SPECTRE18.1: Crystal Oscillator Simulation Using SpectreRF

Rapid Adoption Kit (RAK)

Product Version: IC6.1.8(IC6.1.8-64b.83), SPECTRE 18.1(18.1.0.169.isr2)
January 2019

Copyright Statement

Contents

Purpose

Crystal Oscillator is one of the class of oscillators which brings its own challenges to simulate. One of the key challenges it brings to simulate is a long startup time.

The purpose of this document is to make suggestions regarding the best practices for simulating Crystal Oscillators. A database composed of the circuits needed to reproduce these results is provided separately.

This RAK demonstrates a method to do Crystal Oscillator simulation using SpectreRF Analyses. Initially, a transient analysis is done and then hb/hbnoise analysis is used to determine the oscillation frequency and the phase noise of the oscillator.

Audience

This document is intended for Analog/RF engineers involved in designing Crystal Oscillators.

Terms

RF	Radio Frequency
RAK	Rapid Adoption Kit
HB	Harmonic Balance
PDK	Process Design Kit
ADE	Analog Design Environment

Introduction

The basic strategy is to simulate the Crystal Oscillator using harmonic balance (hb) analysis followed by hbnoise analysis to measure the PM (Phase-Modulated) component of the oscillator phase noise.

First, an estimate of the number of harmonics that will be needed for the hb analysis is determined. This is done using the transient analysis, and it uses a simulation capability introduced from MMSIM 16.1_ISR9. This capability allows estimating the amplitude of the oscillations at the start of the transient analysis. In this transient run, the largest slew rate of the signals in the circuit is determined. From this information, 0 to 100% risetime is determined, and an estimate of the number of harmonics needed is obtained by dividing the period from the risetime.

Once this estimate is made, a harmonic balance (hb) simulation is set up followed by an hbnoise analysis to measure the phase noise of the oscillator.

After that, the Q of the resonator is varied to see what happens to the phase noise produced by the system. The Q is varied by changing the inductor and capacitor values in the resonator while maintaining the resonant frequency.

Setup and Files Required To Perform Simulation

There is a single zipped tar file `xtal_example_database.tar.gz` that is composed of an IC61 library, a `cds.lib` file, and a directory named as `share` in which you need to download the gpdk180 PDK.

1. Extract this zipped tar file in a writable directory using:

```
unix%> tar zxvf  xtal_example_database.tar.gz

unix%> cd xtal_example_database
```

2. Then, step into the `share` directory.

```
unix%> cd share
```

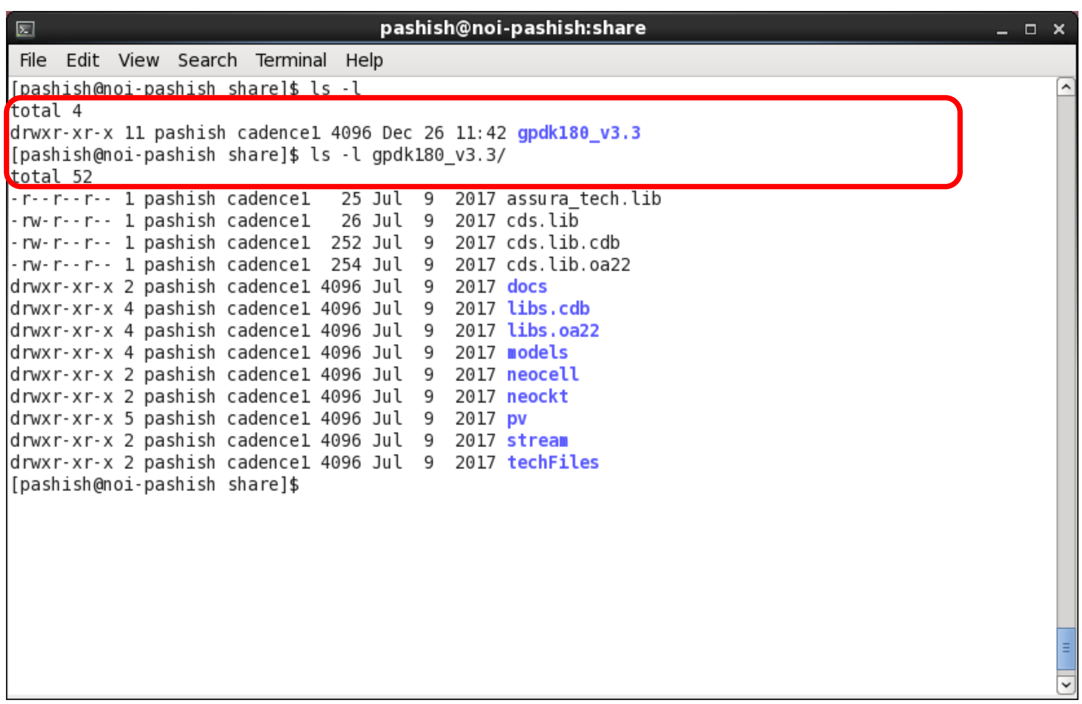
3. You can download gpdk180_v3.3 in the `share` directory from:

