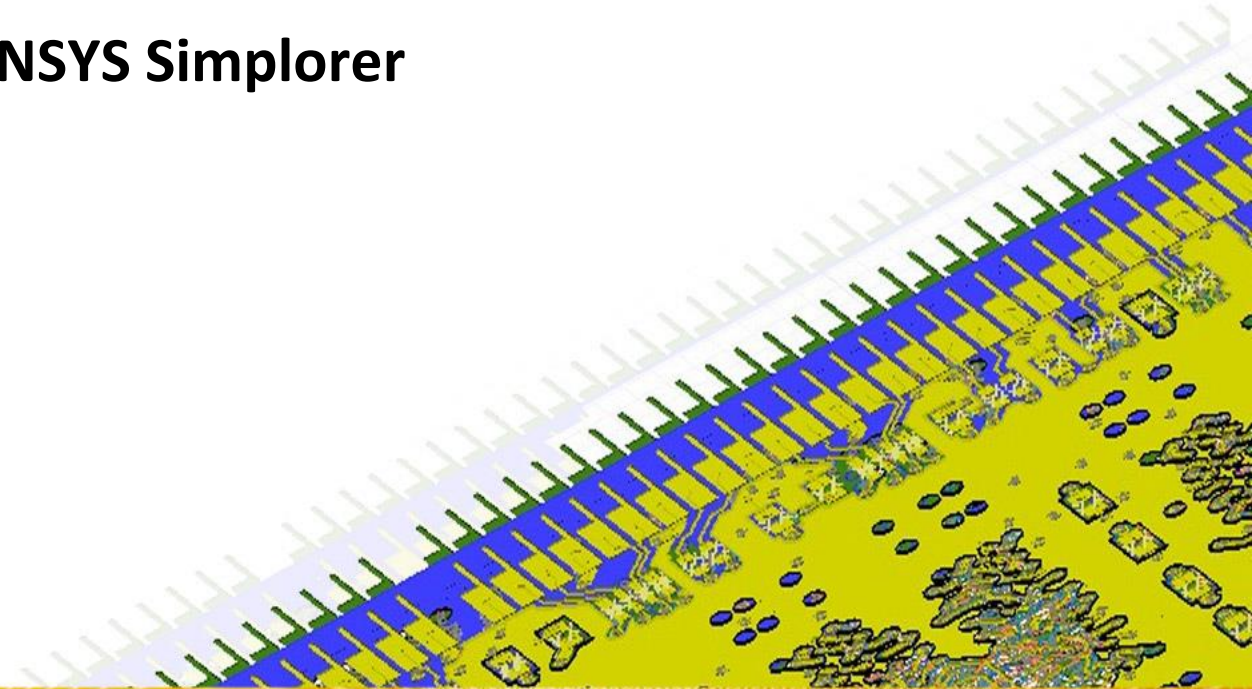




# Workshop 3.1: Transfer Function and Control Blocks

Introduction to ANSYS Simplorer



# Overview

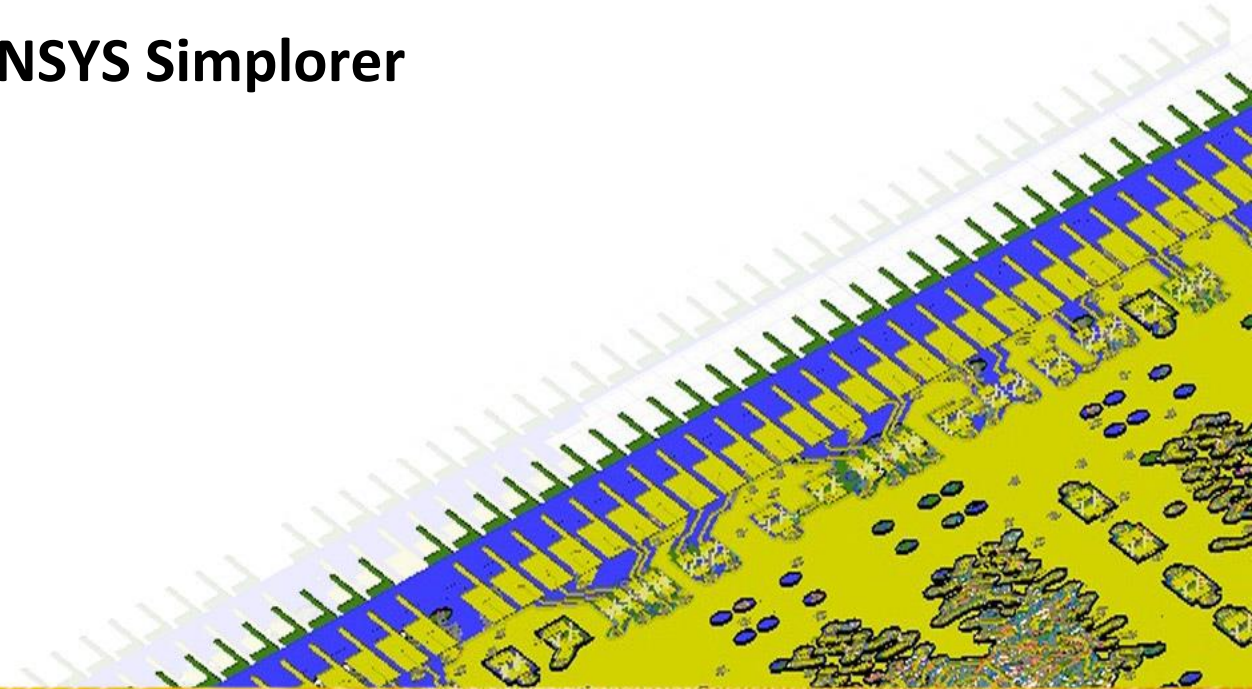
- **Transfer Function and Control Blocks**

- In this example we will investigate the behavior of a Plant represented by the block defining its transfer functions both in open and in closed loop (Feedback + Control)
- In particular we will learn
  - How to use the transfer function blocks
  - How to analyze response to typical signals (unit step)
  - How to control the system in order to comply dynamic requirements, like overshoot and steady state value through a PID controller
  - How to characterize the performances of an active filter and reproduce its behavior through a transfer function block
  - How to interface between conservative and non-conservative components





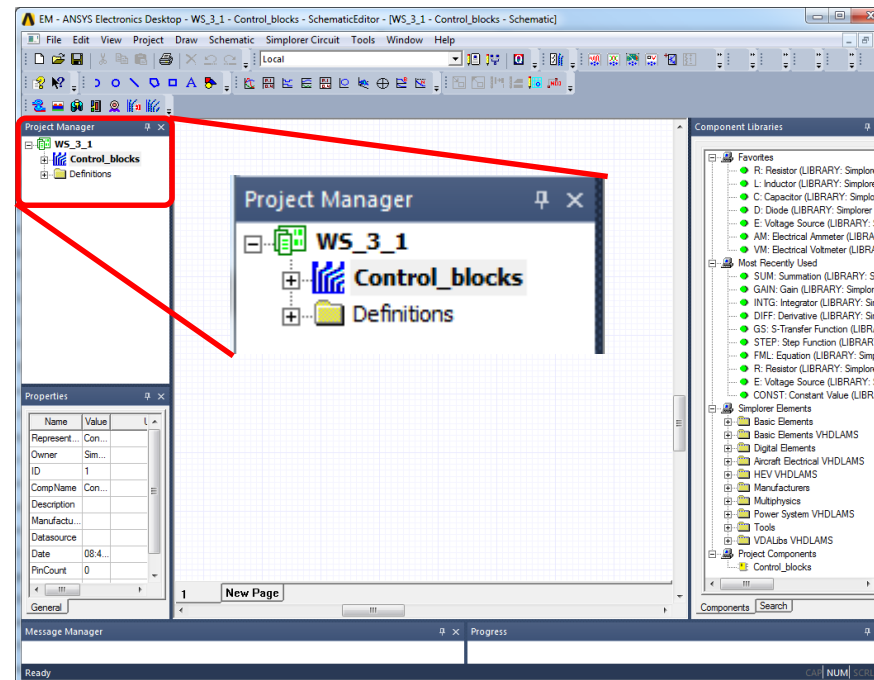
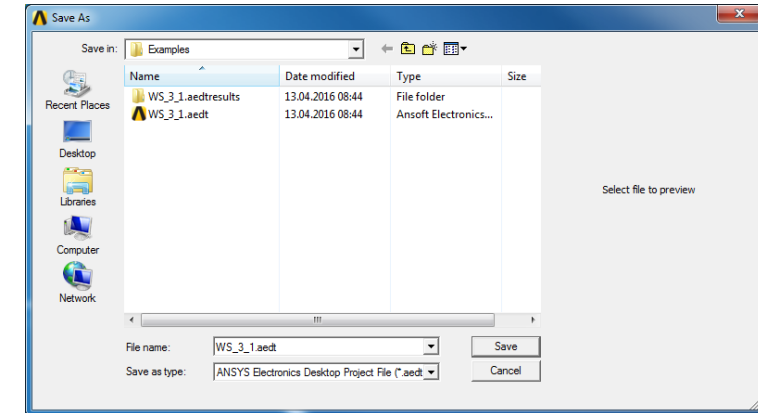
# Open Loop Plant Response

## Introduction to ANSYS Simplorer



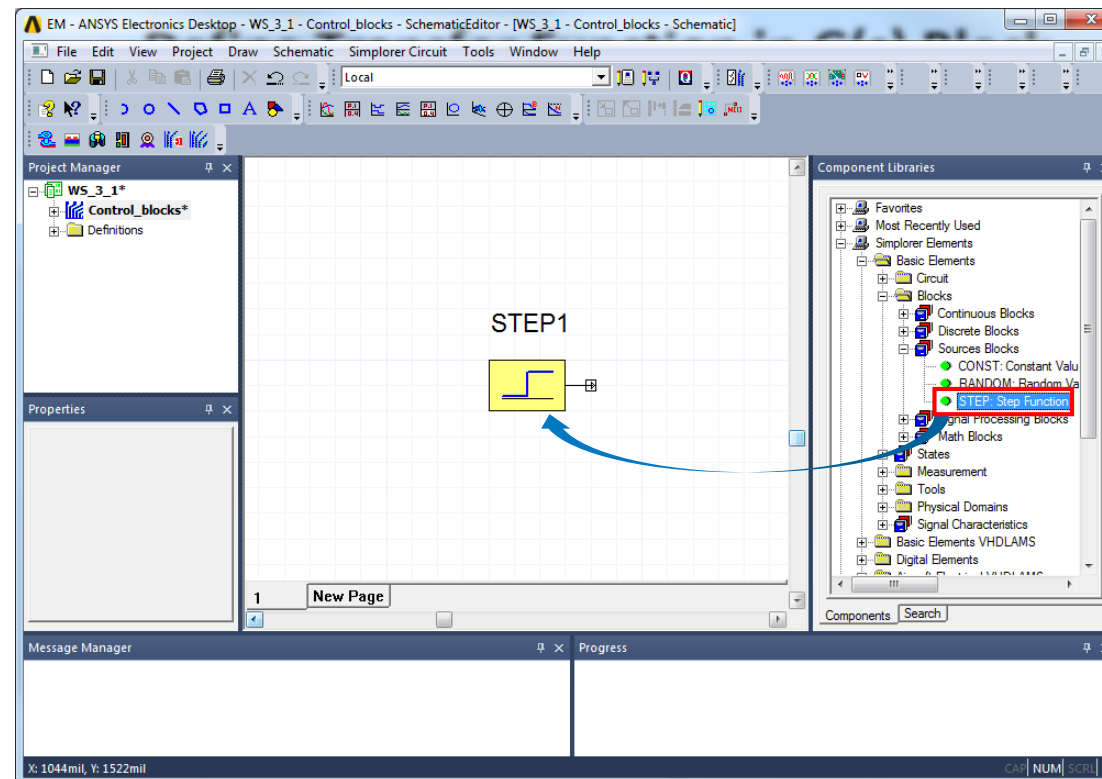
# Insert a Simplorer Design

- Launch the Electronics Desktop 2016
  - Save the Project as **WS\_3\_1.aedt**
  - Insert a Simplorer Design using the icon 
  - Rename the Design as **Control\_blocks**
  - Save again the project using the icon 



# Insert Blocks

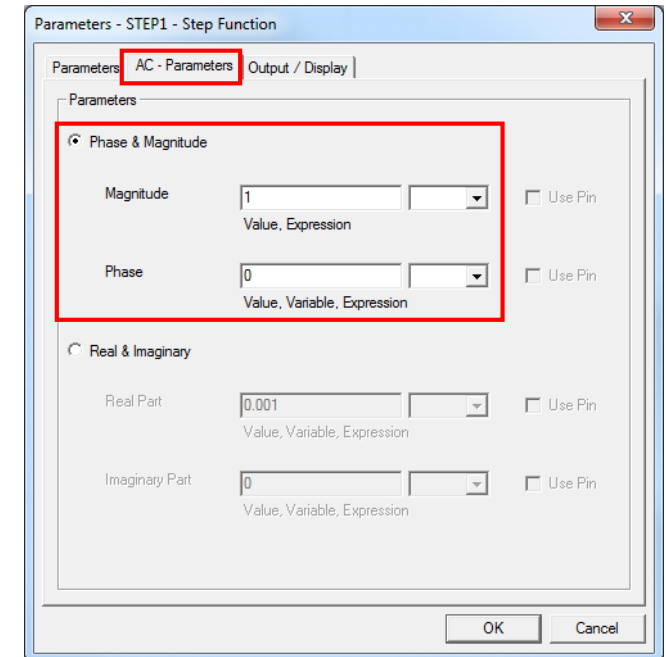
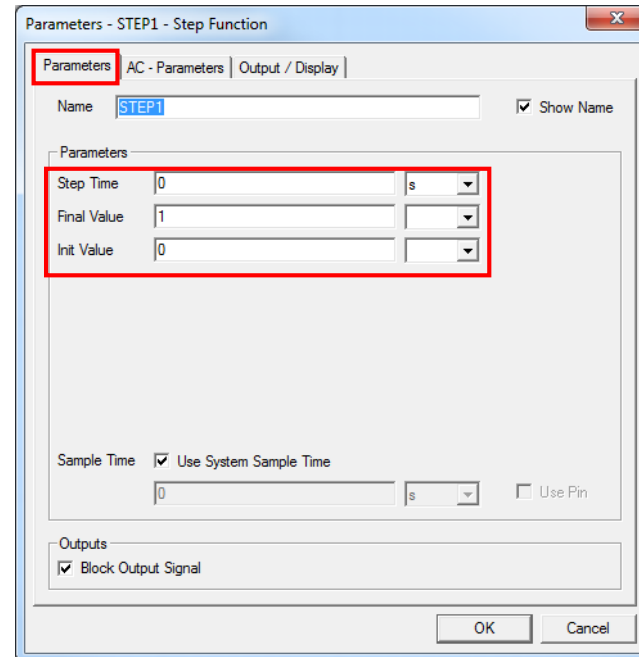
- **Step Function (Source)**
  - In Component Libraries window *Simplorer Elements* → *Basic Elements* → *Blocks* → *Sources Blocks*
  - Select the **STEP: Step Function** block, drag and drop it into the Schematic
  - Press **Esc** key to exit the insert mode





# Set Block Properties

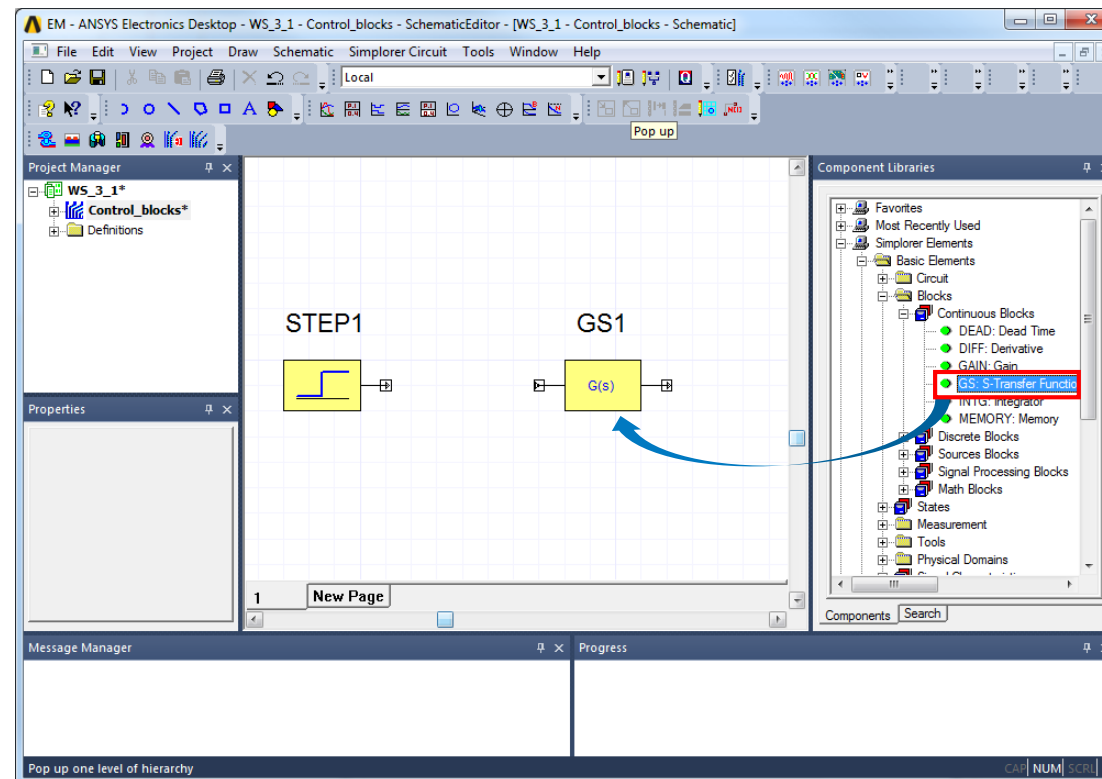
- **Step Function (Source)**
  - Double click on the **STEP** block, select the **Parameters Tab**
    - Step Time: **0 s**
    - Final Value: **1**
    - Init. Value: **0**
  - In the **AC-Parameters Tab**
    - Phase & Magnitude: ☒ Checked
    - Magnitude: **1**
    - Phase: **0**
  - Press **OK**



# Insert Blocks

- **Transfer Function**

- In Component Libraries window *Simplorer Elements* → *Basic Elements* → *Blocks* → *Continuous Blocks*
- Select the **GS: S-Transfer Function** block, drag and drop it into the Schematic
- Press **Esc** key to exit the insert mode



# Set Block Properties

- **Transfer Function**

- Double click on the **GS** block and define the transfer function for the Plant as described in Figure

- **Numerator:**

- Order: 0
- B[0]: 4.87

- **Denominator:**

- Order: 2
- A[0]: 1
- A[1]: 5
- A[2]: 6.25

- Press **OK**

Parameters - GS1 - S-Transfer Function

Parameters | Output / Display

Name: GS1 ☒ Show Name

Input Signal: 0 ☒ Use Pin

Numerator

Order: 0

| Coefficient | Value |
|-------------|-------|
| B[0]        | 4.87  |

Denominator

Order: 2

| Coefficient | Value |
|-------------|-------|
| A[0]        | 1     |
| A[1]        | 5     |
| A[2]        | 6.25  |

