TECHNICAL OVERVIEW

Chapter 5

RF System Design

PathWave Advanced Design System (ADS)

PATHWAVE

Keysight PathWave Advanced Design System (ADS) provides all the necessary capabilities to perform RF system design and simulations. RF system design is an important and critical step to validate the system performance for first pass success. RF system architecture can be implemented using RF System models available in the Analog/RF library.

Case Study 1: Receiver System Design

1. Create a new workspace **Lab5_RF_System_Design_wrk**. Create a new schematic cell and name it **Lab5a_RFSystemDesign**.

2. Place **Amp** and **Mixer2** from the **System-Amps & Mixers** component library palette menu. Set their characteristics as shown in Figure 1.

3. Place the **Chebyshev Bandpass Filter** component at the Mixer output from **FiltersBandpass** component library palette menu. Set the characteristics as shown in Figure 1.

4. Copy and paste the amplifier twice after the BPF component. These amplifiers will be used as two-stage IF amplifiers. Change the following specifications:

   TOI = 20

   SOI = 30

   NF = 3 dB

KEYSIGHT TECHNOLOGIES

5. Once completed, the schematic will look similar to the one shown in Figure 1.

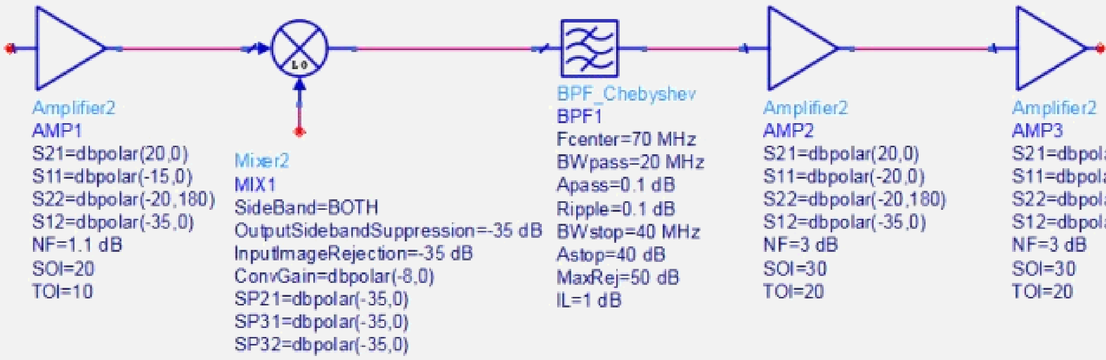

Figure 1. Initial Schematic

6. The next step is to connect the RF and LO sources and setup the simulation to observe the system response. Place **P_1Tone** and **Osc** source from the **Sources-Freq Domain** library. Connect the components and set their characteristics as shown in Figure 2. Make sure to change the PhaseNoise list in the Osc source.

7. Place the **HB** (Harmonic Balance) simulation controller from the **Simulation-HB** library and set its characteristics as show below.

8. Place a **Term** component at the output (after the second IF amplifier). Click **Wire Label**. Enter **vout** in the pop-up window, and click the “+” terminal of the Term component.

