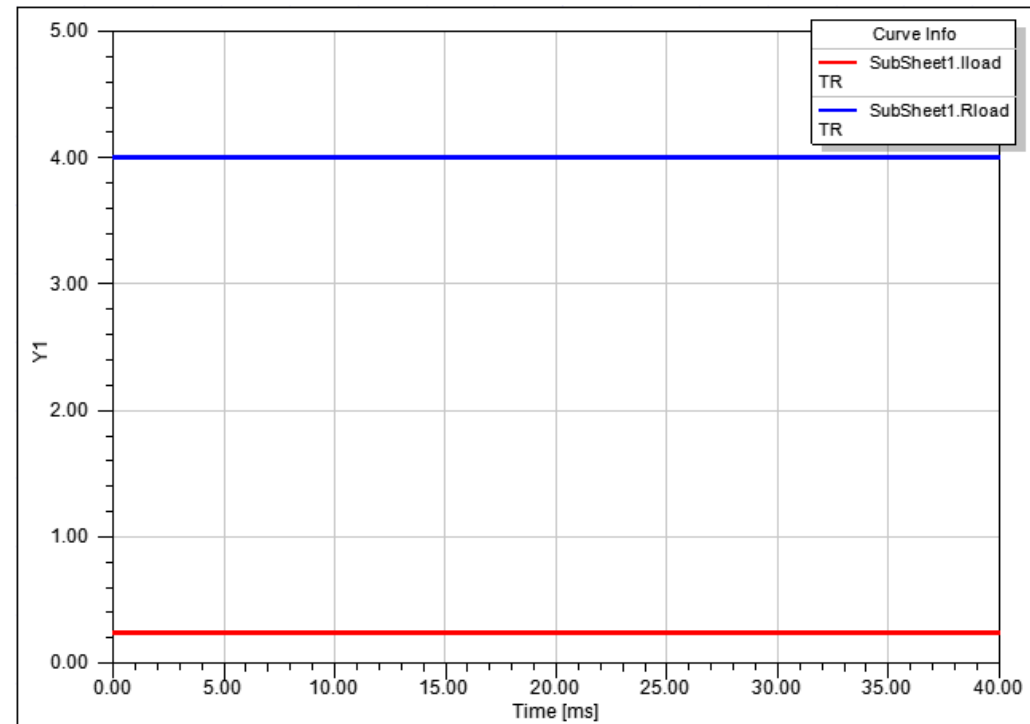
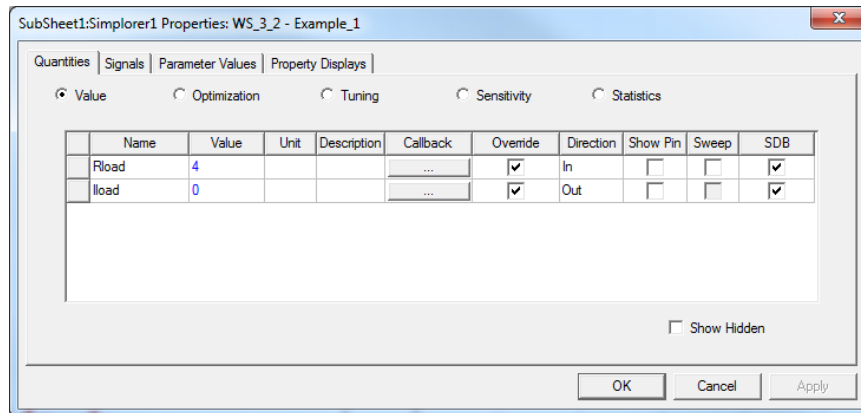
Analyze and View Results

- Select the menu item *Simplorer Circuit* → *Analyze* to run the Simulation
- Open the plot from the Project Manager window and double click on it. Select the Tab *Y Scaling*, check the *Specify Min* and *Specify Max* and set the related *Min* to 0 and *Max* to 3. Press OK
- Check the obtained results ($R_{load} = 0.5$, $I_{load} = 2$)