Sample Time: ☒ Use System Sample Time

Sample Time: 0 s ☐ Use Pin

Outputs

☒ Block Output Signal

OK Cancel

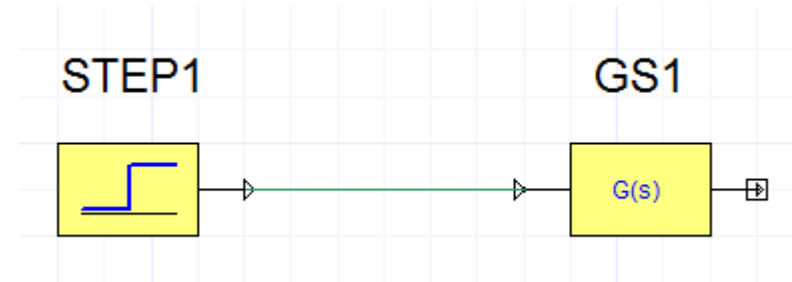
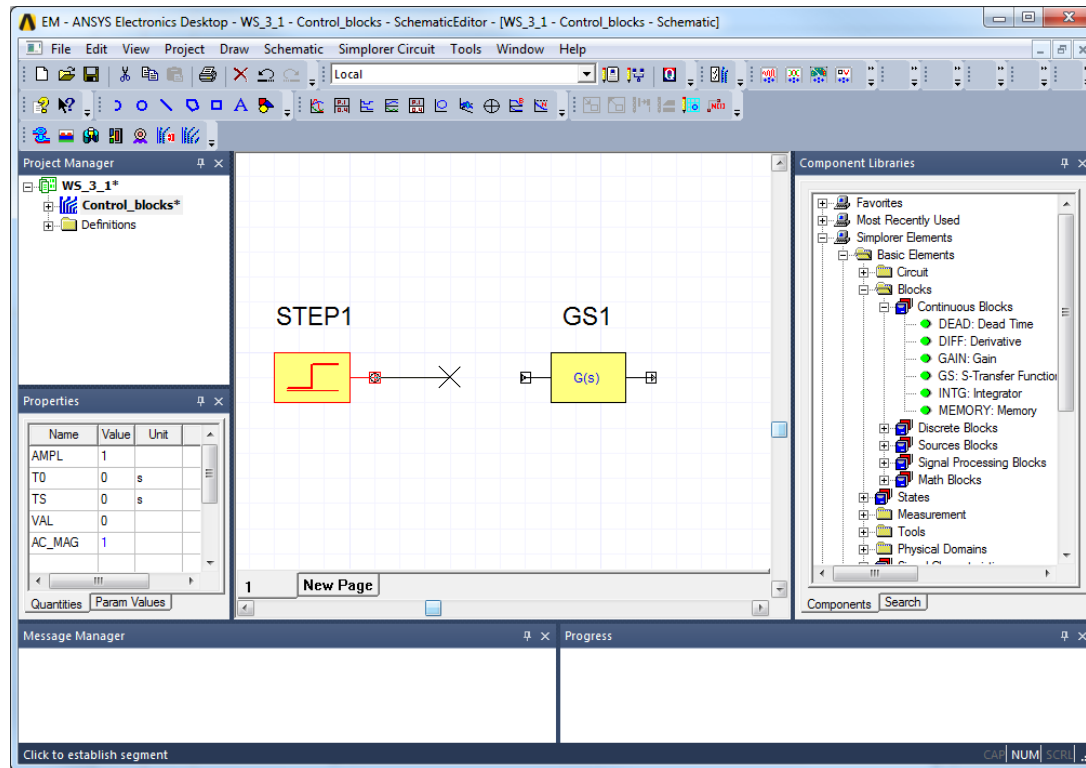


$$G(s) = \frac{4.87}{6.25s^2 + 5s + 1}$$



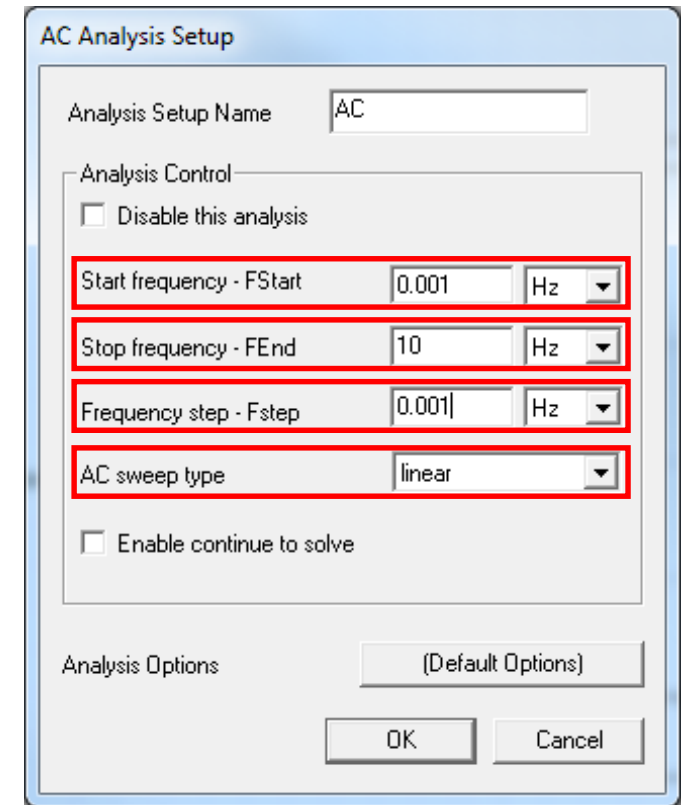
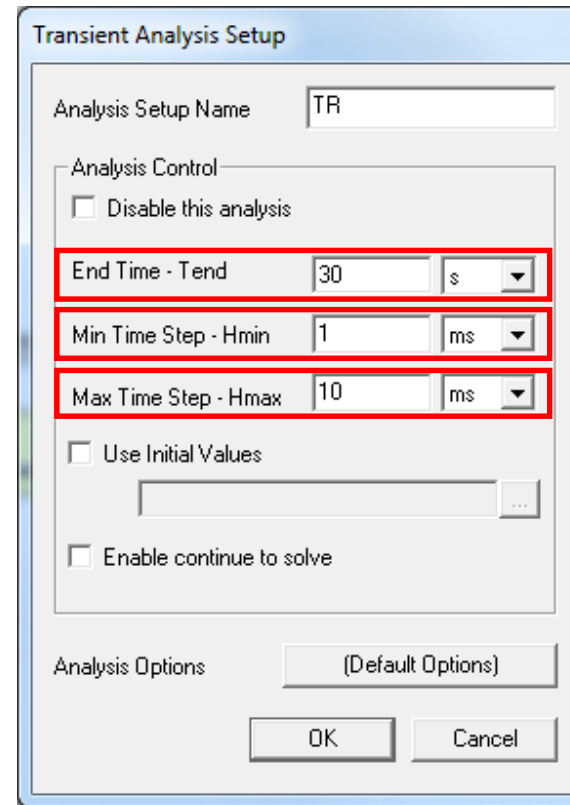
# Connect components

- Place the mouse over the output terminal of **STEP1** Block. The mouse pointer changes its shape becoming a cross. Press the **LMB** and move the cross till the input terminal of **GS1** Block



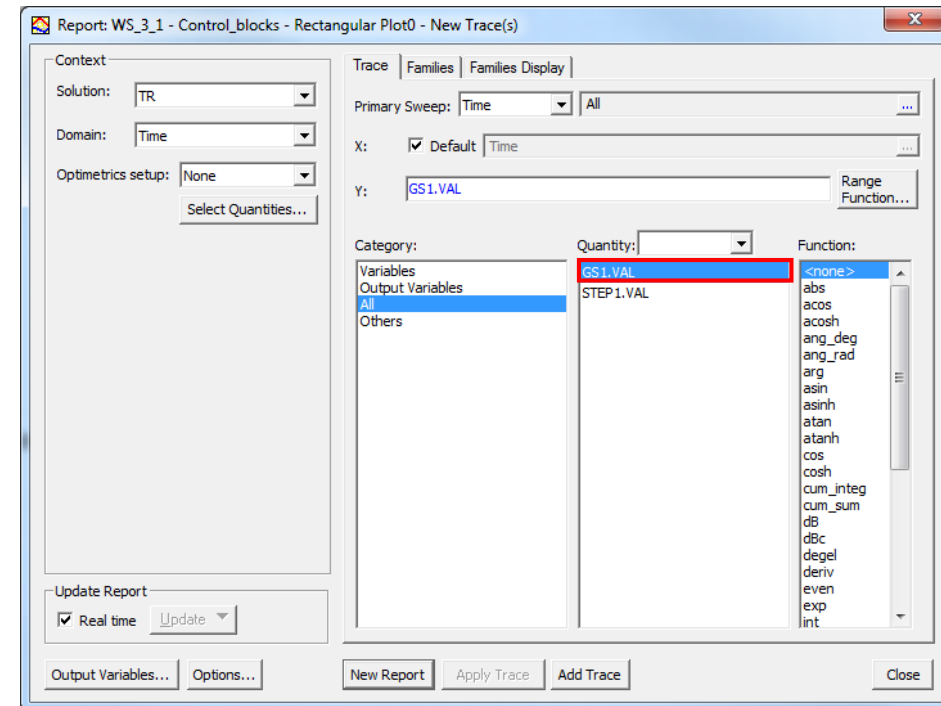
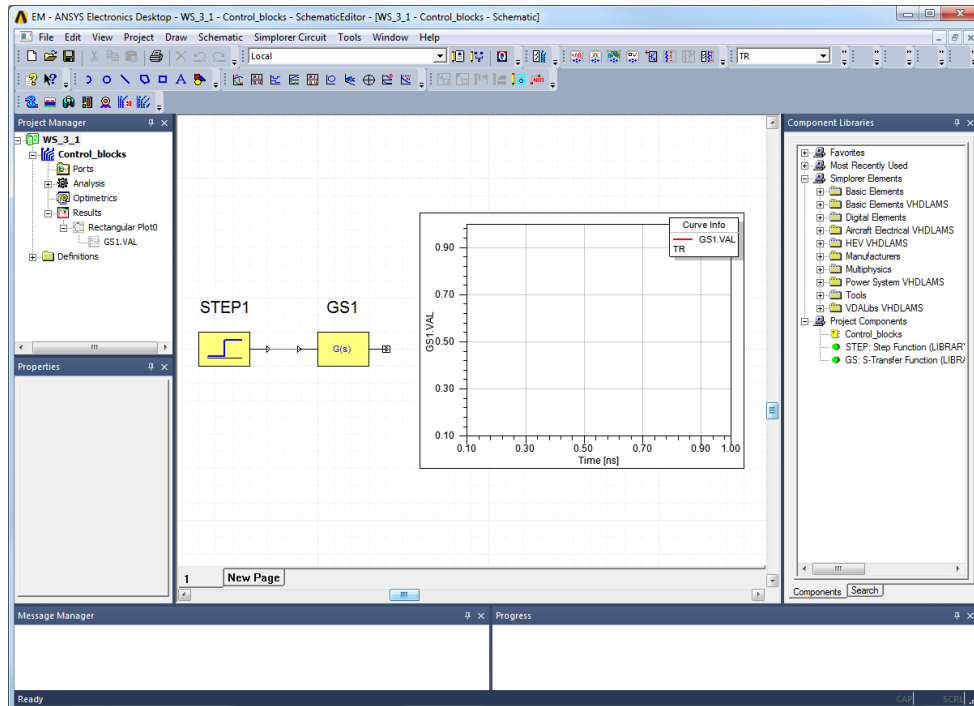
# Setup the Simulation Analysis

- We will perform a Transient analysis as well as an AC analysis, so that we can plot both Plant response in time domain and in frequency domain (Bode plots)
- In the Transient Analysis Setup window:
  - Tend: **30 s**
  - Hmin: **1 ms**
  - Hmax: **10 ms**
  - Press **OK**
- To insert an AC analysis select menu item **Simplorer Circuit → Solution setup → Add AC...**
- In the AC Analysis Setup window:
  - Start Frequency - FStart: **0.001 Hz**
  - Stop Frequency - FEnd: **10 Hz**
  - Frequency Step - FStep: **0.001 Hz**
  - AC sweep type: **linear**
  - Press **OK**



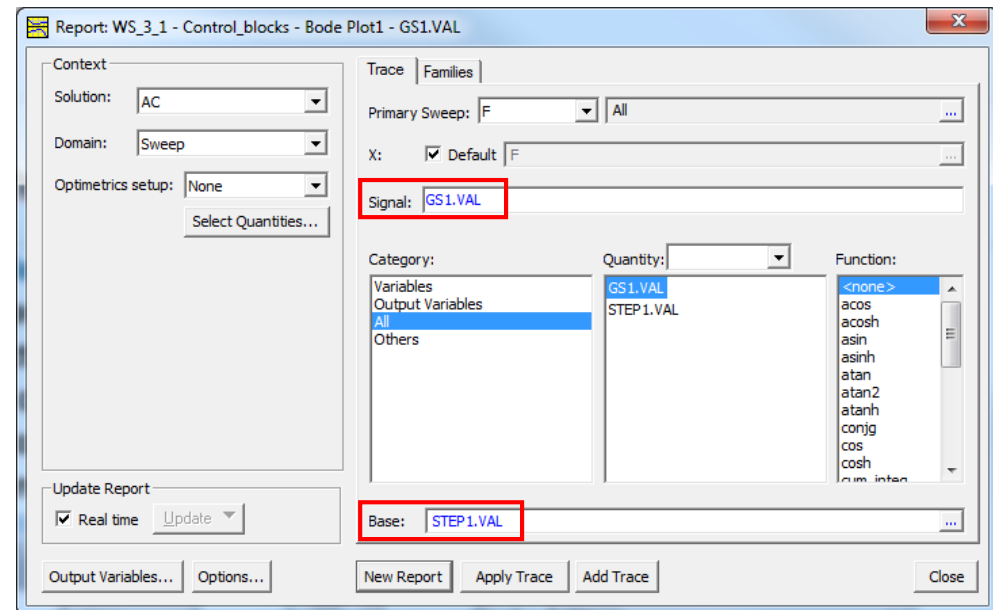
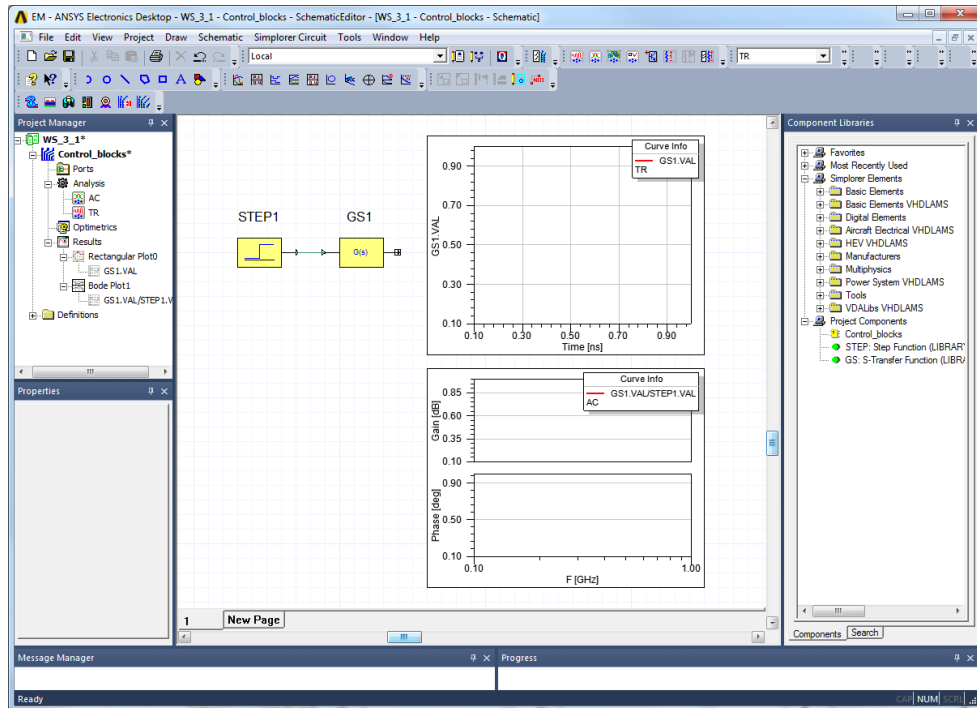
# Prepare the Postprocessing

- Select the menu item **Draw** → **Report** → **Rectangular Plot** and place the plot in the Schematic, for example on the right with respect to the Blocks
- Automatically the **New Trace** window pops-up
- Select the GS1 output signal by checking the quantity **GS1.VAL**
- Click on the **Add Trace** button and then **Close**



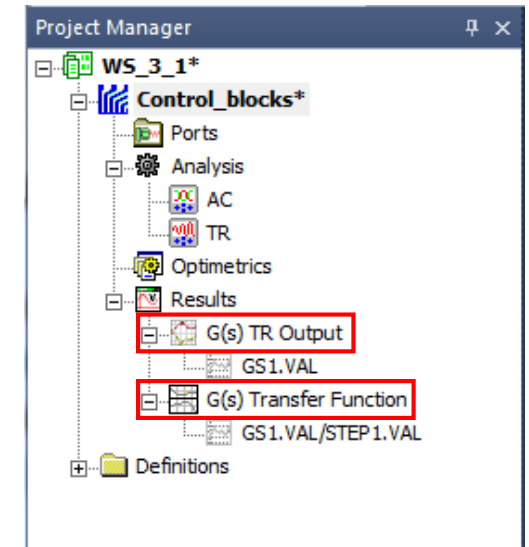
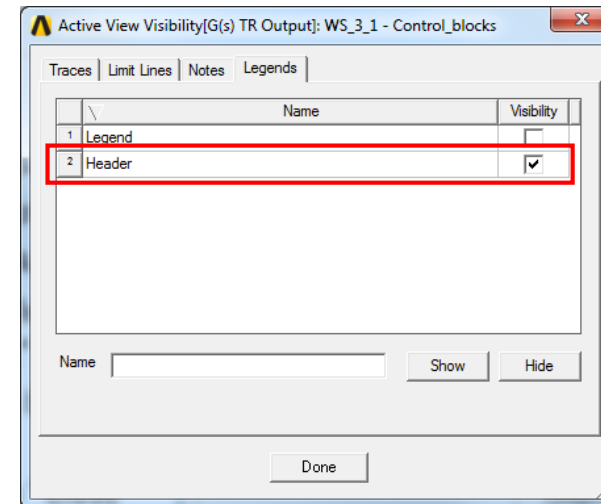
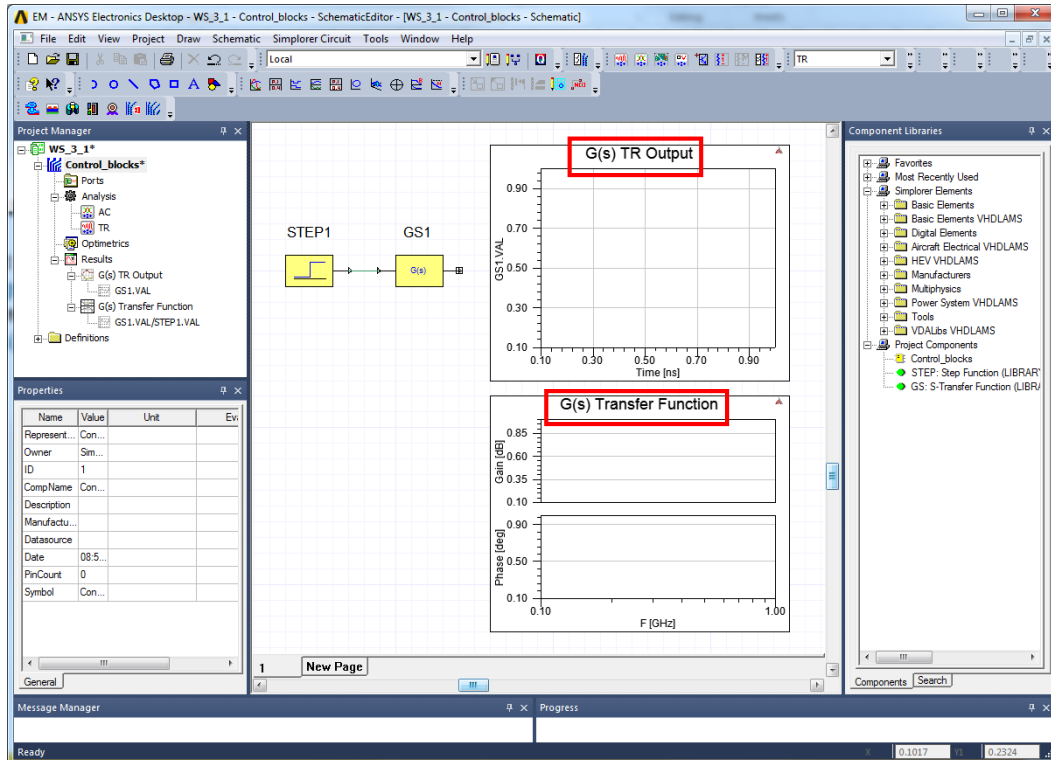
# Prepare the Postprocessing

- Select the menu item **Draw** → **Report** → **Bode Plot** and place a second plot in the Schematic, for example on the bottom of the previous plot
- In the **New Trace** window, select “**Solution**” to be **AC**
- Select the GS1 output signal by checking the quantity **GS1.VAL**
- Select the STEP1 output signal to the “**Base**” by checking the quantity **STEP1.VAL**
- Click on the **Add Trace** button and then **Close**