   http://support.cadence.com/wps/mypoc/cos?uri=deeplinkmin:DocumentViewer;src=wp;q=ProductInformation/GPDK/GPDK.htm

This library is assumed to be located in the `share` directory (as mentioned above).

Below is what you should see inside the `share` directory.

Figure 1 – Linux Terminal View showing the details of gpdk180_v3.3b PDK



4. Create a link to gpdk180_v3.3 by gpdk180 using the following command:

```
unix%> ln -s gpdk180_v3.3 gpdk180
```

5. Move one level up from the `share` directory to the `xtal_example_database` directory.

```
unix%> cd ..
```

6. Then, verify that you have a simulator version that supports estimation of the oscillator amplitude by typing the following command:

```
unix%> spectre –h tran
```

7. Look at the output and verify that the **`linearic`** and **`oscfreq`** options are present in the **Initial-condition parameters** section as shown below. These options will be used later to estimate the amplitude of oscillations at the start of the transient analysis. If your simulator release does not have these options, you will need to start the oscillator from scratch and run the simulation long enough to approach the steady-state amplitude. Alternatively, you might choose to start with an estimate of how many harmonics might be required for the harmonic balance simulation.

Figure 2 - spectre -h tran - Transient Analysis Parameters

```
pashish@noi-pashish:xtal_example_database
File  Edit  View  Search  Terminal  Help

*****************
Transient Analysis
*****************

This analysis computes the transient response of a circuit over the interval from `start' to
`stop'. The initial condition is taken to be the DC steady-state solution, if not otherwise
given.


Synopsis:
Name tran <parameter=value> ...


==========
Parameters
==========

Simulation interval parameters
1       stop (s)          Stop time.
2       tpoints=[...] s   Multiple of pairs<pstep, stop>.
3       start=0 s         Start time.
4       pstep (s)         print step.
5       outputstart=start s
                          Output is saved only after this time is reached.
6       autostop=no       If yes, the analysis is terminated when all event-type measurement
                          expressions have been evaluated. Event-type expressions use
                          thresholding, event, or delay type functions. If the value is
                          spice, autostop is consistent with spice simulator. Possible values
                          are no, yes, and spice.

Time-step parameters
7       maxstep (s)       Maximum time step. The default is derived from `errpreset'.
8       step=0.001*(stop-start) s
                          Minimum time step used by the simulator solely to maintain the
                          aesthetics of the computed waveforms.
9       minstep (s)       Minimum time step. If specified, the error tolerance requirements
                          may be ignored when step size is less than minstep.
10      istep=0.001*(stop-start) s
                          When step size is greater than istep, the local truncation error
                          checking is enabled for algebraic nodes.

Initial-condition parameters
11      ic=all            The value to be used to set the initial condition. Possible values
                          are dc, node, dev, and all.
12      skipdc=no         If set to yes, there is no DC analysis for transient. Possible
                          values are no, yes, useprevic, waveless, rampup, autodc, sigrampup,
                          and dcrampup.
13      rampupratio=0.1   The rampup ratio for skipdc=rampup and sigrampup.
14      rampuptime (s)    The rampup time for skipdc=rampup. The default value is set to
                          `rampupratio'*`stop'.
15      readic            File that contains initial condition.
16      useprevic=no      If set to yes or ns, use the converged initial condition from
                          previous analysis as ic or ns. Possible values are no, yes, and
                          ns.
17      linearic=no       Enable linear IC method to calculate initial conditions
                          automatically from a type of stability analysis in the range
                          [0.5*oscfreq, 1.5*oscfreq]. Overrides user-defined initial
                          conditions if instability is detected. Possible values are no and
                          yes.
18      oscfreq=0.0       Estimation of the oscillation frequency when linear IC method is
                          enabled.
```