9. Once completed the receiver system diagram will look like the design shown in Figure 2.

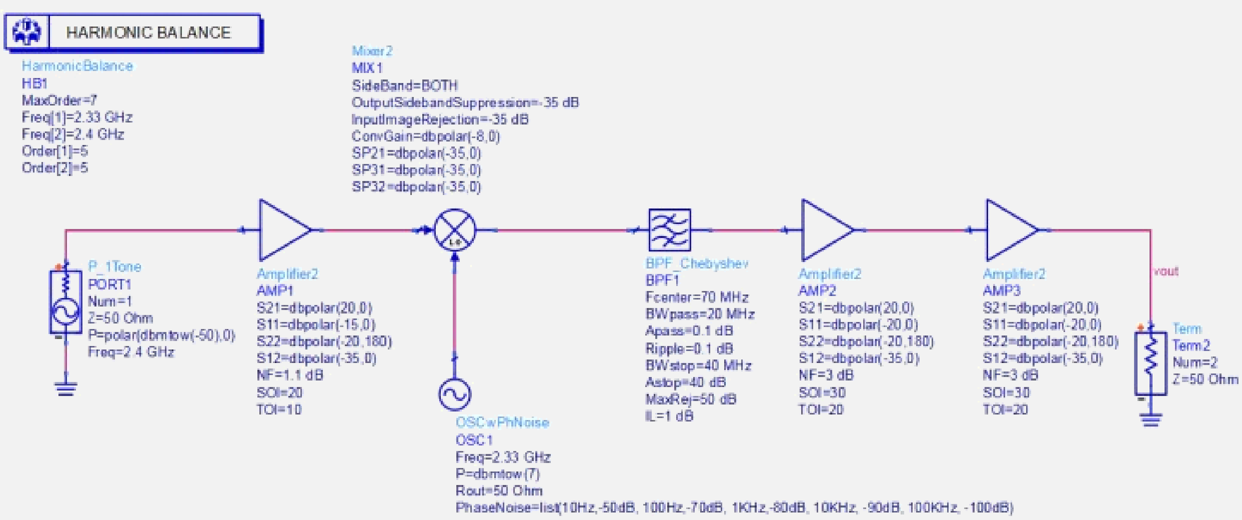

Figure 2. Schematic

10. Save the design and click the **Simulate** icon. Insert a rectangular plot in the data display window. Add **vout** from the measurement list and select **Spectrum in dBm** to observe the output spectrum.

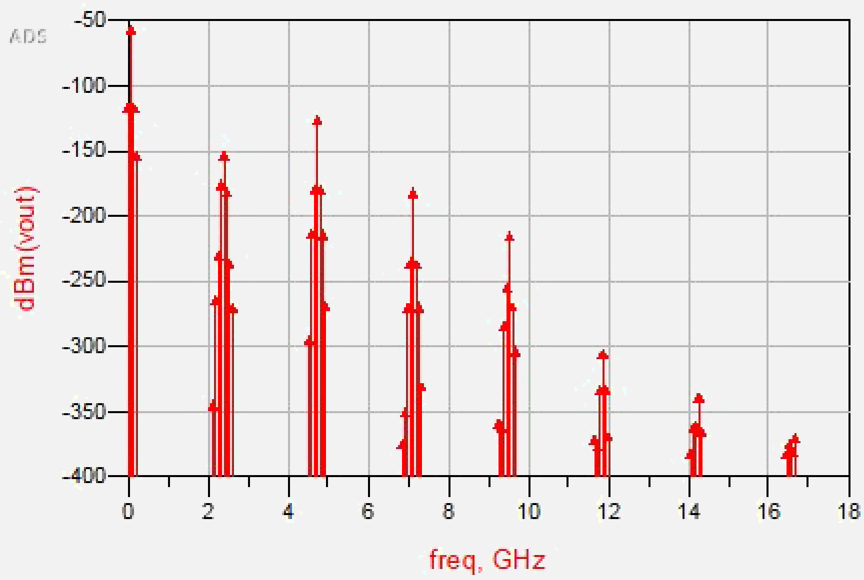

Figure 3. Vout spectrum

Case Study 2: Phase Noise Simulation

Phase noise is an important simulation for receiver systems. The example below shows how to perform phase noise analysis using the Harmonic Balance simulator in Pathwave ADS.

1. Right-click **Lab5a_RFSystemDesign** and click **Copy Cell**.
2. In the pop-up window, give name the copied cell **Lab5b_RFSystemDesign_PhaseNoise**.
3. Open the schematic design for the copied cell. From **Simulation-HB** library place **NoiseCon** (Noise Controller) block onto schematic.
4. Double-click the NoiseCon block to open the properties. Set the following parameters:

   Freq tab:

   Sweep Type = Log

   Start = 10 Hz

   Stop = 100 kHz

   Num. of pts. will automatically become 5. This indicates five noise analysis frequencies (i.e. 10 Hz, 100 Hz, 1 kHz, 10 kHz and 100 kHz). These are the same frequencies specified for the oscillator, which was used as an LO source in the system.