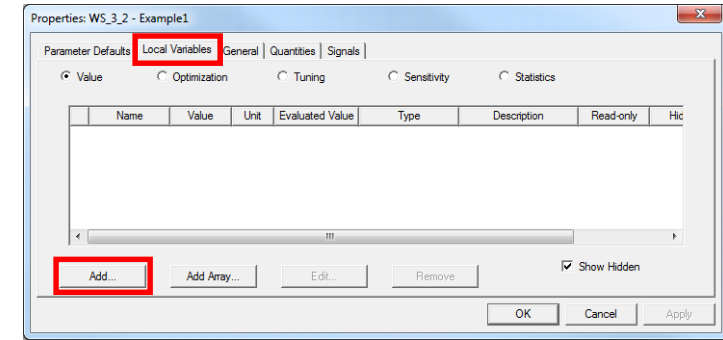
Analyze and View Results

- Double click on the sub circuit block and change Rload **Value** to **4**. Press **OK**
- Select the menu item **Simplorer Circuit** → **Analyze** to run again the Simulation.
- Change the **Y Scaling Max** value to 5
- Check the obtained results (Rload = 4, Iload = 0.25)

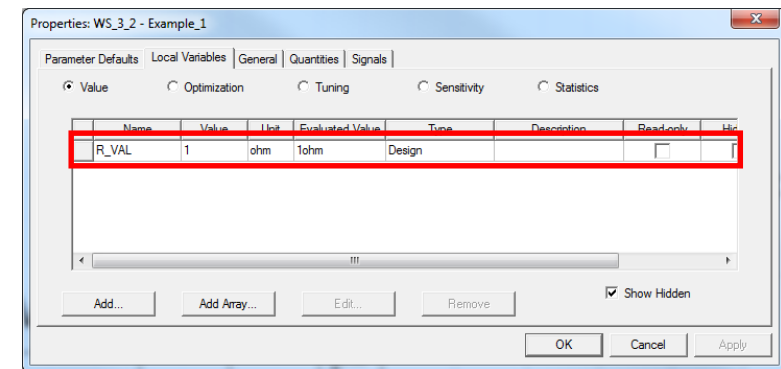
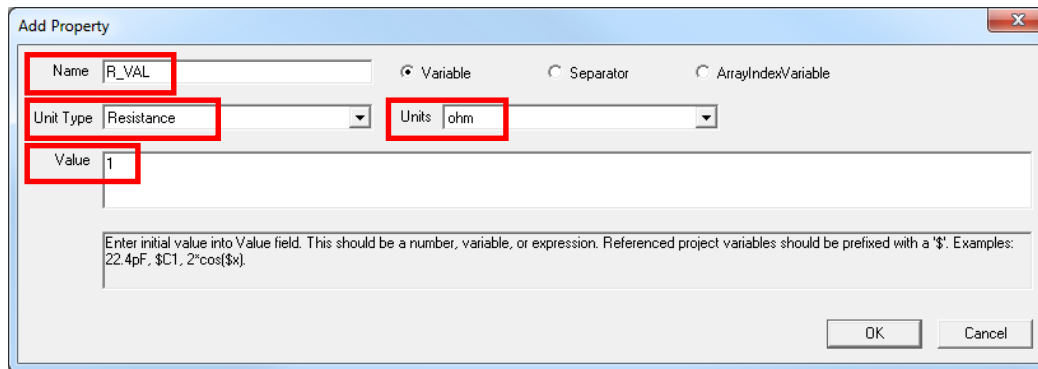


Add a Local Variable

- Select the menu item *Simplorer Circuit* → *Design Properties*
- On the Local Variables Tab, click on the **Add** button



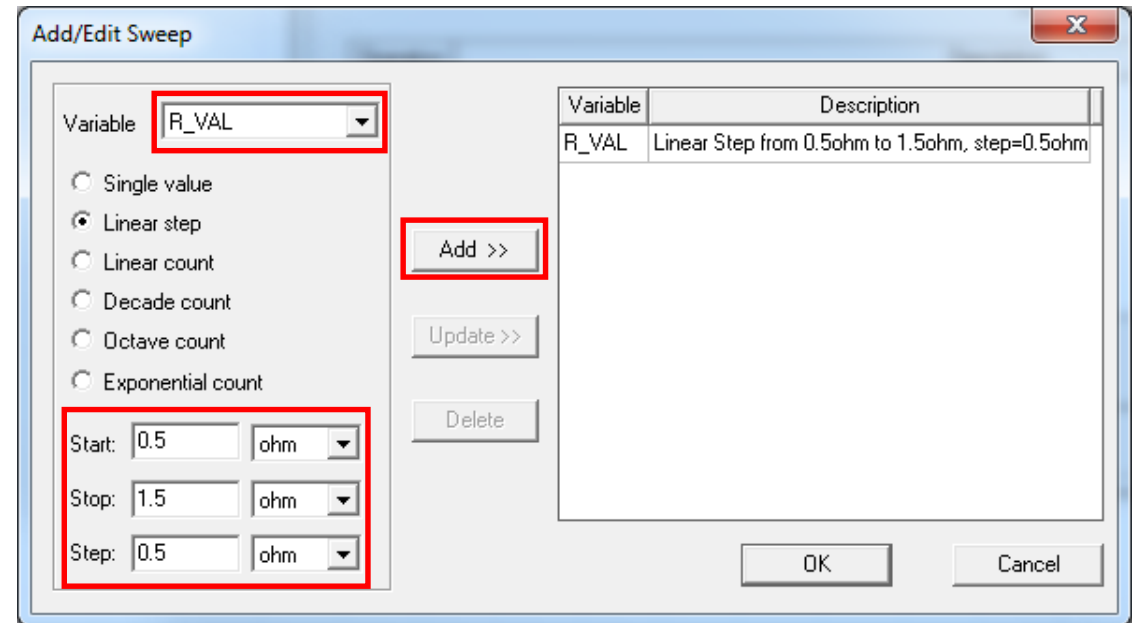
- On the **Add Variable** window, set Name to **R_VAL**, Unit Type to **Resistance**, Units to **ohm** and Value to **1** as in figure