Simulating Crystal Oscillator

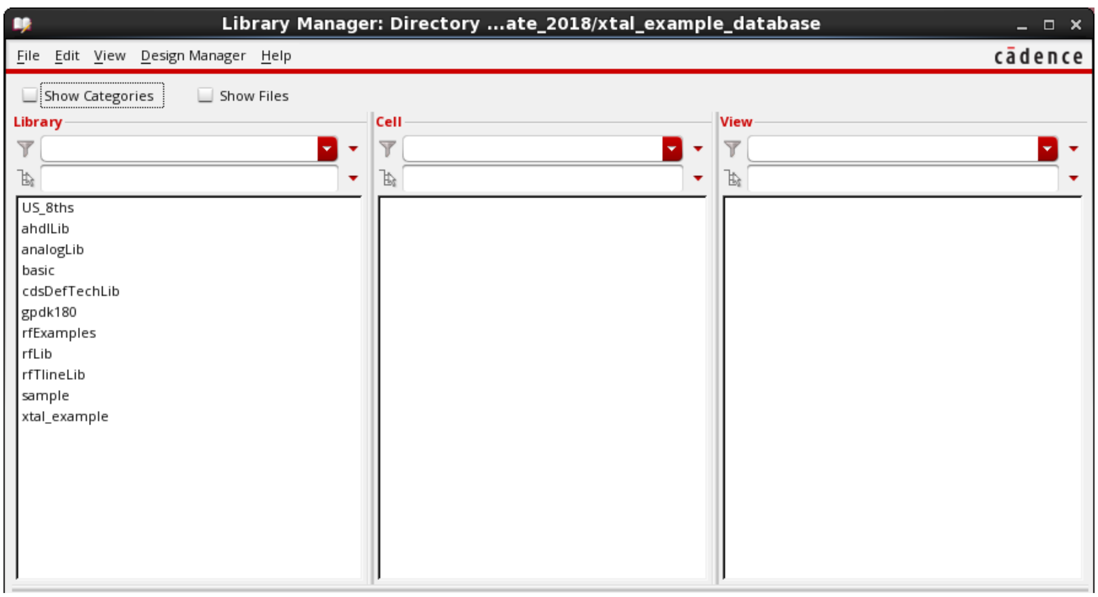
Estimating the Number of Harmonics Required Using Transient Analysis