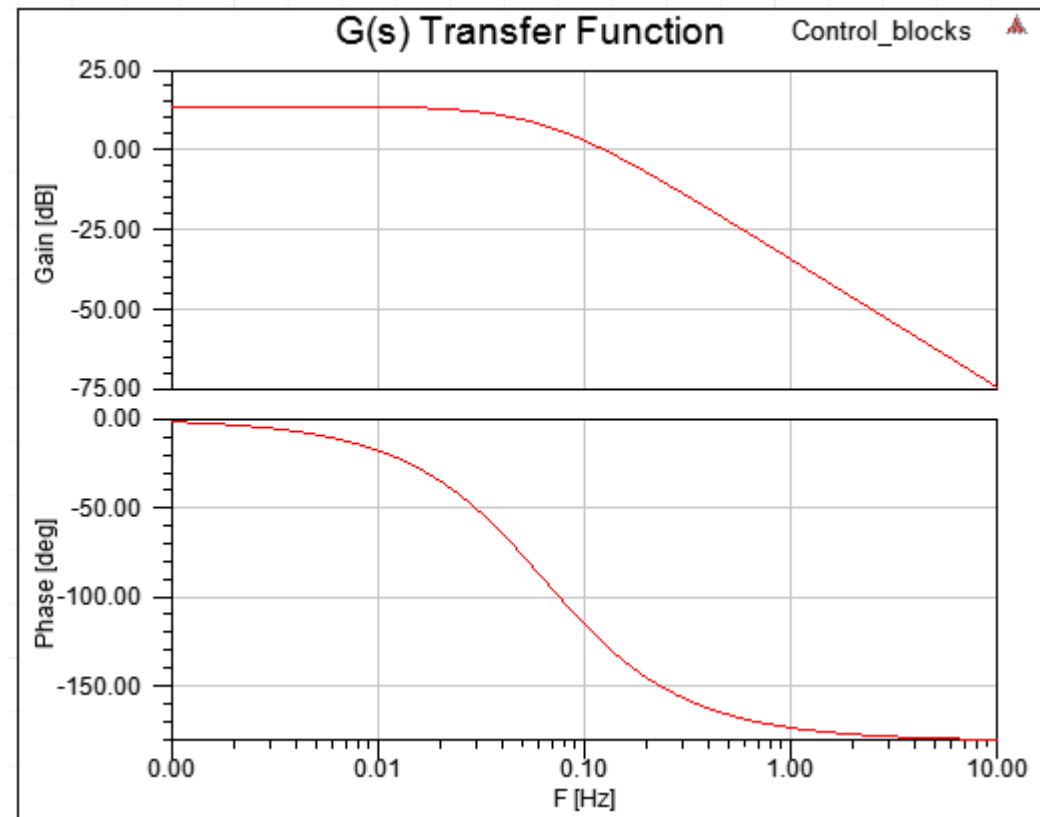
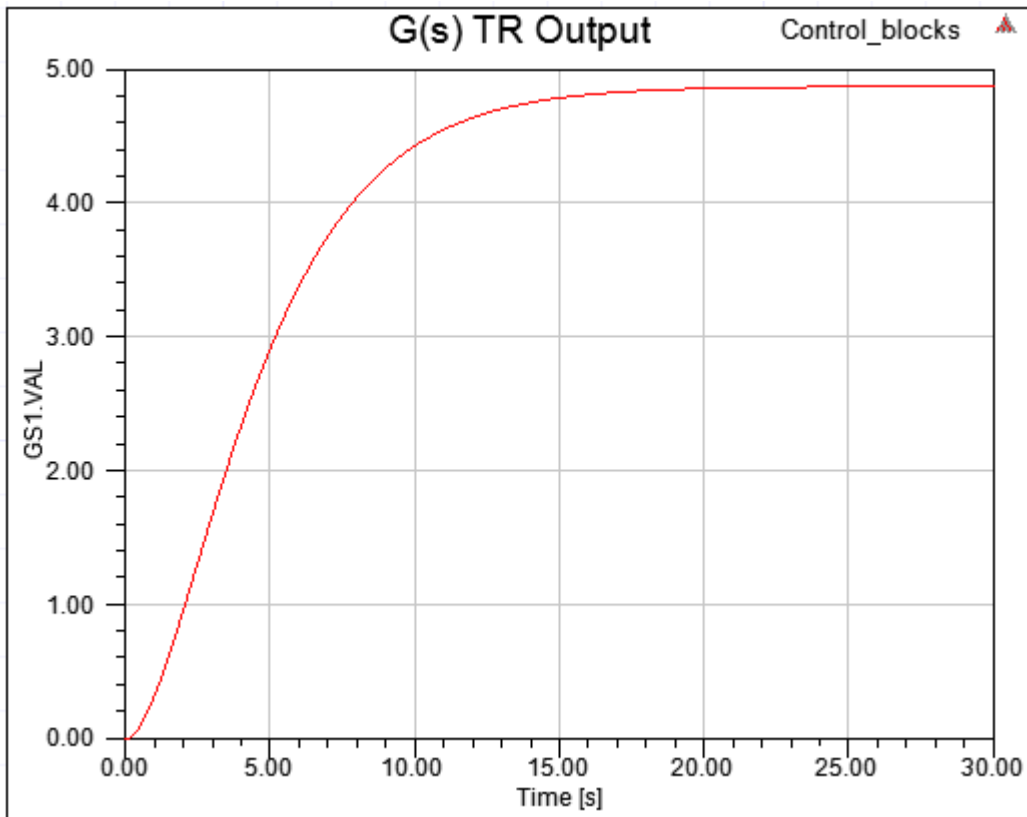
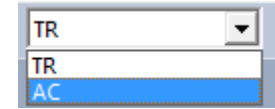
# Prepare the Postprocessing

- Set up each plot to display the header only by selecting each plot **RMB** → **View** → **Visibility**, go to the **Legends Tab**, and select only the **Header** box
- Rename each plot in the Project Manager window under the “Results” section to be **G(s) TR Output** and **G(s) Transfer Function** (for the Bode Plot)



# Analyze and View Results

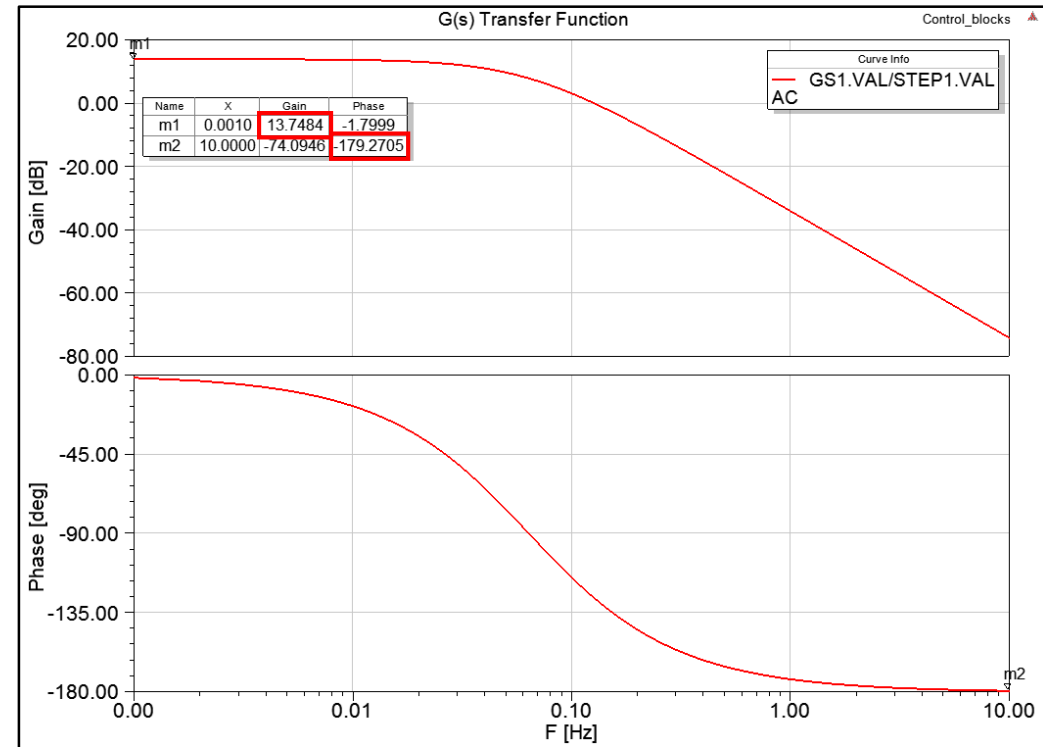
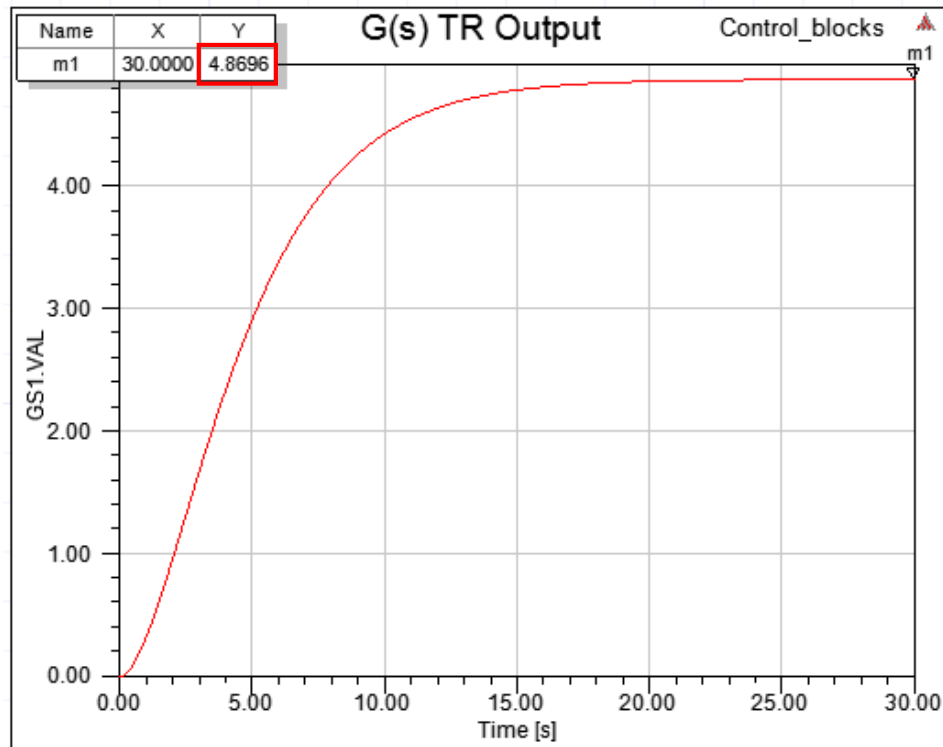
- Select the menu item *Simplorer Circuit* → *Analyze* to run the Simulation
- Change the Analysis type to *AC* from the menu and run again the simulation
- The results should appear as shown below:





# Comments to Results

- Note the Transient output ramps up reaching **4.87** as steady state final value (as shown on TR plot)
- Note the DC gain of **4.87** → **13.75dB** (as shown on the Bode Plot of magnitude)
- Note the phase tends to go to **-180 degrees** due to the double pole
- On both the plots suitable Markers have been added to help analyzing the results (see also next slide)

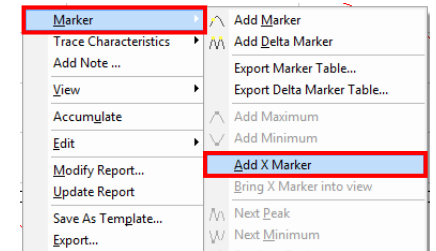
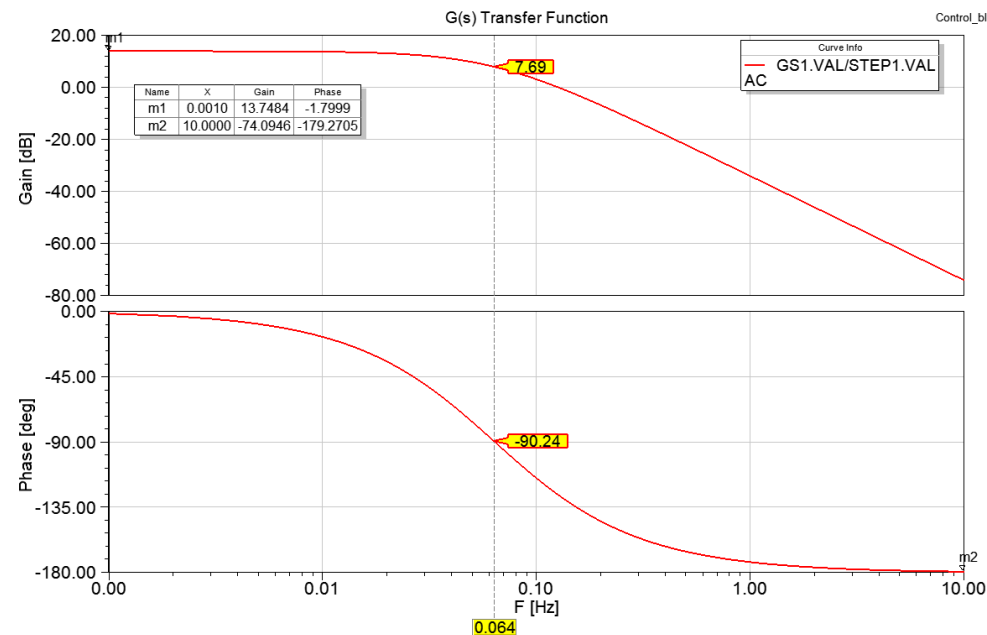


# Comments to Results

- Note the original Plant  $G(s)$  can be re-arranged and factored in order to determine the poles frequency (double pole at  $\omega = 0.4 \text{ rad/s} \rightarrow F = \omega/2\pi = 0.0637 \text{ Hz}$ )

$$G(s) = \frac{4.87}{6.25s^2 + 5s + 1} = \frac{0.7792}{s^2 + 0.8s + 0.16} = \frac{0.7792}{(s + 0.4)(s + 0.4)}$$

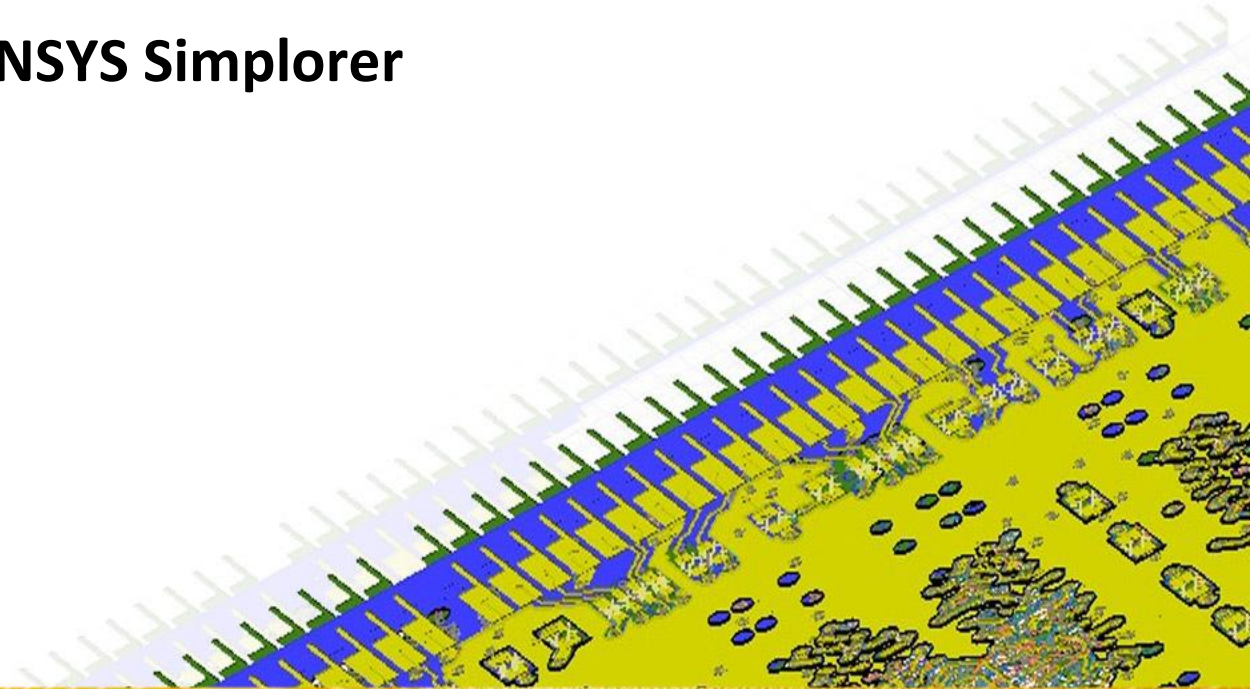
- Double click the Bode Plot from the Program Manager window under **Results**
- Move the mouse into the plot area and **RMB**  $\rightarrow$  **Marker**  $\rightarrow$  **Add X Marker**
- This will add a marker that can be moved along the **X axis** by placing the cursor over the yellow box on the X axis and moving it (note can also select it and move it with the keyboard “ $\rightarrow$ ”)
- Press **Esc** key to exit marker mode
- move the X marker to 0.064Hz (location of double pole)





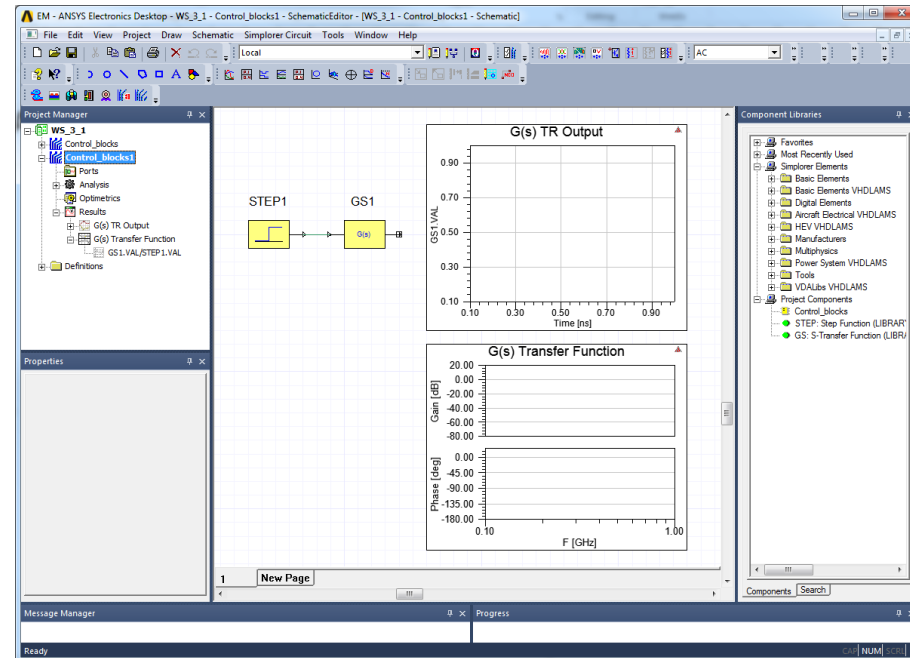
# Feedback Plant Response

## Introduction to ANSYS Simplorer



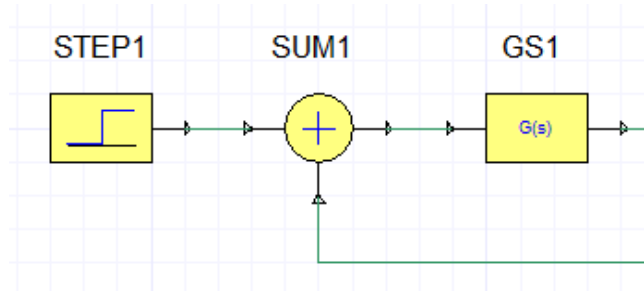
# Copy the Simplorer Design

- In the Project Manager Window select the Design **Control\_blocks**
- **Ctrl+C** to copy the Design
- Select the Project **WS\_3\_1**
- **Ctrl+V** to paste the Design
- A new Design named **Control\_blocks1** is created
- The new Design is identical to the first one but not solved
- Save the Project



# Add Feedback

- Delete the connection between STEP1 and GS1 Blocks
- In Component Libr. window *Simplorer Elements* → *Basic Elements* → *Blocks* → *Signal Processing Blocks*
- Select the **SUM: Summation** block, drag and drop it into the Schematic
- Flip the **SUM1** block Vertically
- Move the SUM1 block in-between the STEP1 and GS1 blocks and connect them as shown