Nodes tab:

Select **Pos Node = vout** from the drop-down box, which is the output node where we provided a label in the earlier lab exercise

Click **Add**.

Phase Noise Tab:

Phase Noise Type = Phase Noise Spectrum

Under **Specify Phase Noise Carrier**, specify **Frequency** as **70 MHz**. Alternatively, we can also specify carrier mixing indices such as {-1,1} etc.

5. Click **OK**. The Noise Controller is setup. We have one extra step in linking this Noise Controller to our HB simulation controller.

6. Double-click HB controller, and go to the **Noise** tab.

   Check **Noise Cons** option.

   From **Edit** drop-down box, select **NC1** (name of noise controller).

   Click **Add**.

   Click **OK**. Close the HB simulation controller properties box.

7. Run the simulation. A new data display window will come up. Insert a new rectangular plot and select **vout** to be plotted in **dBm** to see the same spectrum as in the earlier lab.

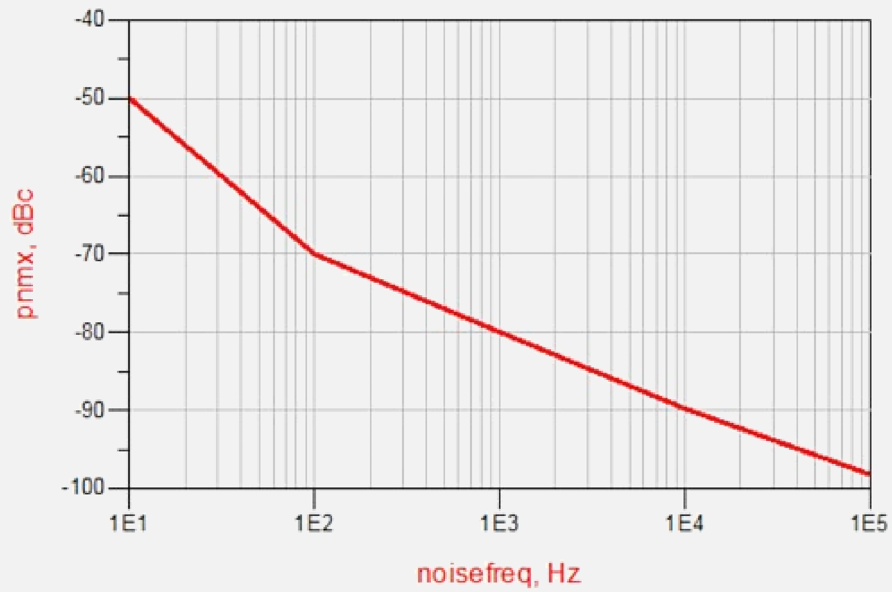

Figure 4. Vout spectrum after noise

8. Insert a new rectangular plot. In the window that pops up, select **pnmx** (i.e. Phase Noise). In the **Plot Options**, change the **X-axis** to **Log**. Click **OK** to see the Phase Noise plot at various offsets.

.

Case Study 3: Two-Tone Simulation of the Receiver System

Performing a two-tone simulation is important for system level analysis. The example here shows how to perform two-tone simulations on frequency converting based systems.

1. Right-click **Lab5a_RFSystemDesign** and click **Copy Cell**.

2. In the pop-up window, enter the name as **Lab5c_RFSystemDesign_2Tone** and click **OK**.

3. Open the schematic of the copied cell. For a two-tone simulation, we need to change the **P_1Tone** source, which is currently used for the RF source. Delete the one-tone source. From the **Sources-Freq Domain** library, place a **P_nTone** source on the schematic.

4. Double-click the **P_nTone** source and edit the properties as below:

   Click **Freq[1]** and enter **2.399 GHz**. Click **Apply**.

   Click **Add** to insert a second tone frequency with the name **Freq[2]**. Enter the frequency as **2.401 GHz**. Click **Apply**.

   Click **P[1]**, which is the power in the first tone. Enter the power as **polar(dbmtow(-50),0)**. Click **Apply**.