8. Launch Virtuoso.

```
unix%> virtuoso &
```

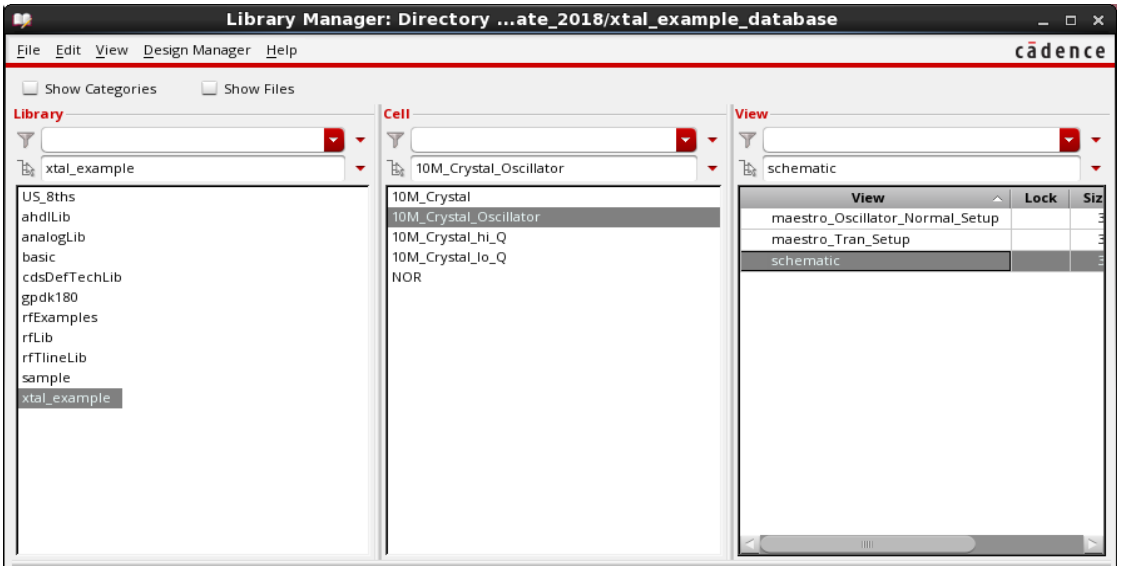
9. Open the Library Manager by going to **Tools > Library Manager** in the CIW window.

Figure 3 – Library Manager window



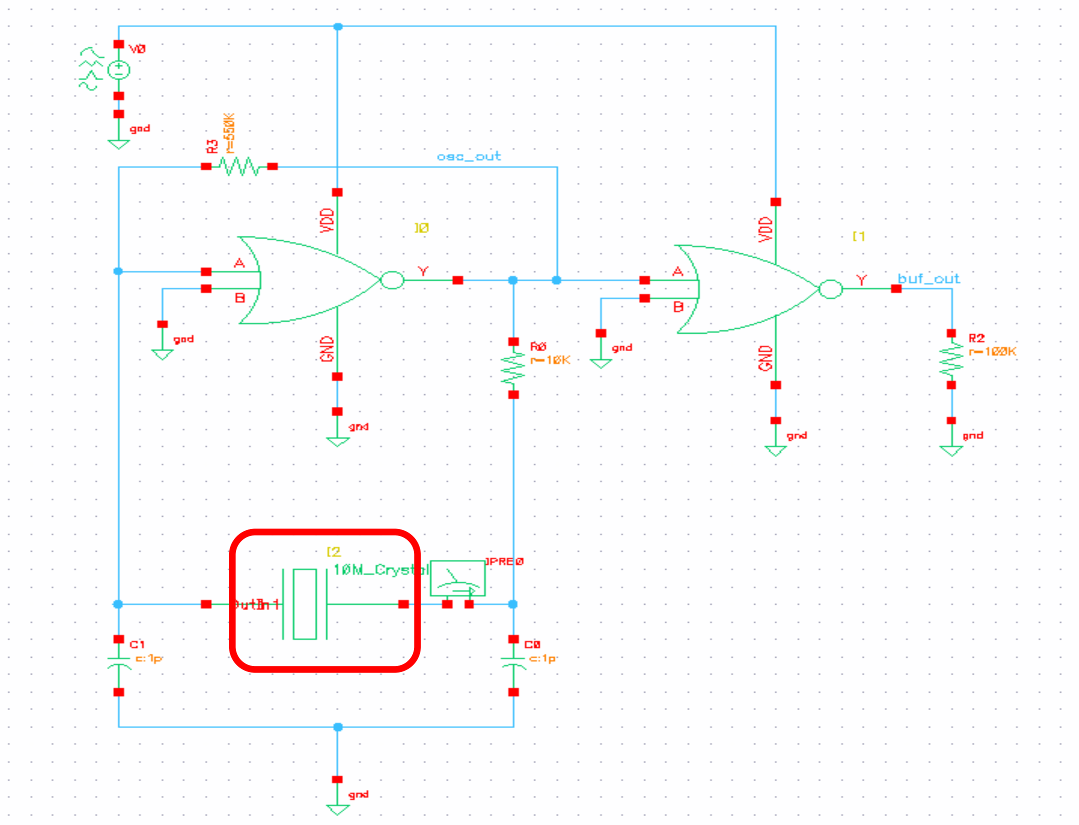
10. From the Library Manager, open the schematic view by going to **xtal_example > 10M_Crystal_Oscillator > schematic** as shown in Fig 4.