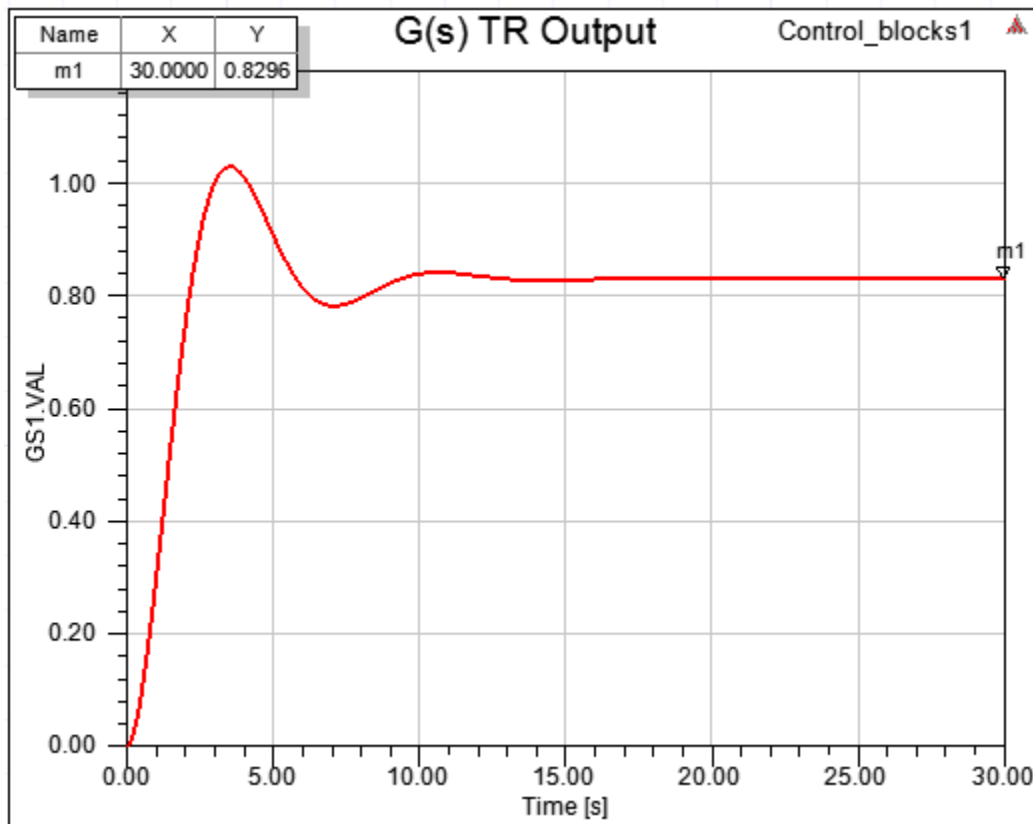
- Double click on the **SUM1** block and select the sign for the feedback from the **GS1** block to be “-” which would represent negative feedback

|  | Name     | Use Pin                             | Sign | Input Signal |
|--|----------|-------------------------------------|------|--------------|
|  | INPUT[0] | <input checked="" type="checkbox"/> | +    | STEP1.VAL    |
|  | INPUT[1] | <input checked="" type="checkbox"/> | -    | GS1.VAL      |

- Note this configuration now represents a closed loop system without compensation

# Analyze and View Results

- Be sure that the Transient Analysis (TR) is selected
- Select the menu item *Simplorer Circuit* → *Analyze* to run the Simulation
- The results should appear as shown below:



- Note that now the steady state value is less than 1, since the overall transfer function in the present case is:

$$\frac{G(s)}{1 + G(s)} = \frac{4.87}{6.25s^2 + 5s + 5.87}$$

- Calculating that function for  $s=0$  it gives the final value:

$$\left. \frac{G(s)}{1 + G(s)} \right|_{s=0} = \frac{4.87}{5.87} = 0.8296$$

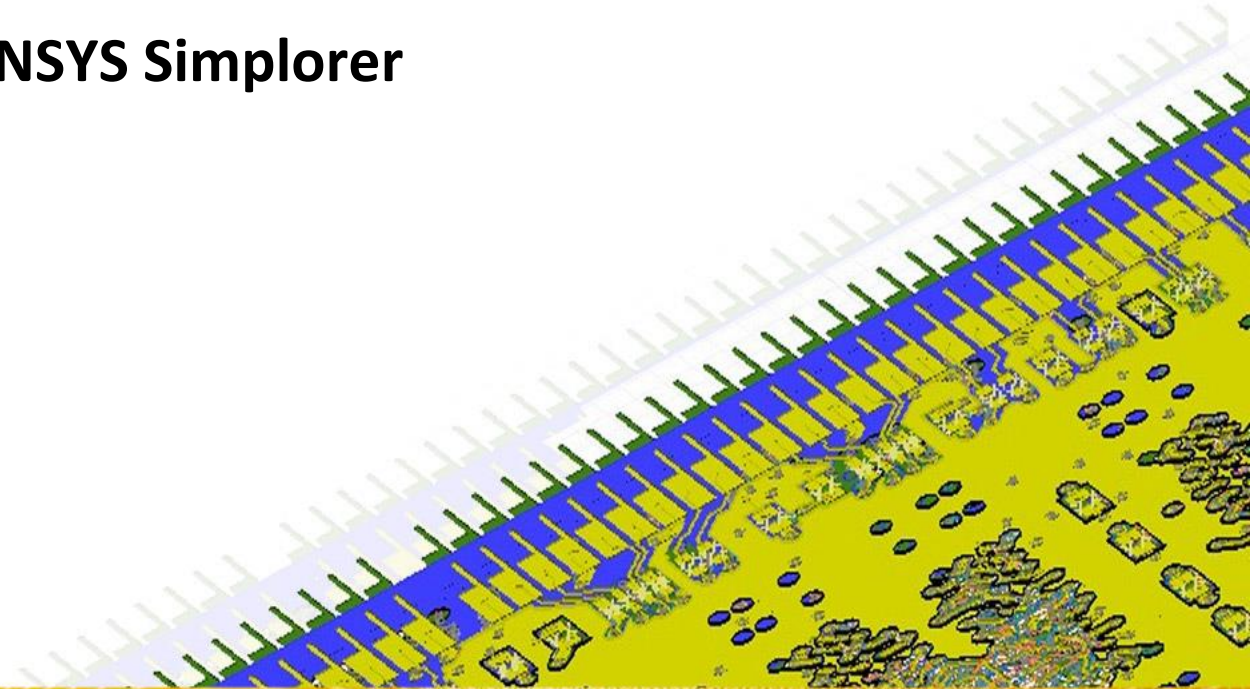
- The marker on the Plot confirms the expected results
- Save the Project





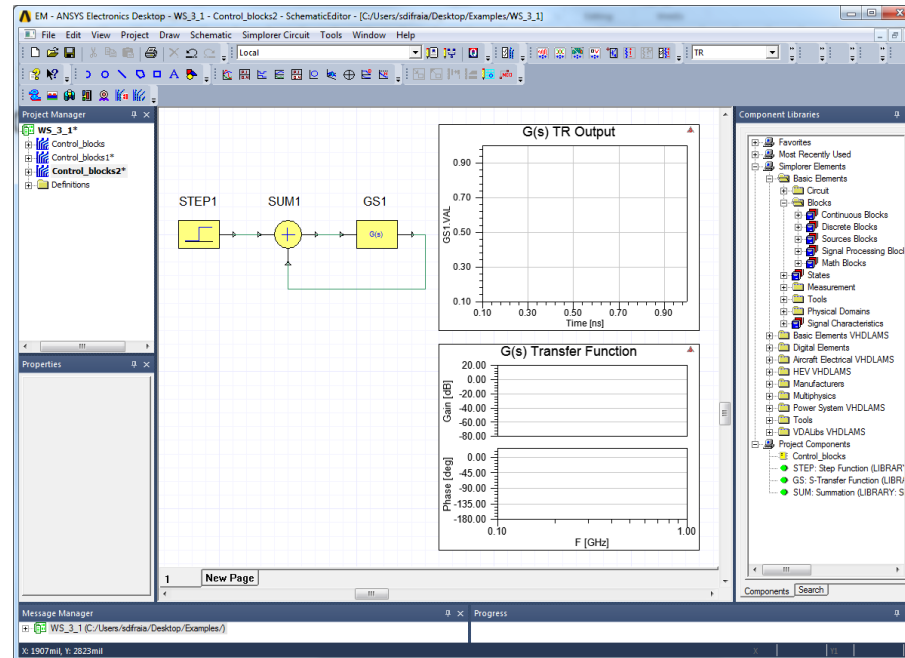
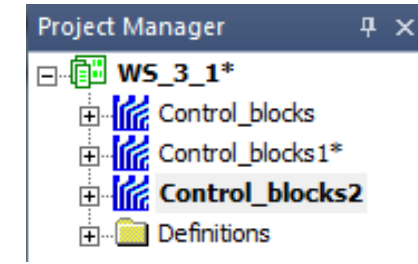
# Feedback Plant Response with PID Controller

Introduction to ANSYS Simplorer



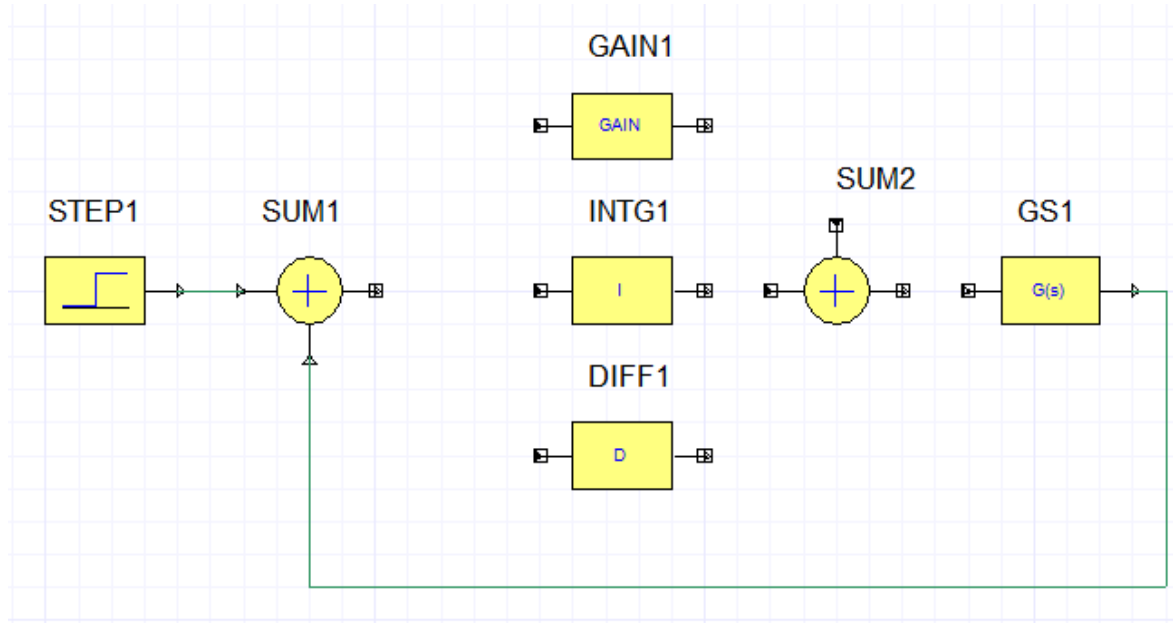
# Copy the Simplorer Design

- In the Project Manager Window select the Design **Control\_blocks1**
- **Ctrl+C** to copy the Design
- Select the Project **WS\_3\_1**
- **Ctrl+V** to paste the Design
- A new Design named **Control\_blocks2** is created
- The new Design is identical to the previous one but not solved



# Insert Blocks

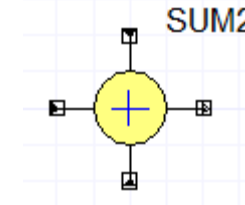
- Delete the connection between SUM1 and GS1 Blocks
- In Component Lib. window *Simplorer Elements* → *Basic Elements* → *Blocks* → *Signal Processing Blocks*
- Select the **SUM: Summation** block, drag and drop it into the Schematic
- In Component Lib. window *Simplorer Elements* → *Basic Elements* → *Blocks* → *Continuous Blocks*
- Select the **GAIN: Gain** block, drag and drop it into the Schematic
- Select the **INTG: Integrator** block, drag and drop it into the Schematic
- Select the **DIFF: Derivative** block, drag and drop it into the Schematic
- Arrange the blocks as shown



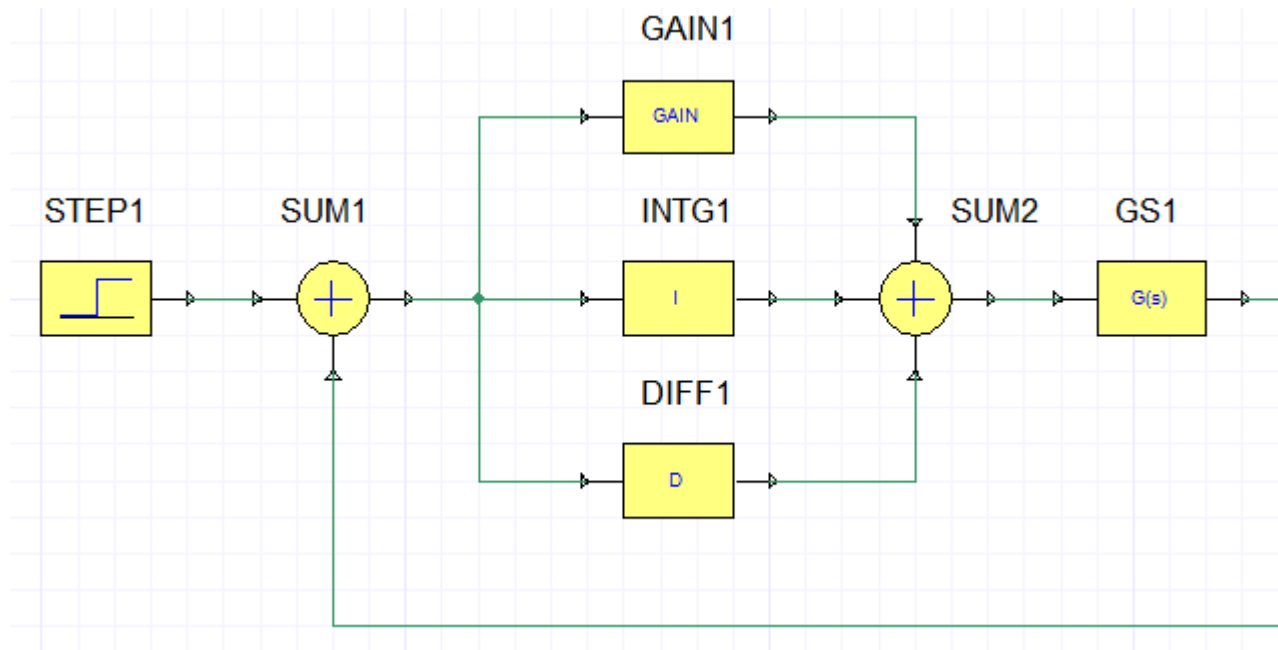
# Insert Blocks

- Double click on the **SUM2** block and add input pin by selecting another box for the input

|  | Name     | Use Pin                             | Sign | Input Signal |
|--|----------|-------------------------------------|------|--------------|
|  | INPUT[0] | <input checked="" type="checkbox"/> | +    | _Empty       |
|  | INPUT[1] | <input checked="" type="checkbox"/> | +    | _Empty       |
|  | INPUT[2] | <input checked="" type="checkbox"/> | +    | _Empty       |

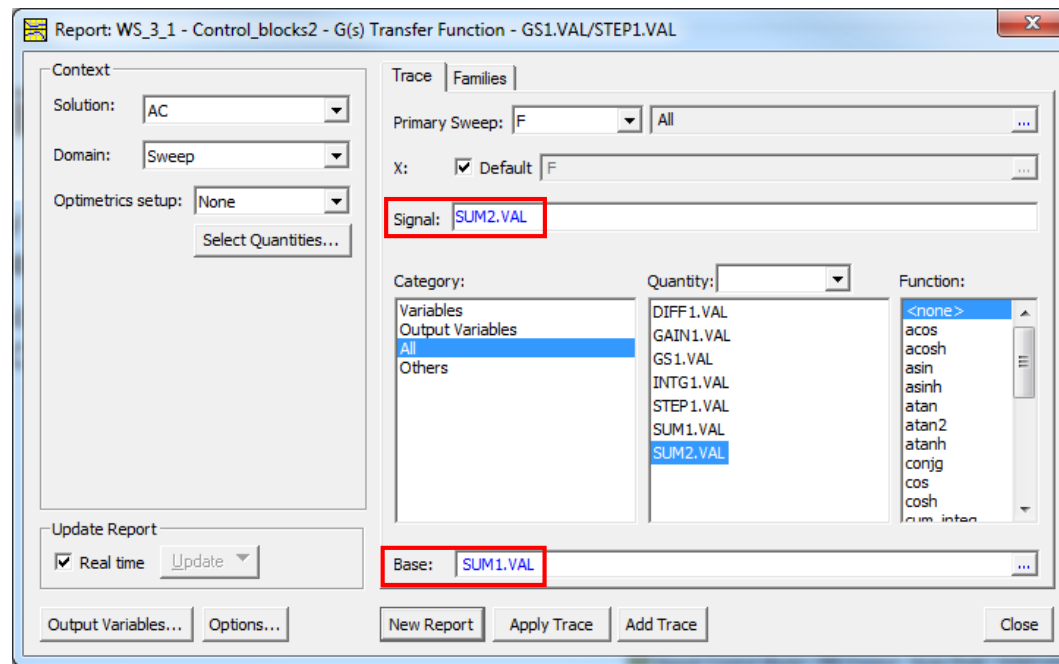


- Connect the blocks as shown below



# Prepare the Postprocessing

- Select the **Bode Plot** and **RMB** → **Modify Report**
- Select the SUM2 output signal by checking the quantity **SUM2.VAL**
- Select the SUM1 output signal to the “**Base**” by checking the quantity **SUM1.VAL**
- Click on the **Apply Trace** button and then **Close**
- The Bode Plot will display the transfer function of the **PID controller section**

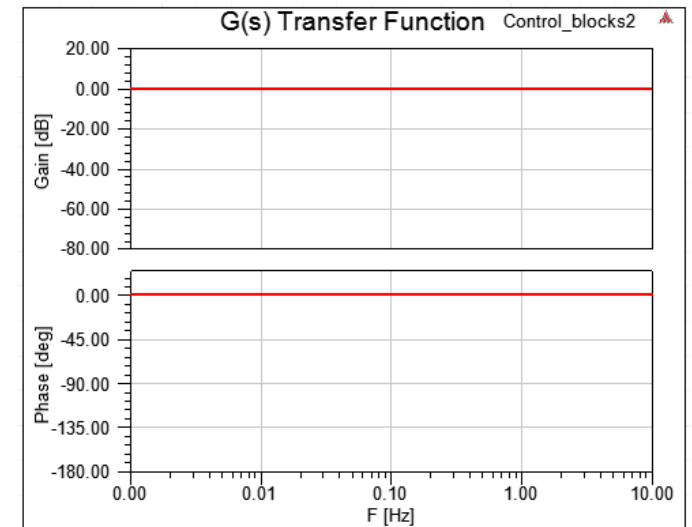
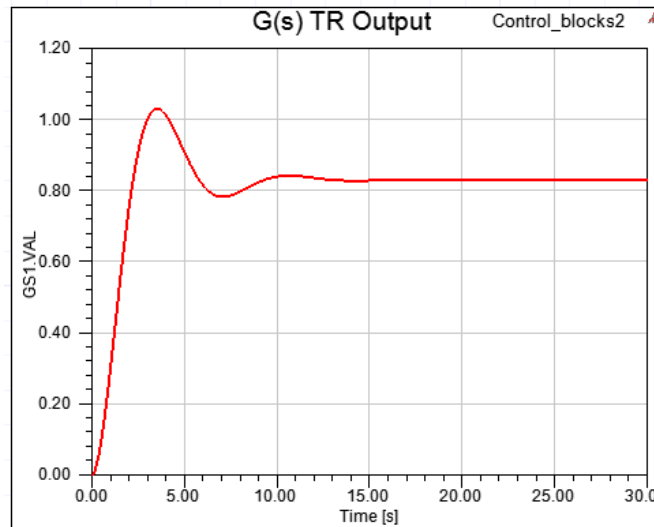