   Click **Add** to insert power for the second tone. Enter the same power as for the first tone. Please note that for a two-tone test, it is mandatory to have the same power in both tones. If this is not the case, the analysis will not be valid.

5. Since we have three source frequencies in the schematic, we need to modify the HB simulator to specify these frequencies for proper mixing product calculations. Double click the HB controller and specify the following frequencies, in order:

   Freq[1] = 2.33 GHz

   Freq[2] = 2.399 GHz

   Freq[3] = 2.401 GHz

6. Click the **Simulate** button to run the simulation. Using the graph zoom icon in the data display, zoom in near 70 MHz to see the two-tone simulation results as shown in Figure 5.

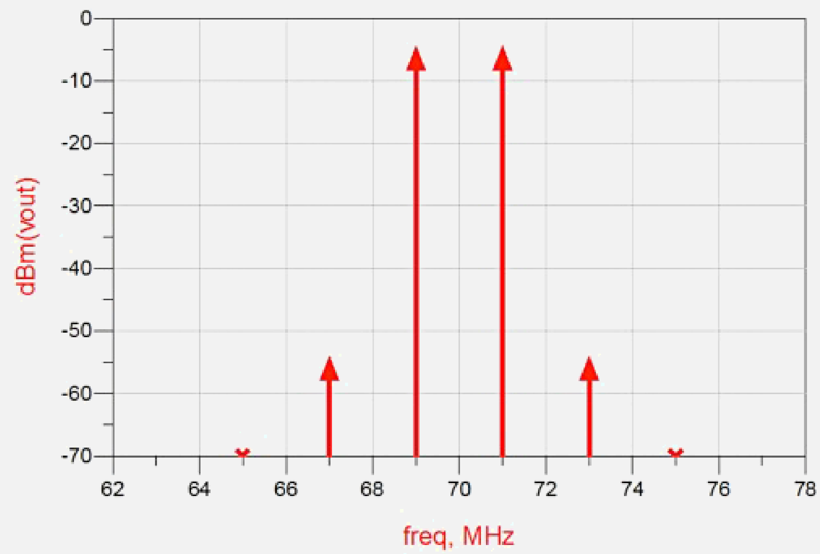

Figure 5. Vout spectrum 2tone simulation

Case Study 4: RF System Budget Analysis

Performing RF system budget analysis is very useful to characterize the system behavior and analyze how a system behaves as the sign passes through each component. The easiest way to perform RF system budget analysis is by using the Budget Controller, which offers more than 40 built-in budget measurements.

When using the Budget Controller, it is important to remember that the system should only have two-port components, except for S2P files, and an AGC Amplifier with Power Control. The PathWave ADS Simulation-Budget library provides a special mixer with an internal LO to analyze super-heterodyne systems.

Step 1: Modifying the RF System Design

1. Right click **Lab5a_RFSystemDesign** and click **Copy Cell**. Set the name of the copy to **Lab5d_RFSystemDesign_Budget**.

2. In order to perform budget analysis on our receiver system, we need to replace the mixer and LO source with the **MixerWithLO** component from the **Simulation-Budget** library as shown in Figure 7. For this component, modify the following parameters:

   ConvGain=dbpolar(-8,0)

   DesiredIF=RF minus LO

3. Delete the HB controller. Insert the **Budget** controller from the **Simulation-Budget** library. Double-click the controller, and modify the parameters under the **Setup** tab as shown in Figure 6.

4. In the **Measurements** tab, select the following measurements and select **>>Add>>**. The measurements should appear under **Selected Measurements**.