Figure 4 - Opening schematic view of 10M_Crystal_Oscillator cell from xtal_example library using Library Manager



The 10M_Crystal_Oscillator circuit schematic appears as shown in Fig 5.

Figure 5 - Schematic of 10M_Crystal_Oscillator cell



The background of all the schematics in this document have been changed to white. This is done so that the schematic will be visible if this document is printed.

You can also refer to Article [How to change display background color of Schematic and Layout windows from black to white] to get more details on how to change the schematic background to white.

11. Select the crystal symbol at the bottom of the schematic, click RMB, and descend-read into the underlying schematic in a new tab as shown in Fig 6. The schematic which appears, as shown in Fig 7, is representative of a commercially available 10MHz crystal, and is not representative of any specific manufacturer.

Figure 6 - Selecting the crystal symbol in the schematic of 10M_Crystal_Oscillator cell

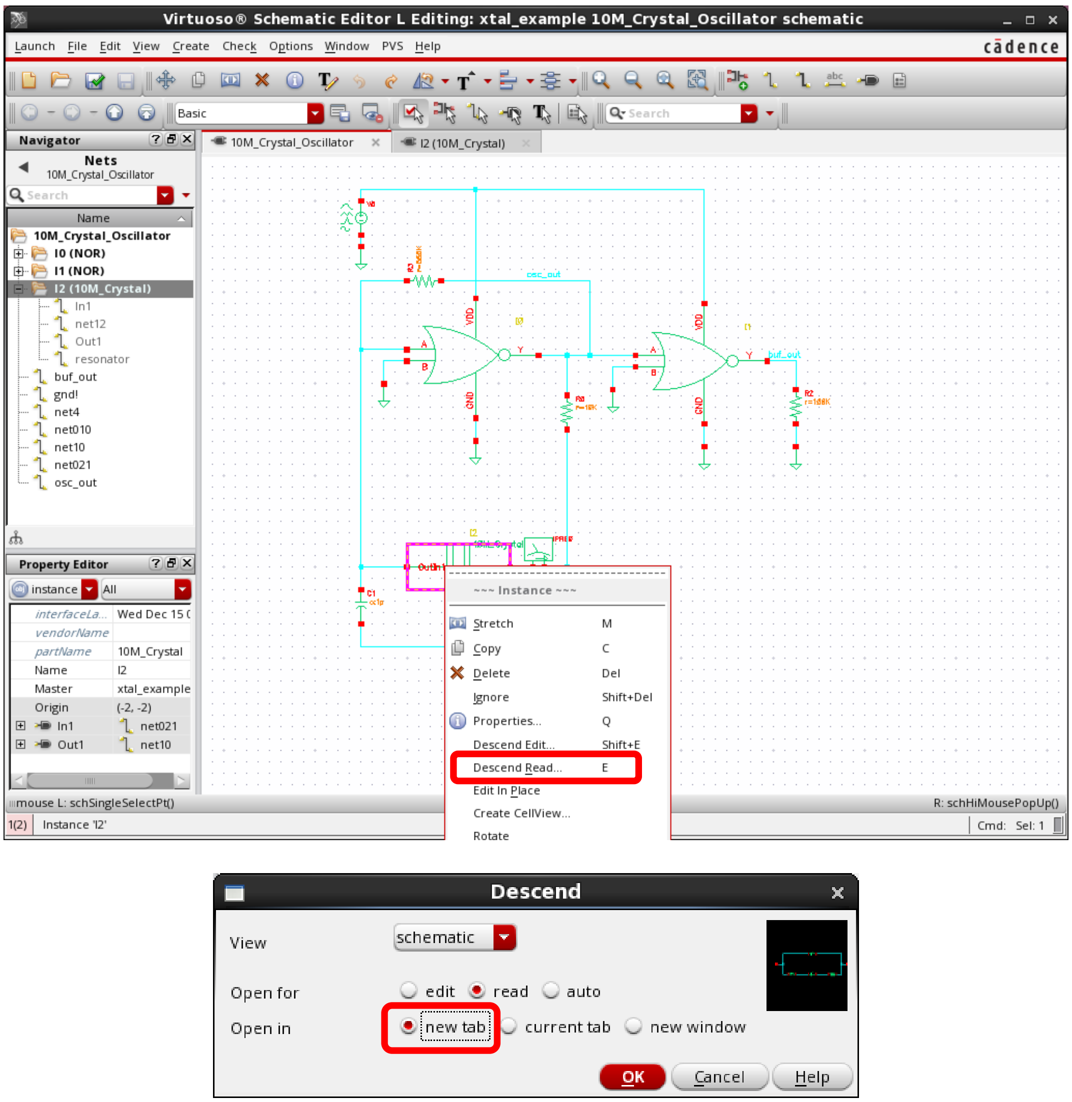
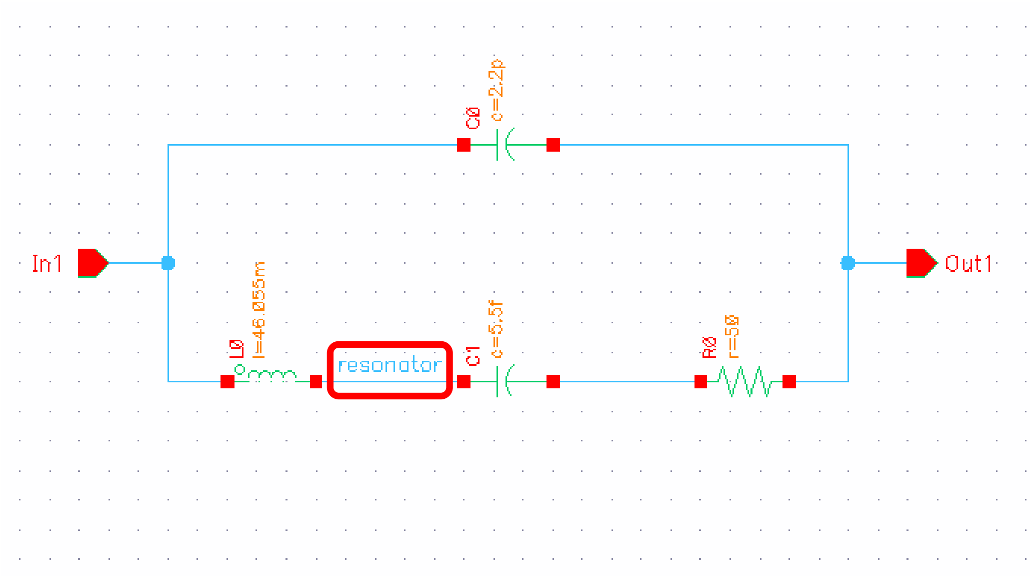
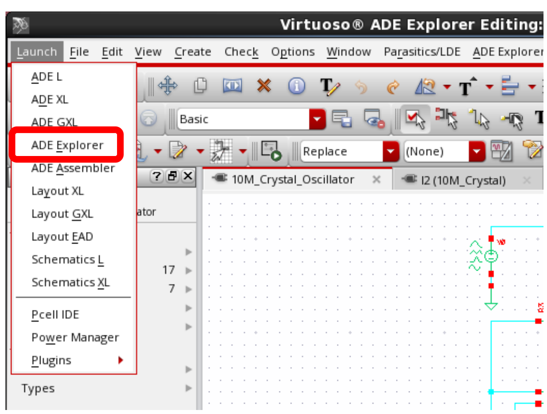