# Set PID Controller Coefficients

- Calling  $K_p$ ,  $K_i$  and  $K_d$  the three gain coefficients associated with **Gain**, **Integrator** and **Derivative** blocks respectively, the Transfer Function of a PID controller can be written as:

$$K_p + \frac{K_i}{s} + K_d s = \frac{K_d s^2 + K_p s + K_i}{s}$$

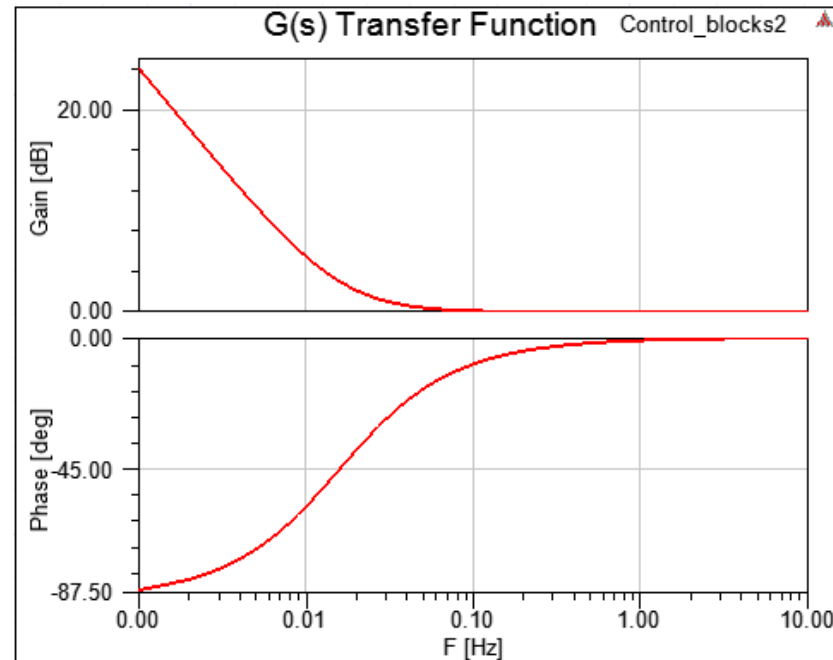
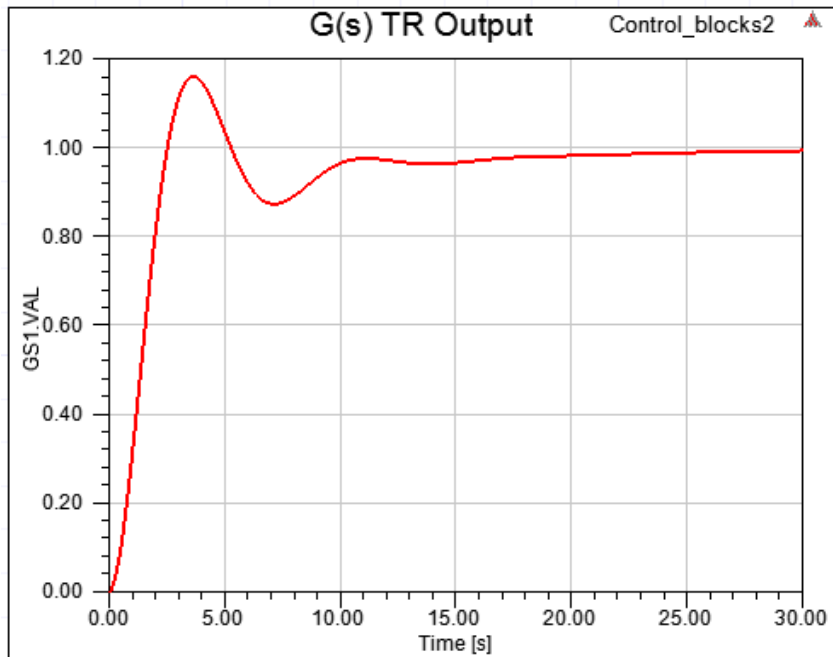
- For a pure proportional control, set:
  - $K_p = 1$
  - $K_i = 0$
  - $K_d = 0$
- Run both TR and AC simulations
- Note the results are identical to the ones obtained for Design **Control\_blocks1**. Overshoot and steady state values are in this case not controlled





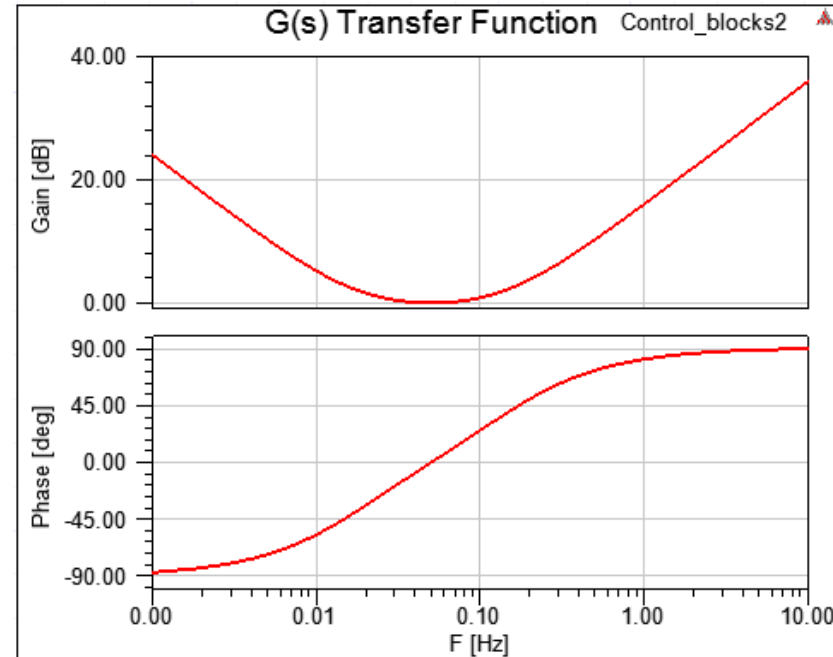
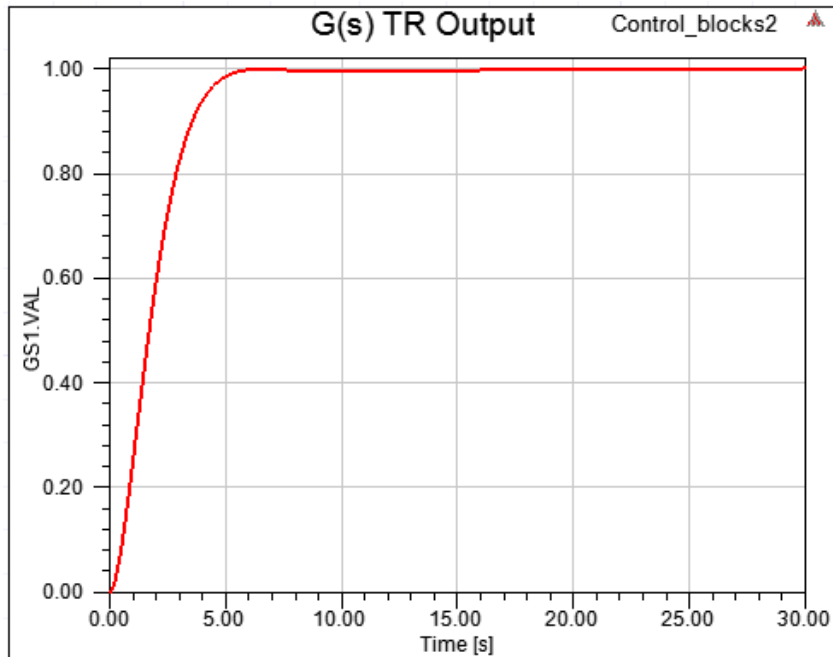
# Set PID Controller Coefficients

- For a Proportional-Integral control (PI), set:
  - $K_p = 1$
  - $K_i = 0.1$
  - $K_d = 0$
- Run both TR and AC simulations. Note the overshoot is still significant, although the steady state value is now close to 1



# Set PID Controller Coefficients

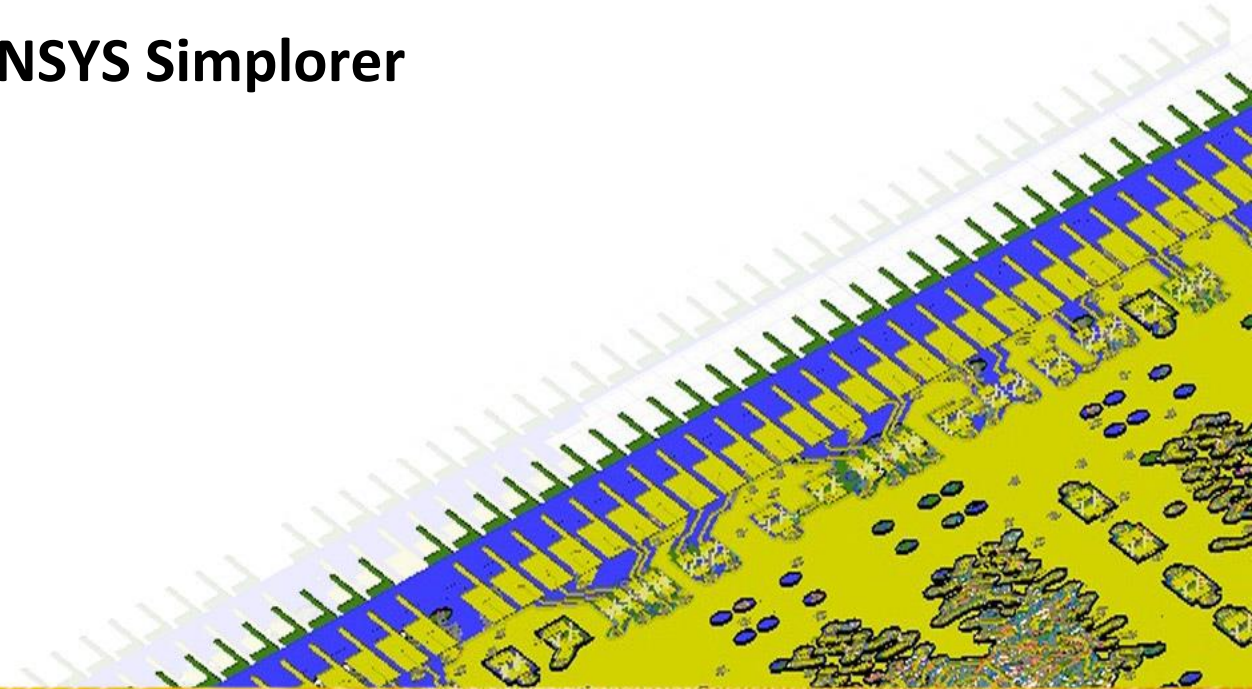
- For a complete and optimized PID controller set:
  - $K_p = 1$
  - $K_i = 0.1$
  - $K_d = 1$
- Run both TR and AC simulations and save the Project. Note how both Overshoot and steady state value are now **almost ideally controlled** (Overshoot = 0 and steady state value = 1).





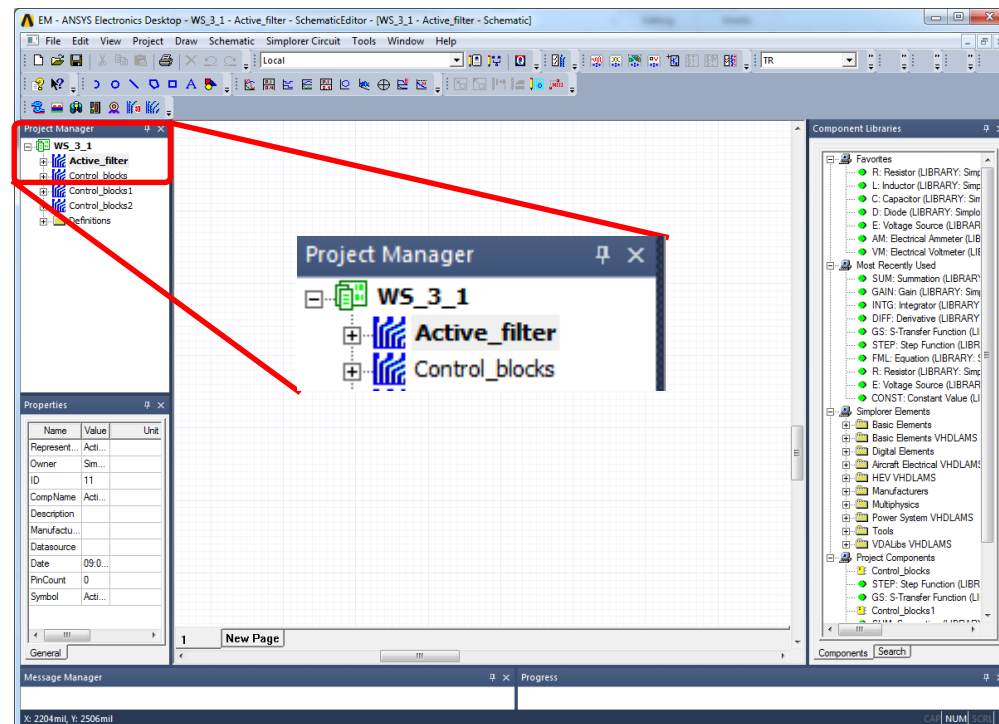
# Active Filter Performances Characterization

Introduction to ANSYS Simplorer



# Insert a New Simplorer Design

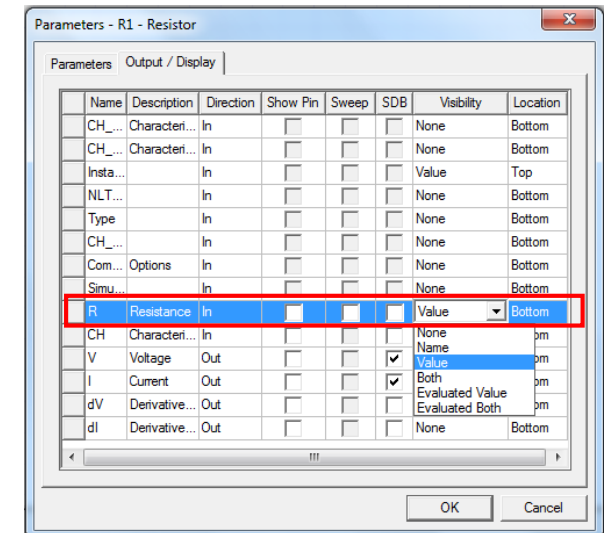
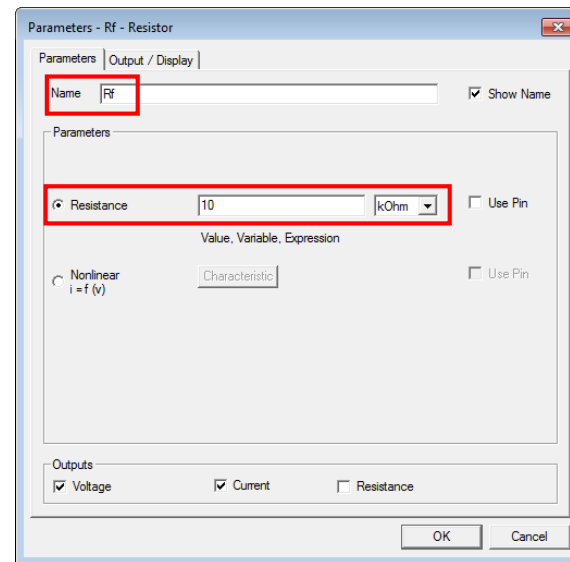
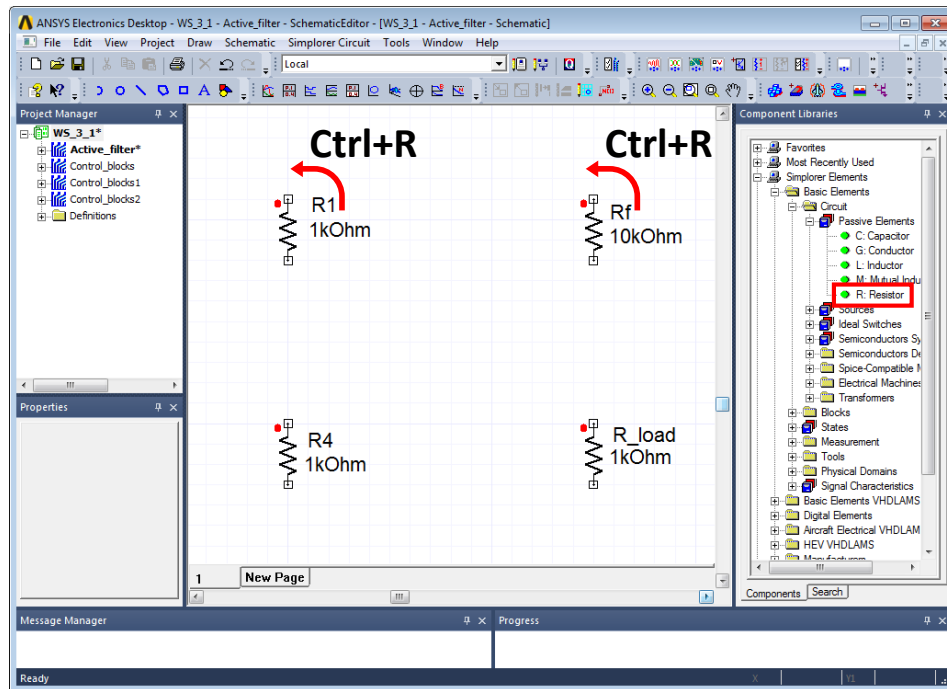
- Insert a Simplorer Design using the icon 
- Rename the Design as **Active\_filter**
- Save the project using the icon 



# Insert Components

- Resistors

- In Component Libraries window *Simplorer Elements* → *Basic Elements* → *Circuit* → *Passive Elements*
- Select the **Resistor**, drag and drop it 4 times into the Schematic. Press **Esc** key to exit the insert mode
- Double click on the Resistor upper right, change the value to **10 kΩ** and name it **Rf**. Rotate the two upper Resistors using the shortcut **Ctrl+R**. Name the Resistor in the lower right position **R\_load**
- In the **Output/Display Tab** under Visibility, select Value for Resistance for all the 4 resistors