   Cmp_NF_dB
   Cmp_OutP1dB_dBm
   NF_Refln_dB
   OutTOI_dBm
   OutSOI_dBm

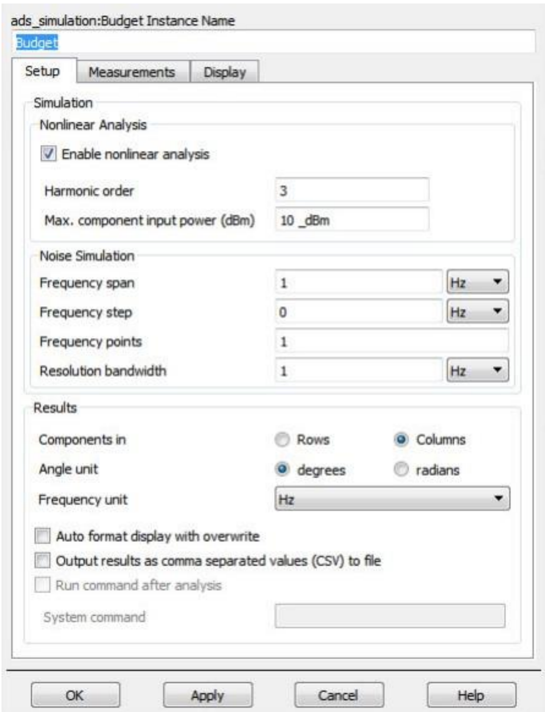

Figure 6. Budget simulation

Step 2: Performing Budget Analysis

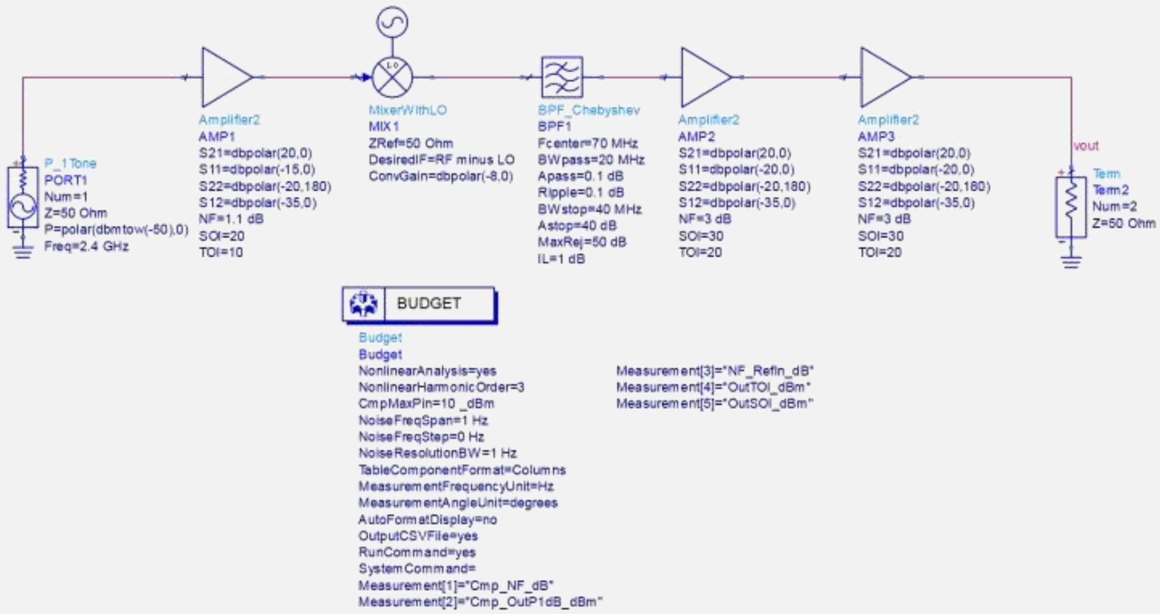

Figure 7. Budget analysis schematic

1. The overall schematic should look like Figure 7. Click **Simulate**.

2. Insert a rectangular graph on the data display page. In the pop-up window, select **OutTOI_dBm** from the measurement list, click the **Add Vs>>** button and select **Cmp_RefDes**. This will plot the budget measurement results versus the component names so that it is easier to compare the results to the schematic.