- Press **OK** twice

Parametric Sweep Setup

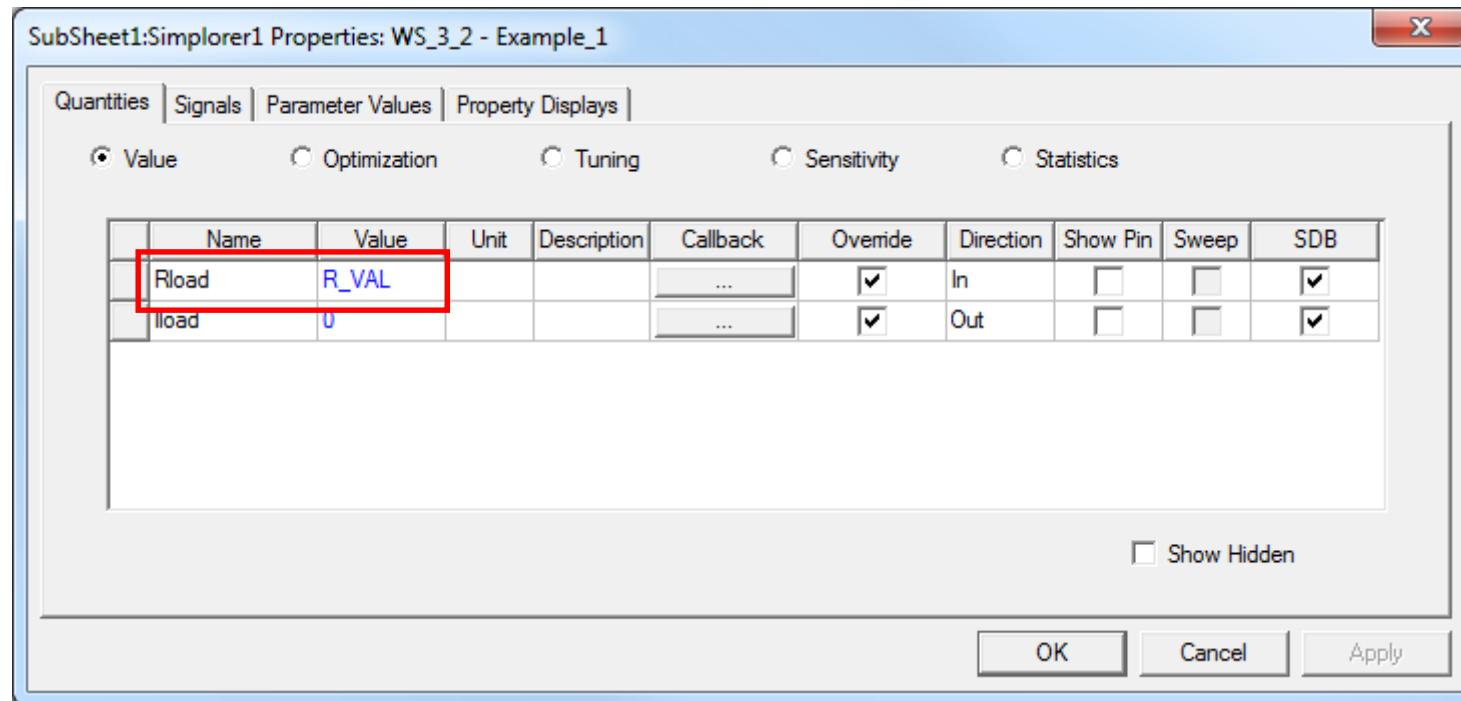
- Launch Setup Sweep Analysis window,
 - Select the menu item *Simplorer Circuit* → *Optimetrics Analysis* → *Add Parametric*
- Add Parameter Sweep for **R_VAL**
 - In Setup Sweep Analysis window, select Add
 - In Add/Edit Sweep window,
 - Variable: **R_VAL**
 - Linear Step: ☒ Checked
 - Start: **0.5 Ω**
 - Stop: **1.5 Ω**
 - Step: **0.5 Ω**
 - Select **Add>>**
 - Press **OK** twice