# Insert Components – Alternative method

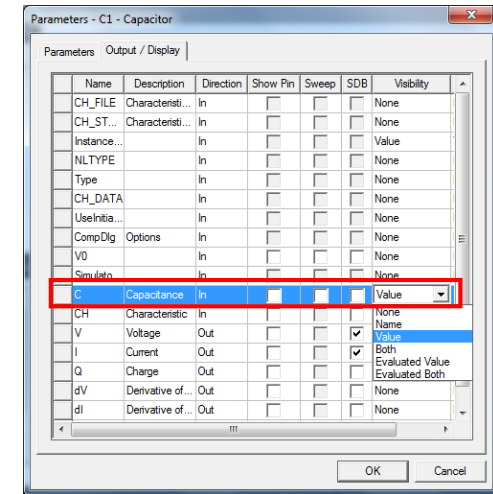
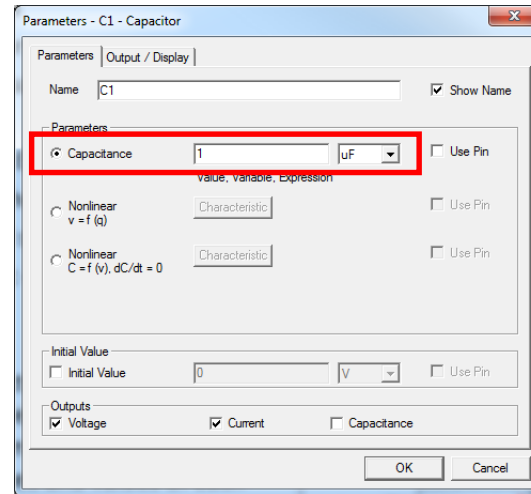
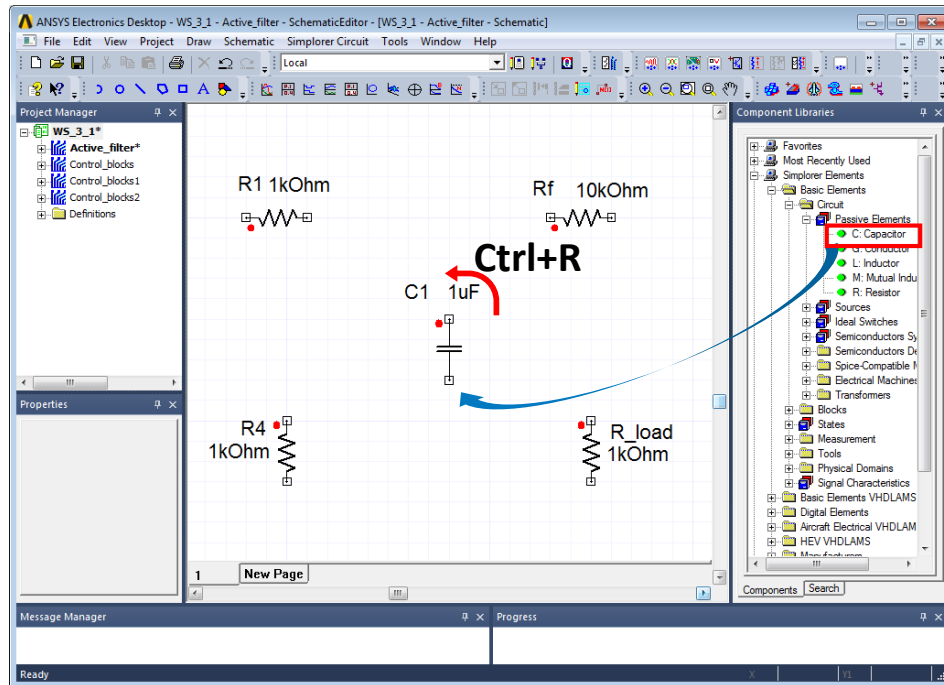
- In order to create the circuit faster, it is also possible to copy and paste already inserted components, where some properties (like e.g. values or value visibilities) have been already set
- In case of the 4 Resistors previously inserted, one could proceed the following way:
  - Insert the first Resistor and make the Resistance value visible through the **Output/Display** tab, as previously described
  - Select the inserted Resistor
  - Use the **Ctrl+C** and **Ctrl+V** keys to copy and paste the component 3 further times, obtaining a total of four identical Resistors
  - Double click on the upper right Resistor to change its value to **10 kΩ**. Name it **Rf**
  - Use the **Ctrl+R** key to rotate the two Resistors as in the previous slide
  - Name the resistor in the lower right position **R\_load**
- The above described procedure is always applicable and very useful, especially when one has to deal with numerous identical components



# Insert Components

- Capacitor

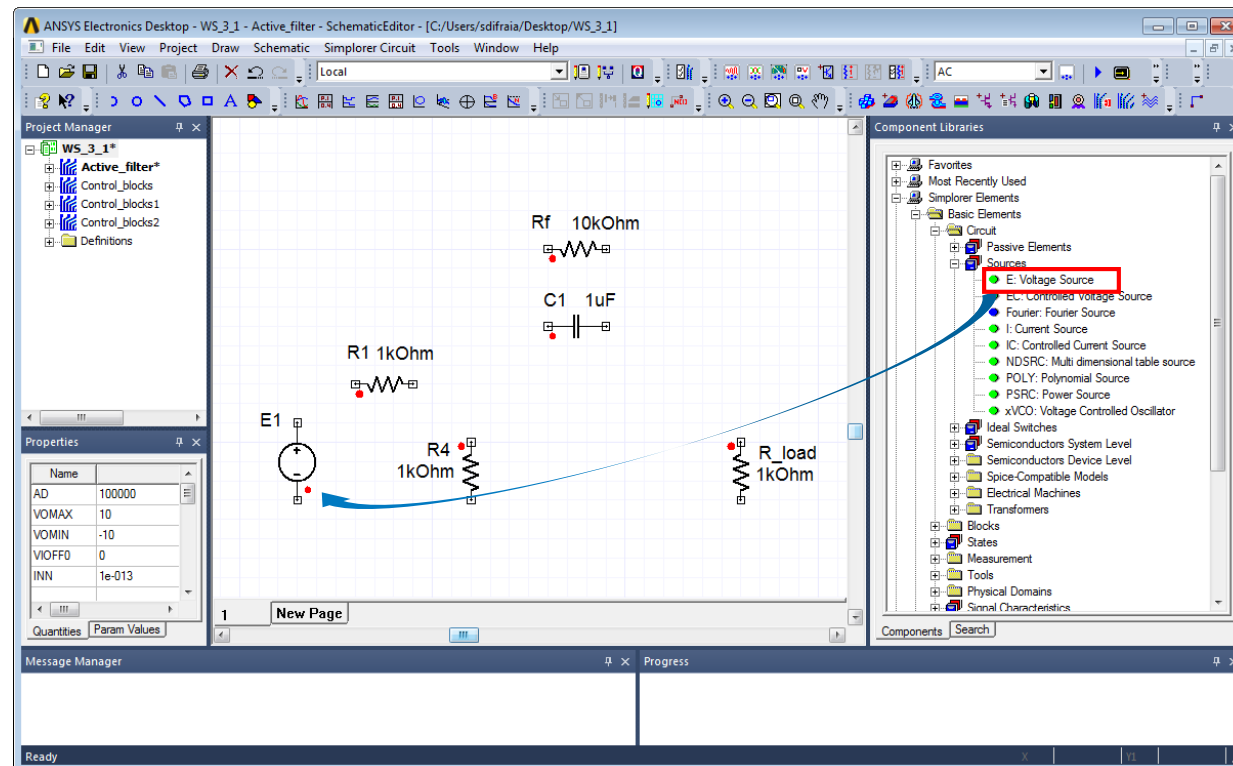
- In Component Libraries window *Simplorer Elements* → *Basic Elements* → *Circuit* → *Passive Elements*
- Select the **Capacitor**, drag and drop it into the Schematic. Press **Esc** key to exit the insert mode
- Rotate the Capacitor using the shortcut **Ctrl+R**
- Double click on the Capacitor, change the value to **1  $\mu$ F**
- In the **Output/Display Tab** under Visibility, select Value for Capacitance



# Insert Components

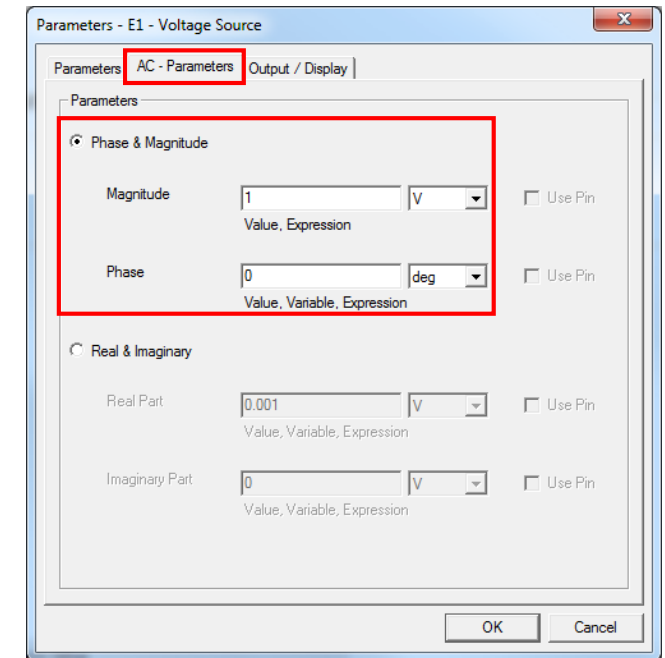
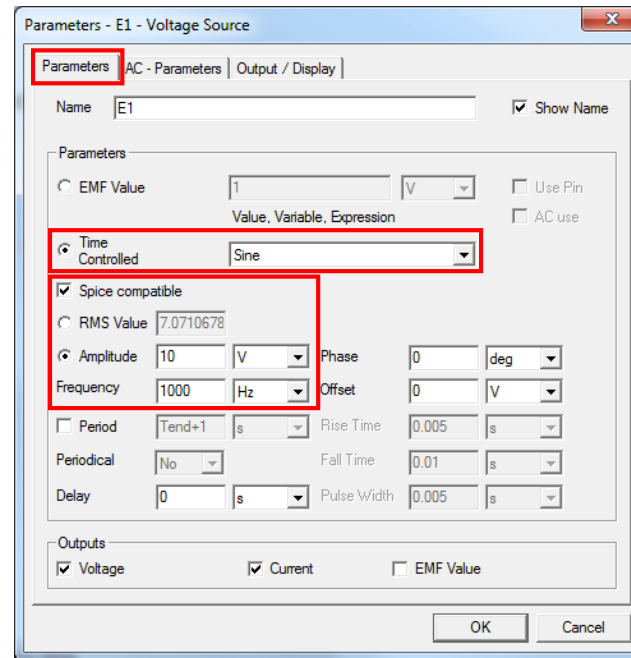
- Voltage Source

- 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



# Set Component Properties

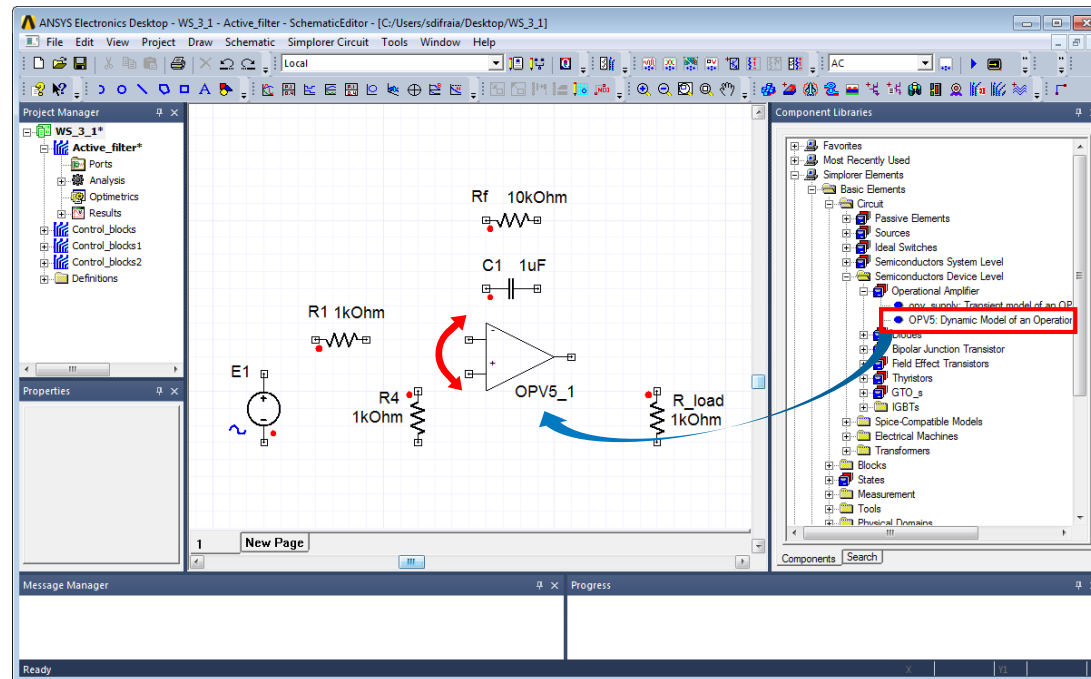
- Double click on the **Voltage Source** and select the **Parameters Tab**
  - Time Controlled: ☒ **Checked – Sine**
  - Spice Compatible: ☒ **Checked**
  - Amplitude: **10 V**
  - Frequency: **1000 Hz**
- Select the **AC-Parameters Tab**
  - Phase and Magnitude: ☒ **Checked**
  - Magnitude: **1 V**
  - Phase: **0 deg**




# Insert Components

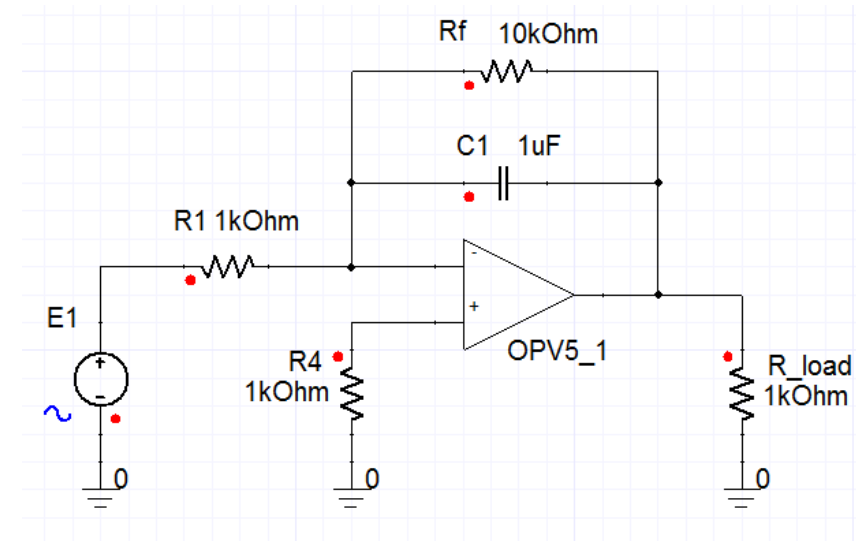
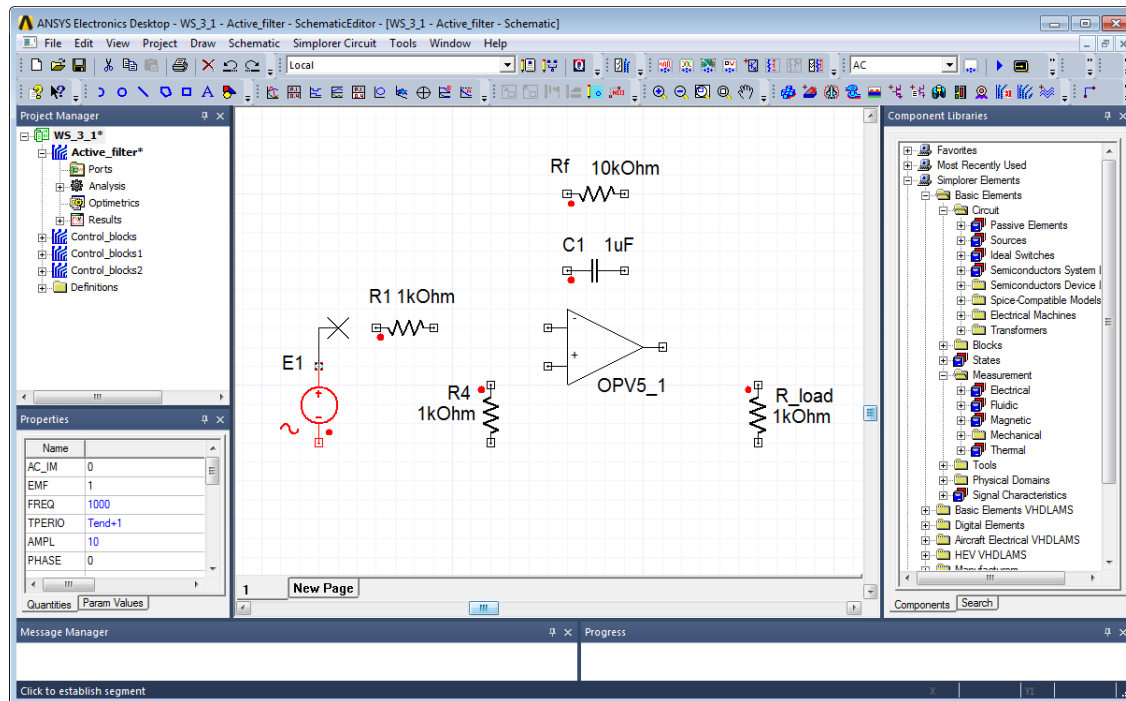
- **Operational Amplifier**

- In Component Libraries window *Simplorer Elements* → *Basic Elements* → *Circuit* → *Semiconductors Device Level* → *Operational Amplifier*
- Select the **OPV5: Dynamic Model of an op-amp**, drag and drop it into the Schematic. Press **Esc** key to exit the insert mode. Use the shortcut **Ctrl+D** to fit the view
- Flip the **op-amp** Vertically so that the “-” is on top



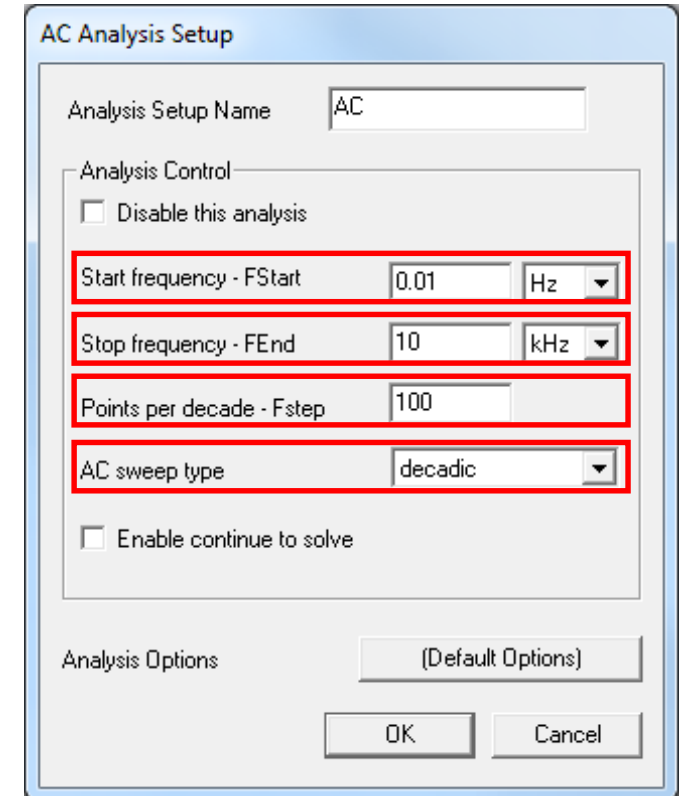
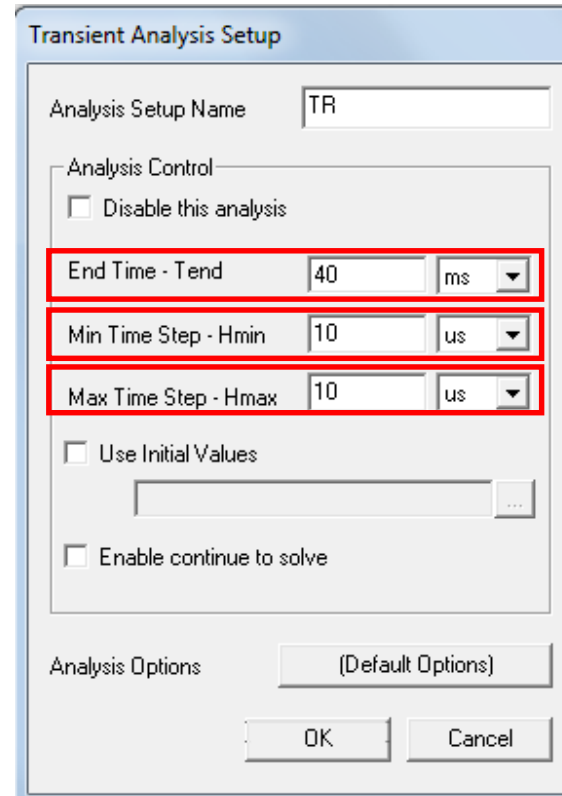
# Connect Components

- 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 3 times 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**



# Setup the Simulation Analysis

- We will perform a Transient analysis as well as an AC analysis, so that we can plot results both in time domain and in frequency domain (Bode plots)
- In the Transient Analysis Setup window:
  - Tend: **40 ms**
  - Hmin: **10  $\mu$ s**
  - Hmax: **10  $\mu$ s**
  - Press **OK**
- To insert an AC analysis select menu item **Simplorer Circuit  $\rightarrow$  Solution setup  $\rightarrow$  Add AC...**
- In the AC Analysis Setup window:
  - Start Frequency - FStart: **0.01 Hz**
  - Stop Frequency - FEnd: **10 kHz**
  - Points per decade - FStep: **100**
  - AC sweep type: **Decadic**
  - Press **OK**