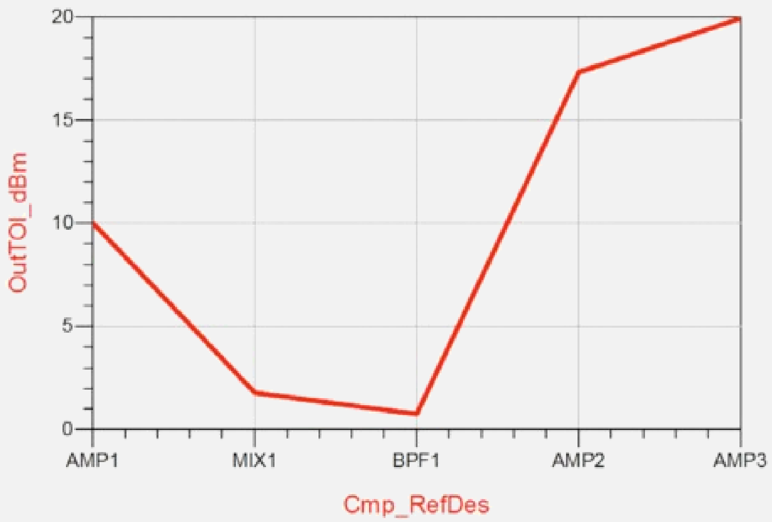

Figure 8. OutTOI spectrum

3. Following the same process as the previous step, create rectangular plots for **Cmp_OutP1dB_dBm**, **OutSOI_dBm**, and **NF_RefIn_dB**. Make sure to plot each of these parameters versus **Cmp_RefDes**. The results are shown in Figures 9 to 11.