Note: this will add 3 sweeps for R_VAL: 0.5 Ω , 1 Ω , 1.5 Ω

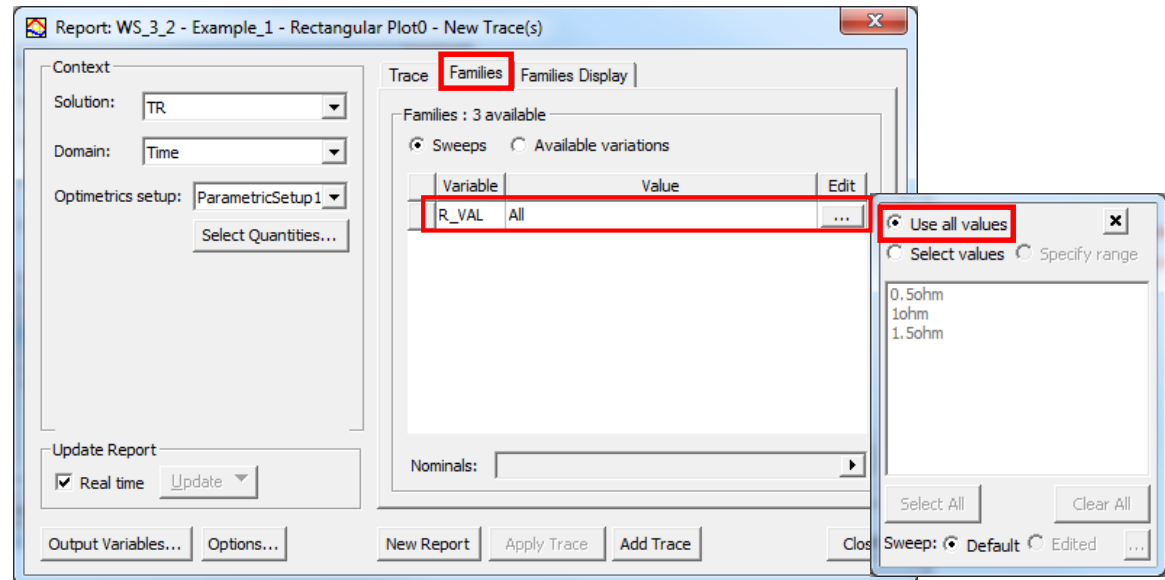
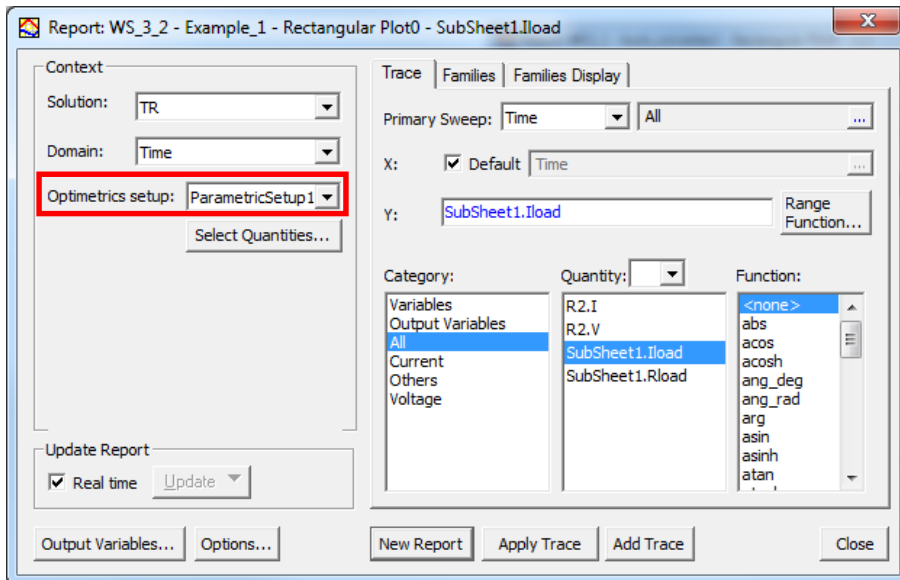
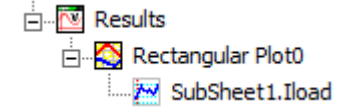
Pass the Local Variable to Sub-Circuit