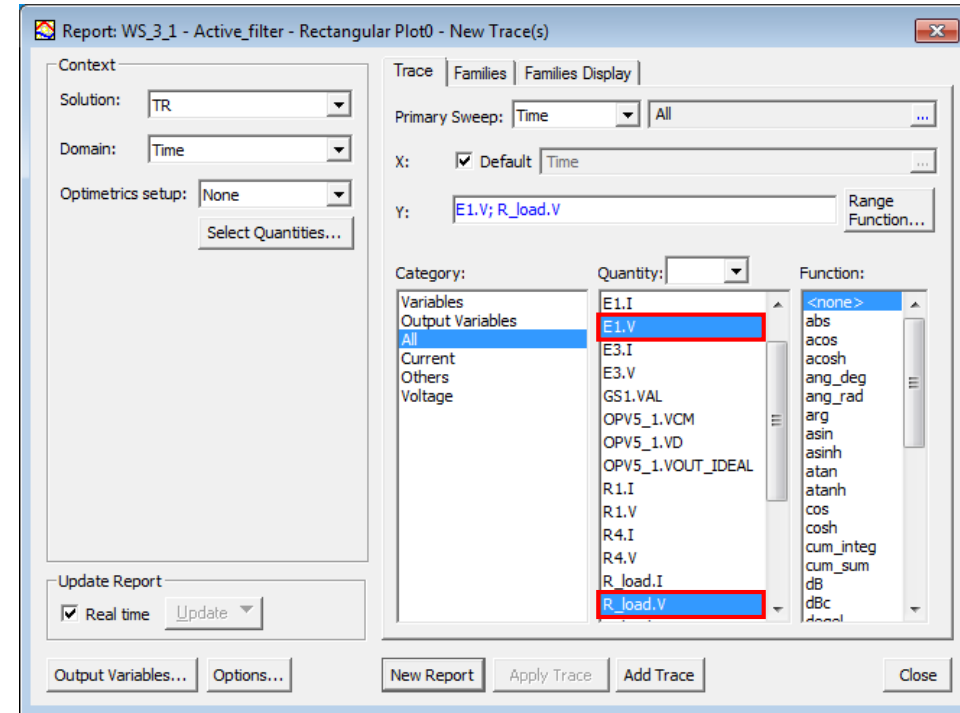
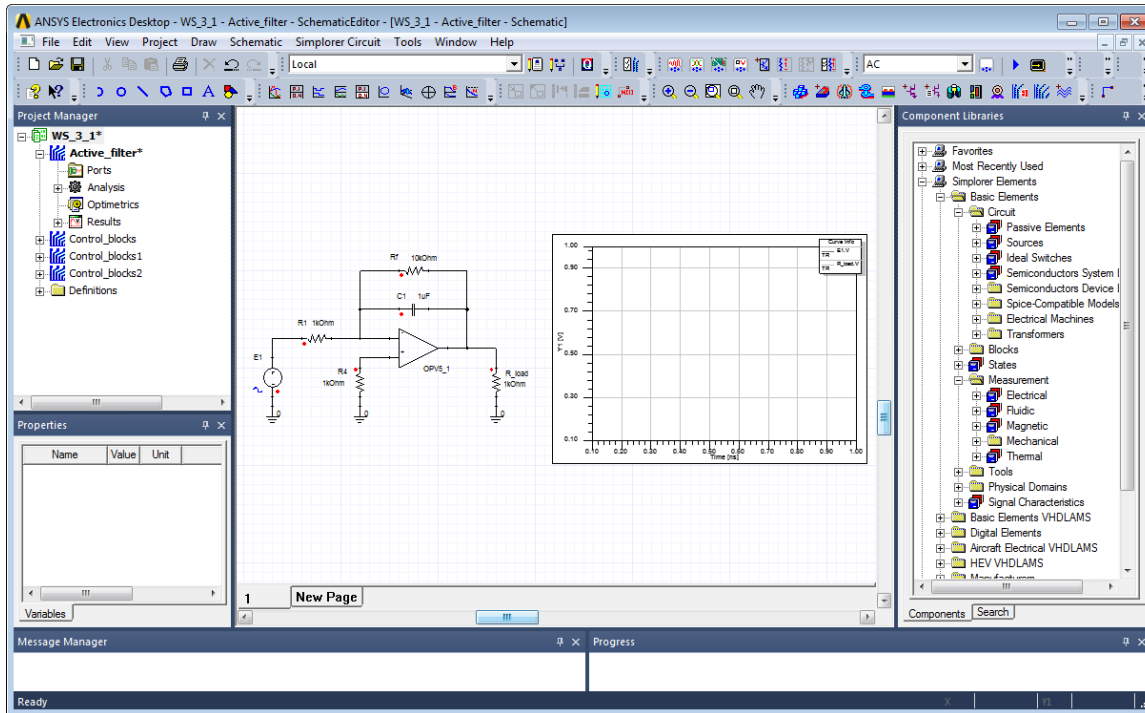


**Note:** you can insert an AC analysis also through RMB over Analysis in Project Manager window  $\rightarrow$  Solution Setup  $\rightarrow$  Add AC



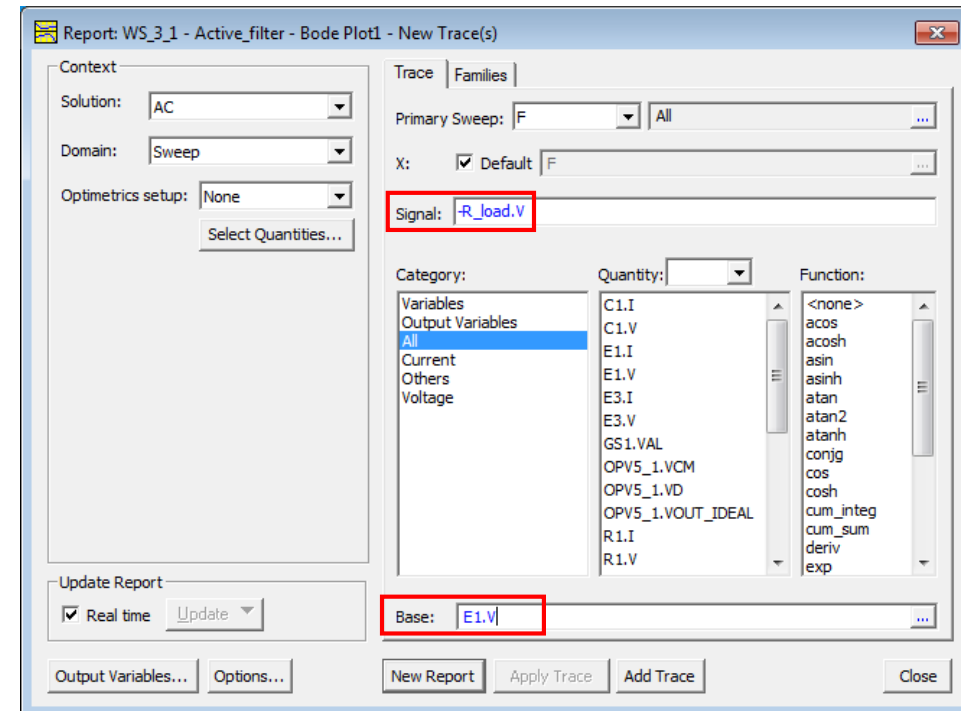
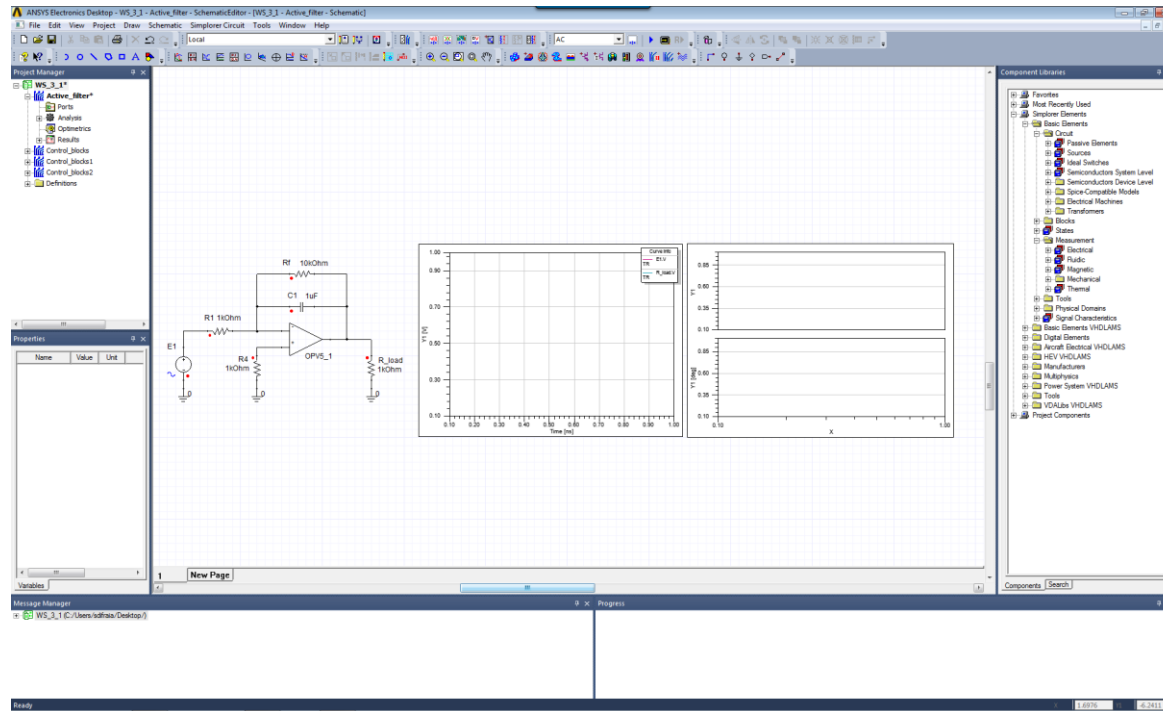
## Prepare the Postprocessing

- 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
- Select the Voltages across E1 and R\_load as output signals by checking the quantities **E1.V** and **R\_load.V**
- Click on the **Add Trace** button and then **Close**



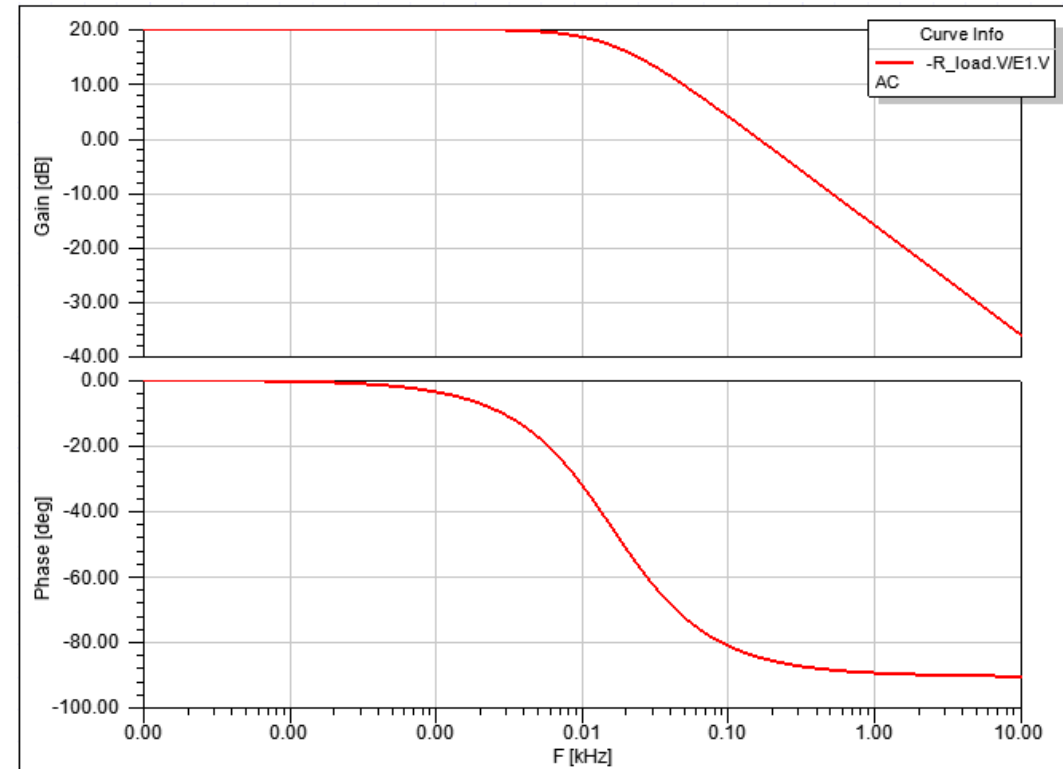
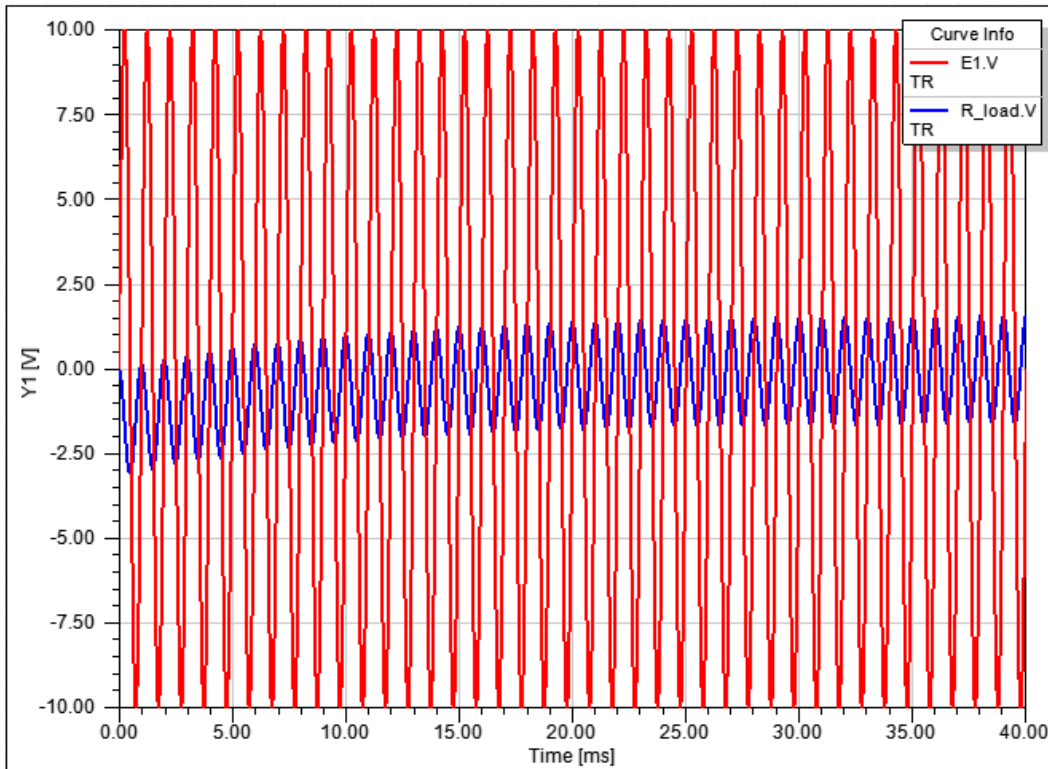
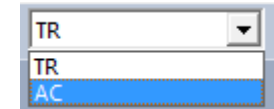
# Prepare the Postprocessing

- Select the menu item **Draw** → **Report** → **Bode Plot** and place a second plot in the Schematic, for example on the right of the previous plot. In the **New Trace** window, select “Solution” to be **AC**
- Select the **–Rload.V** quantity to be the “Signal”. The “–” is there to compensate for the fact the active filter has a 180 deg phase shift as in inverting configuration
- Select the **E1.V** signal to be the “Base”
- Click on the **Add Trace** button and then **Close**



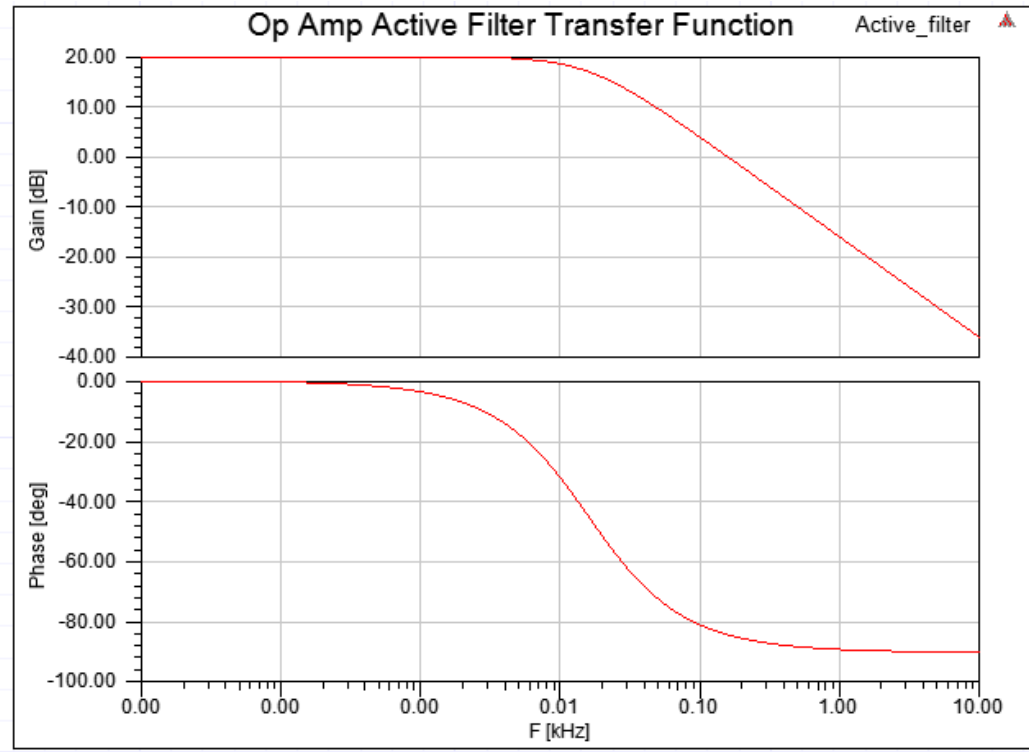
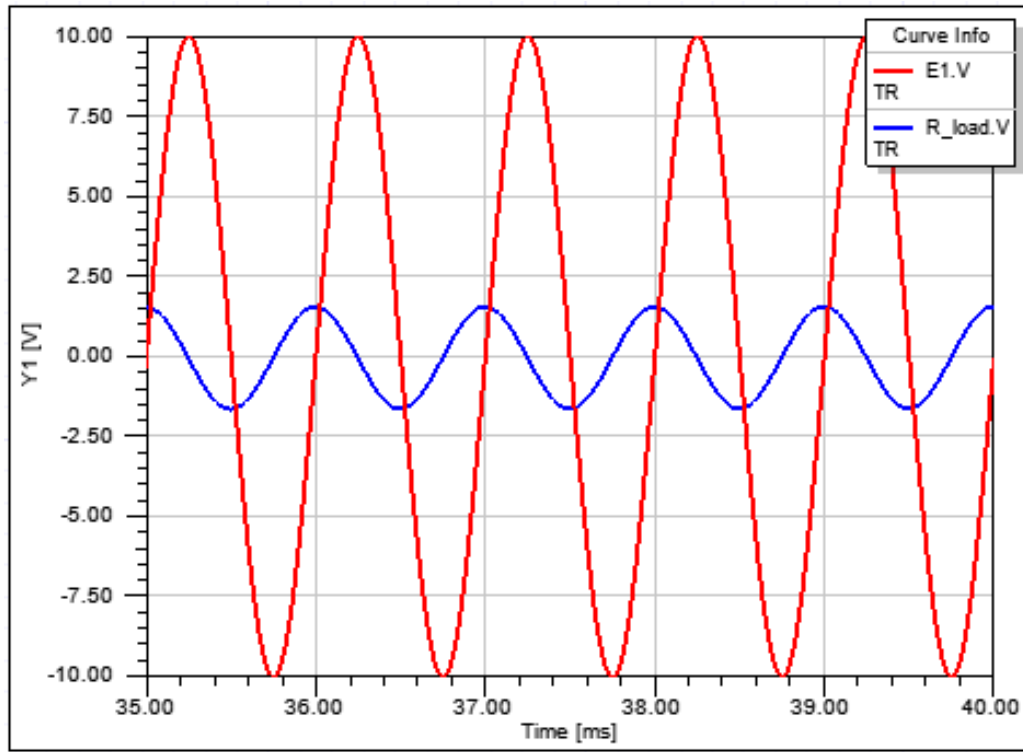
# Analyze and View Results