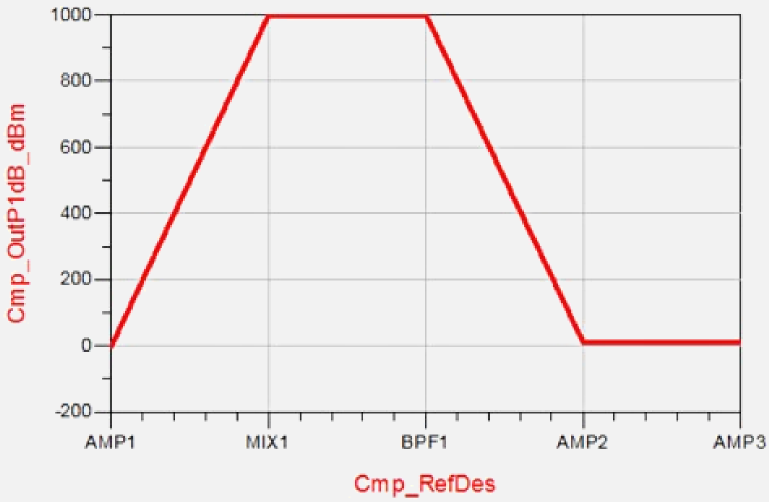

Figure 9. OutP1 spectrum

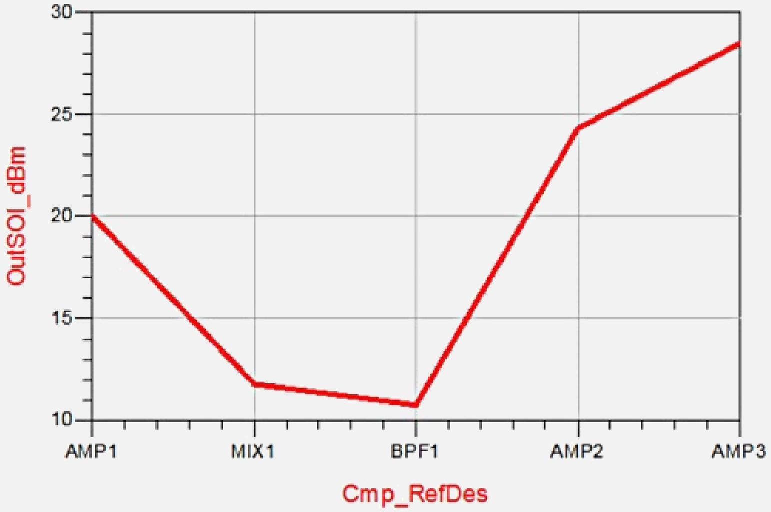

Figure 10. OutSOI spectrum

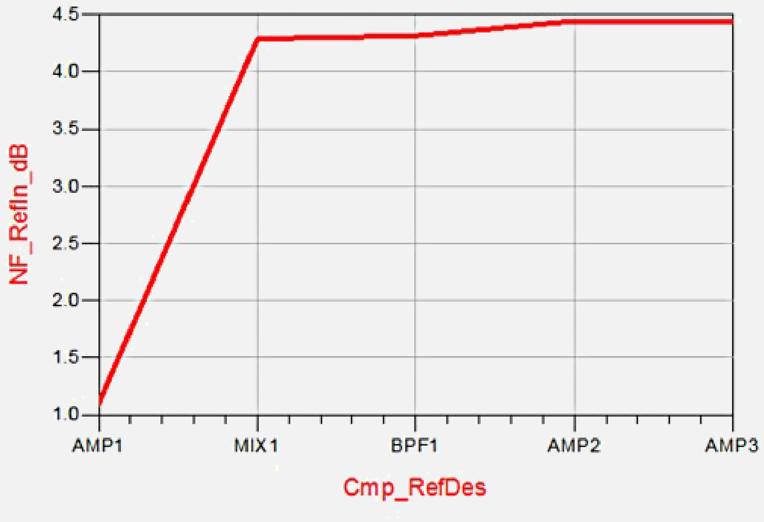

Figure 11. NF spectrum

Case Study 5: Exporting RF Budget Analysis Results to Excel

The PathWave ADS budget controller can export the budget simulation results to Microsoft Excel. To use this feature, double click Budget Controller and select the options as shown in Figure 12. You will need to find the installation path for Microsoft Excel (specially, Excel.exe). By default, the path is C:\Program Files (x86)\Microsoft Office\root*OfficeVersion*\Excel.exe.

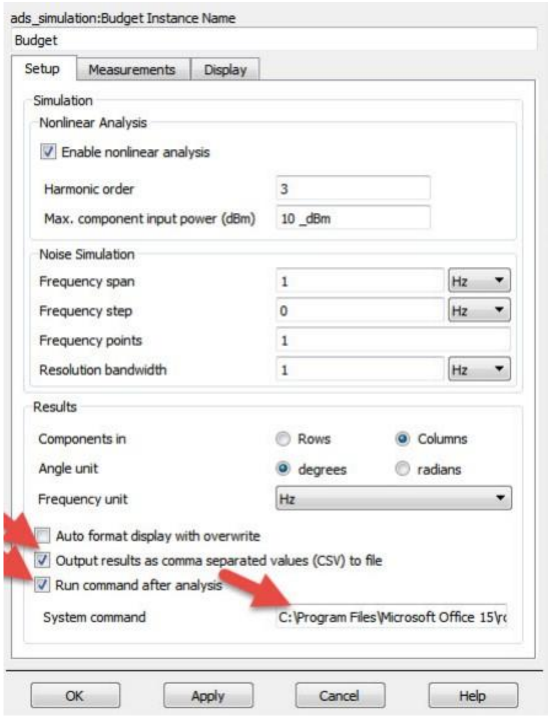

Figure 12. Exporting to Excel

Click the **Simulate** icon. Excel will open with the budget simulation results. Scroll down the Excel sheet to see the budget simulation results, as shown in Figure 13.

42	Meas_Name	AMP1	MIX1	BPF1	AMP2	AMP3
43	Cmp_NF_dB	1.131617	8.036397	1.021553	3.021827	3.021827
44	Cmp_OutP1dB_dBm	-0.63804	1000	1000	9.366897	9.366965
45	NF_RefIn_dB	1.1	4.285689	4.312709	4.441894	4.443175
46	OutTOI_dBm	10	1.755889	0.729914	17.30733	19.91895
47	OutSOI_dBm	20	11.75589	10.72991	24.30329	28.46084

Figure 13. Excel data

Conclusion

Congratulations! You have completed RF System Design. Check out more examples: www.keysight.com/find/eesof-ads-rfmw-examples.