- On the Top Level Schematic window double click on the Sub-Circuit block
- Change the Value for Rload to **R_VAL** in order to pass the Design Variable values to the Sub-Circuit



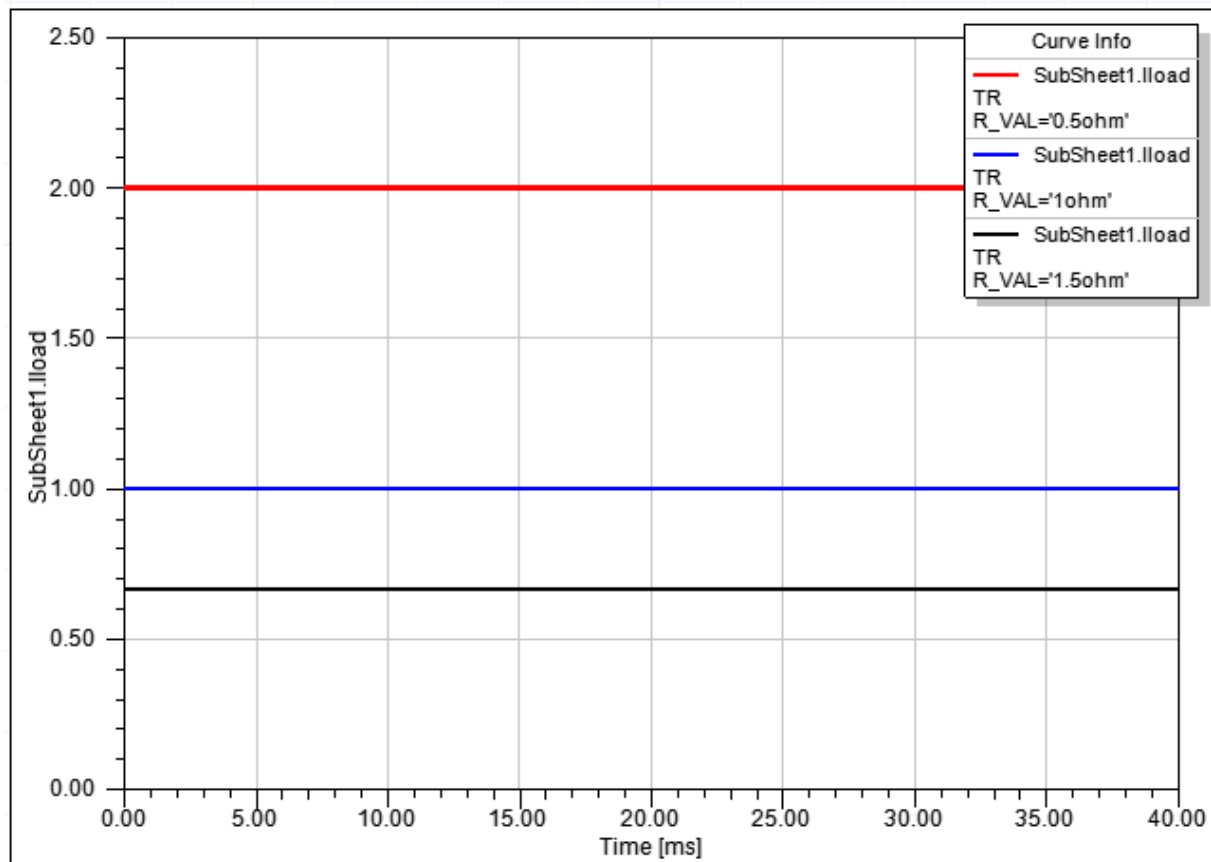
Prepare the Postprocessing

- From the Project Manager Window, delete Rload curve from the previously set plot
- Select from the Schematic Window the same plot
- *RMB on plot area → Modify Report*
- Optimetrics Setup: **ParametricSetup1**
- On the **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 three different values of load current are related to the three different values of load resistance, confirming that the R_VAL variable values have been passed to the Sub-Circuit and taken as R1 resistance values

Saving the Project

- This completes the workshop
- Save the file with the name **WS_3_2** in the working folder