- Select the menu item *Simplorer Circuit* → *Analyze* to run the Simulation
- Change the Analysis type to *AC* from the menu and run the simulation again
- The results should appear as shown below
- Menu item *File* → *Save*



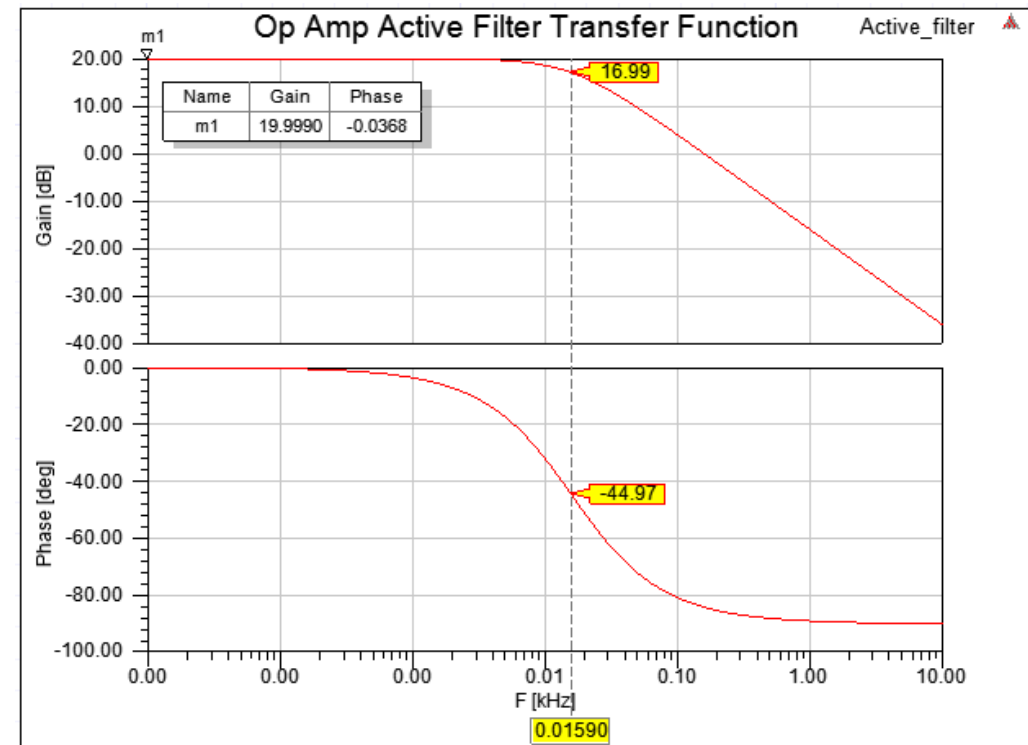
# Adjust the Plots

- Adjust the TR Plot to view the axis from 35 ms to 40 ms
- Adjust the Bode (AC plot) so it only displays the Header, and change the name of the Header to “Op Amp Active Filter Transfer Function”
- The results should appear as shown below



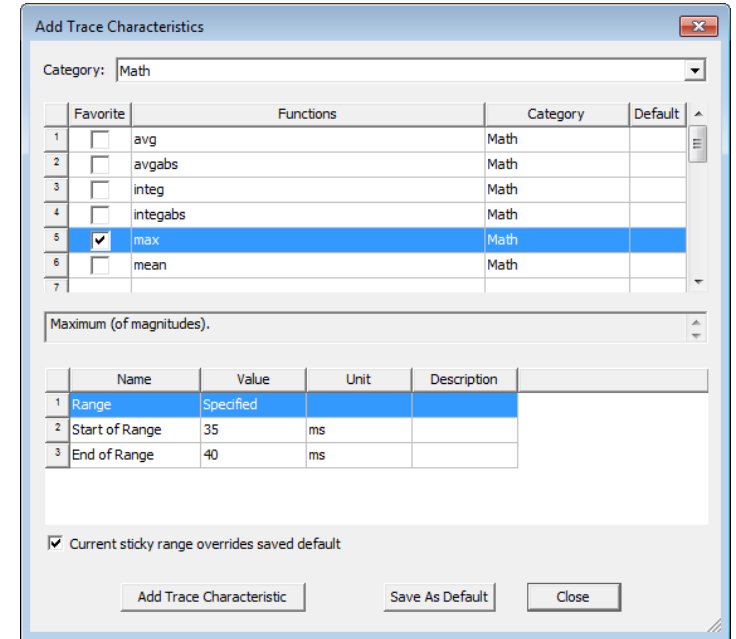
# Evaluate Results

- DC Gain of the filter is given by  $A_v(dc) = R2/R1 = 10k\Omega/1k\Omega = 10 \rightarrow 20\text{dB}$
- Filter pole break frequency ( $F_p$ ) is given by  $\omega_p = 1/(R2*C1) = 100 \text{ rad/s} \rightarrow F_p = \omega_p/2\pi = 100/2\pi = 15.9 \text{ Hz}$
- Open the Bode plot from the Results section in the Program Manager window
  - Add Marker for the DC gain (**RMB**  $\rightarrow$  **Marker**  $\rightarrow$  **Add Marker**)
  - Add X Marker for  $F_p$  (**RMB**  $\rightarrow$  **Marker**  $\rightarrow$  **Add X Marker**)
  - Press **Esc** Key to exit marker mode
- Note that from the Bode Plot, the DC gain is **20 dB**
- Note  $F_p$  occurs at approximately **3 dB** from the DC gain (**17 dB**)
- Move the X Marker to 1 kHz, and note the Filter Gain at 1kHz is approx. -16 dB  $\rightarrow$  voltage gain =  $10^{(-16/20)} = \mathbf{0.158}$
- Note also the final phase is -90 deg, validating a single pole
- Clear all markers (**RMB**  $\rightarrow$  **Marker**  $\rightarrow$  **Clear All**)

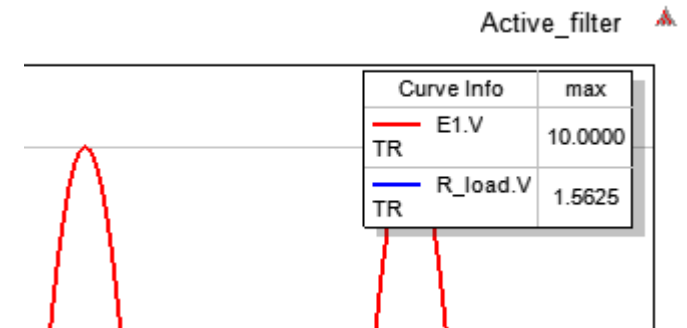


# View Trace Characteristics

- Open the Rectangular Plot for the TR analysis and measure the peak value of the input and output voltages using **RMB → Trace Characteristics → All...**
- From the pull down Menu, choose Category: **Math**
- From the list of Functions select: **max**
  - Range: **Specified**
  - Start of Range: **35 ms**
  - End of Range: **40 ms**
- Press **Add Trace Characteristic**



- This should yield the results on the right
- Save the Project

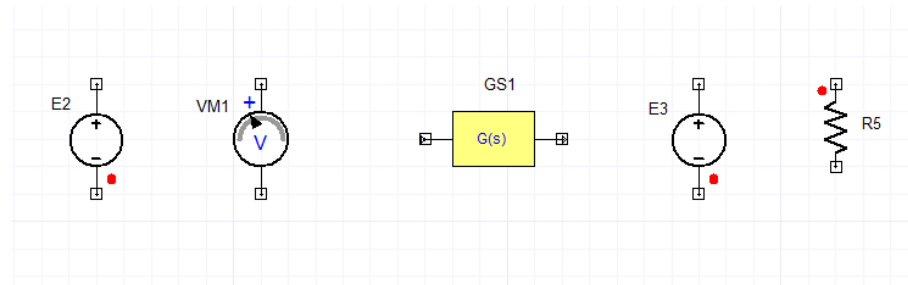


- Note the output voltage at 1000Hz is 1.56 V<sub>p</sub> where the input voltage is 10 V<sub>p</sub>, which yields the gain of 1.56/10 = 0.156 (as shown by the Bode Plot at 1kHz of approximately -16dB)

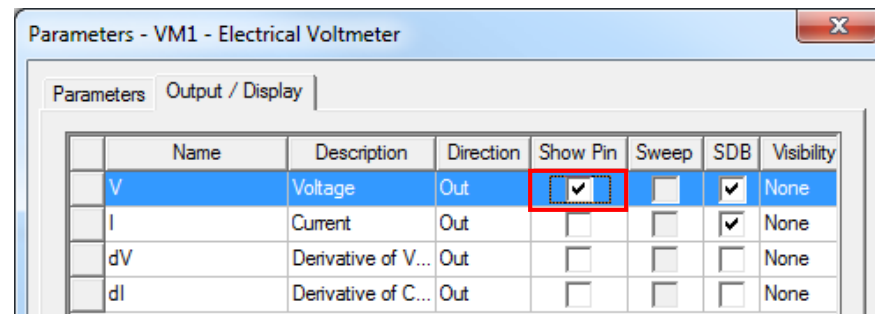


# Create an Equivalent Transfer Function

- A  $G(s)$  transfer function block will now be created to represent the **op-amp** Active filter
- Add the following components to the “Active Filter” design
  - **Basic element** → **Blocks** → **Continuous Blocks** → **GS: S-Transfer function** (one)
  - **Voltage Source** (two)
  - **Resistor** (one)
  - **Basic Element** → **Measurement** → **Electrical** → **VM: Electrical Voltmeter** (one)
- Arrange them as shown

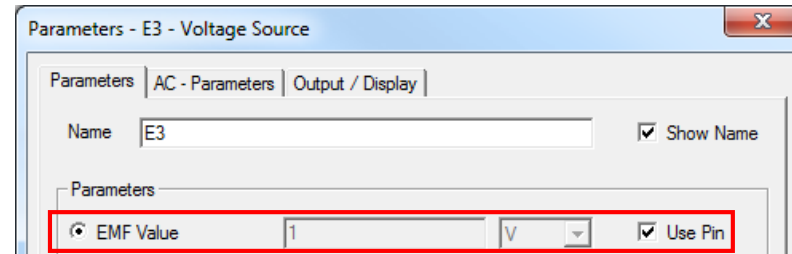


- Double click on the VM1 Voltmeter, select the “Output/Display” tab and select the box to “**Show Pin**” for the Voltage “**V**”. Flip the Voltage Meter Horizontally

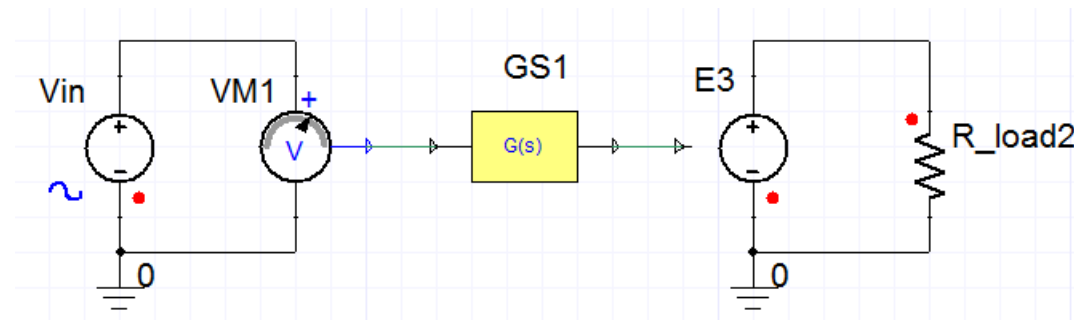


# Create an Equivalent Transfer Function

- Double click on the E3 Voltage source and select the EMF value to be a pin (this pin determines the EMF value of the source)



- Name E2 as **Vin** and R5 as **R\_load2**. Connect the circuit as shown below. The **GS1** block represents the Active Filter, **Vin** is the input to the filter, **VM1** the interface with the control blocks because the VM1 output pin represents a non-conservative node, which is compatible with the control blocks I/O
- In order to interface the control block non-conservative node output with conservative components, it is connected as the input pin to the voltage source **E3**
- To complete the circuit add twice the Ground component



# Set up G(s) Block Properties

- Double click on the voltage source **Vin** and define it to be the same as **E1**
- Since the active filter pole frequency “ $F_p$ ” was 15.9Hz ( $\omega_p = 100$  rad/s), and DC gain was 20dB (10), the representing transfer function is:  **$G(s) = 1000/(s + 100)$**
- Note **-1000** for the numerator taking into account the 180 deg phase shift due to the op-amp inverting configuration

Parameters - GS1 - S-Transfer Function

Parameters | Output / Display

Name: GS1 ☒ Show Name

Input Signal: VM1.V ☒ Use Pin

**Numerator**

Order: 0

| Coefficient | Value |
|-------------|-------|
| B[0]        | -1000 |

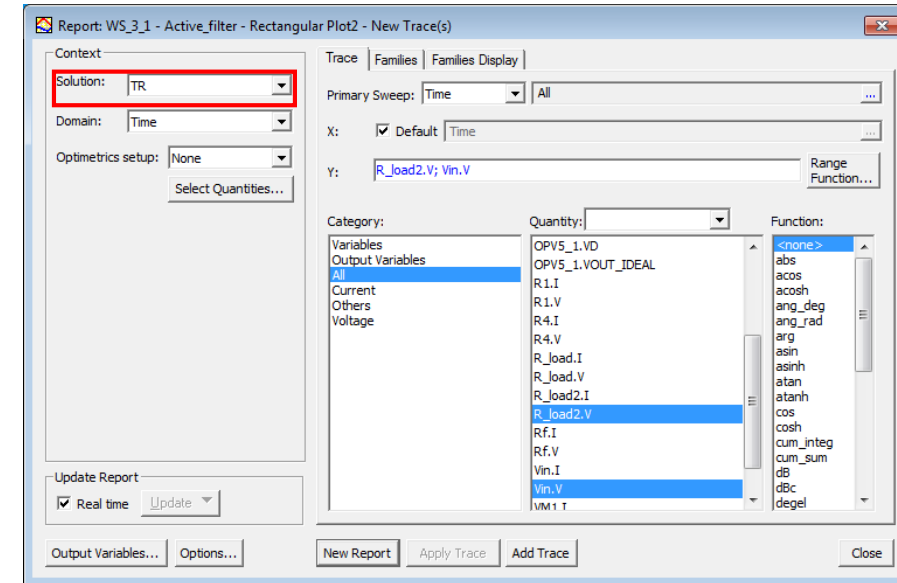
**Denominator**

Order: 1

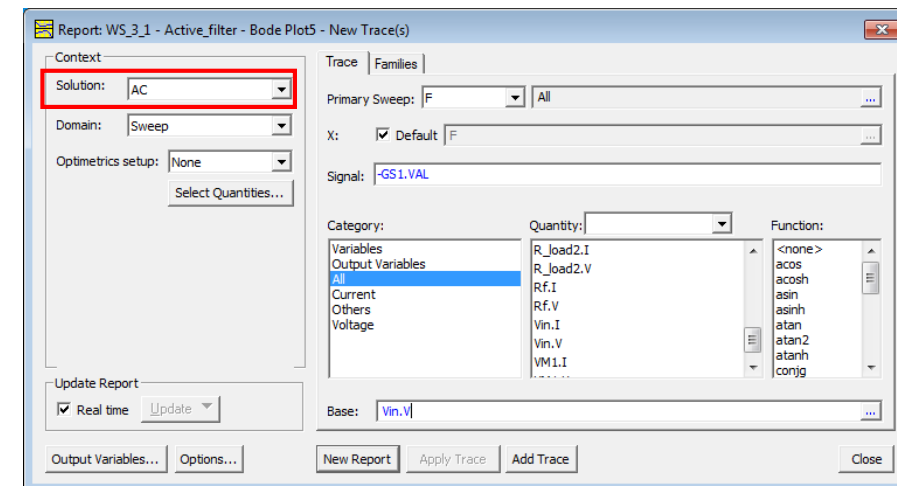
| Coefficient | Value |
|-------------|-------|
| A[0]        | 100   |
| A[1]        | 1     |

# Preparing the Postprocessing

- Add a Rectangular plot, make sure to select the “TR” as the “Solution” in the upper left, define the signals to be **Vin.V** and **R\_load2.V**
- *Add Trace → Close*

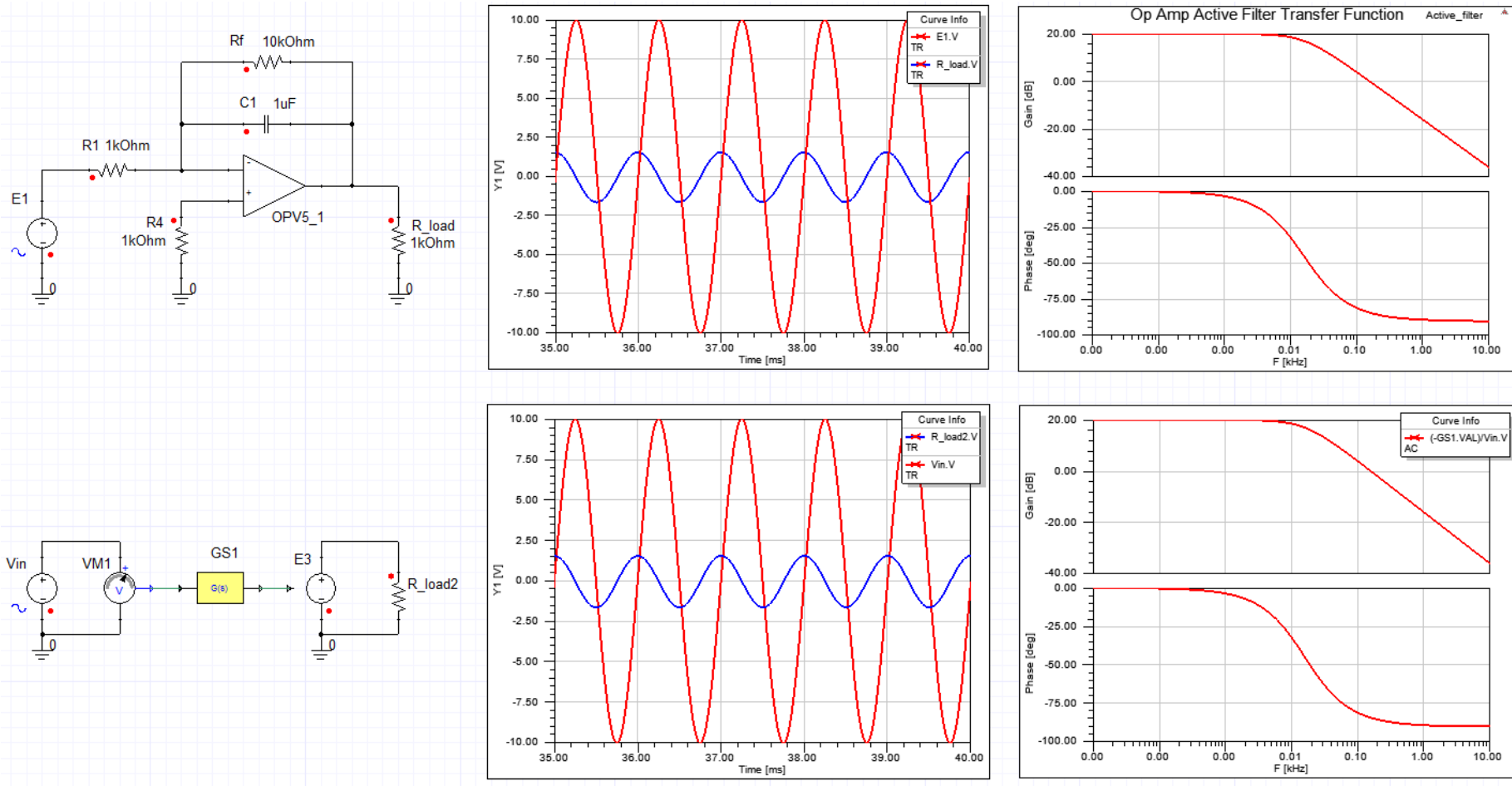


- Add a Bode plot, define the base to be **Vin.V** and the Signal to be **– GS1.VAL**
- *Add Trace → Close*



# Analyze and View Results

- Run both the **TR** and **AC** analysis, adjust the Rectangular TR plot to display only from 35 ms to 40 ms as was done previously. The results should appear as shown below. Note results are the same as for the Active Filter implementation



# Saving the Project

- This completes the workshop
- Save the file with the name **WS\_3\_1** in the working folder