Figure 7 – 10M Crystal Oscillator schematic (representative only)



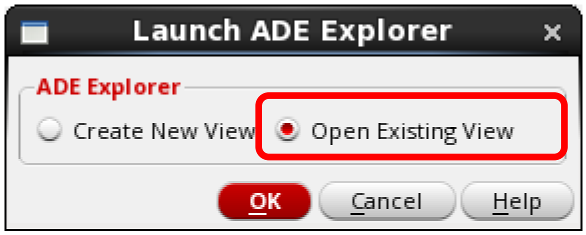
12. Click the **10M_Crystal_Oscillator** tab to get back to the top-level schematic.
13. Go to **Launch > ADE Explorer** in the schematic window.

Figure 8- Open ADE Explorer from the schematic window



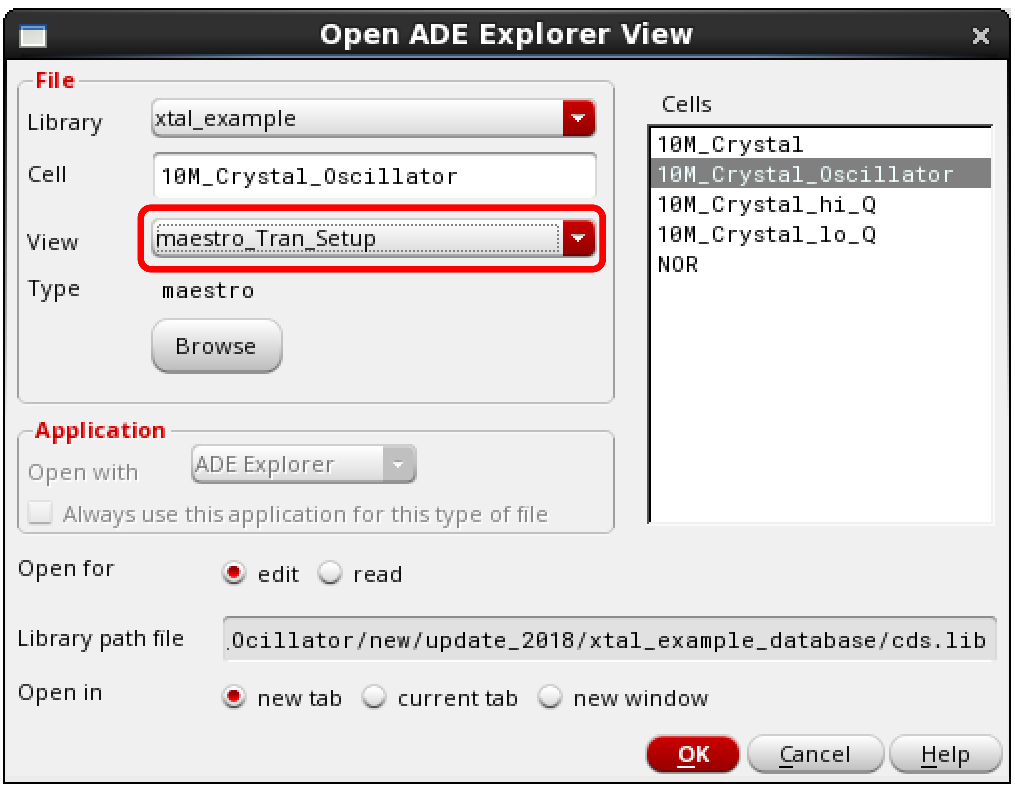
14. In the **Launch ADE Explorer** dialog box, select "**Open Existing View**" if not already selected and click **OK**.

Figure 9 - Open an existing maestro view in ADE Explorer



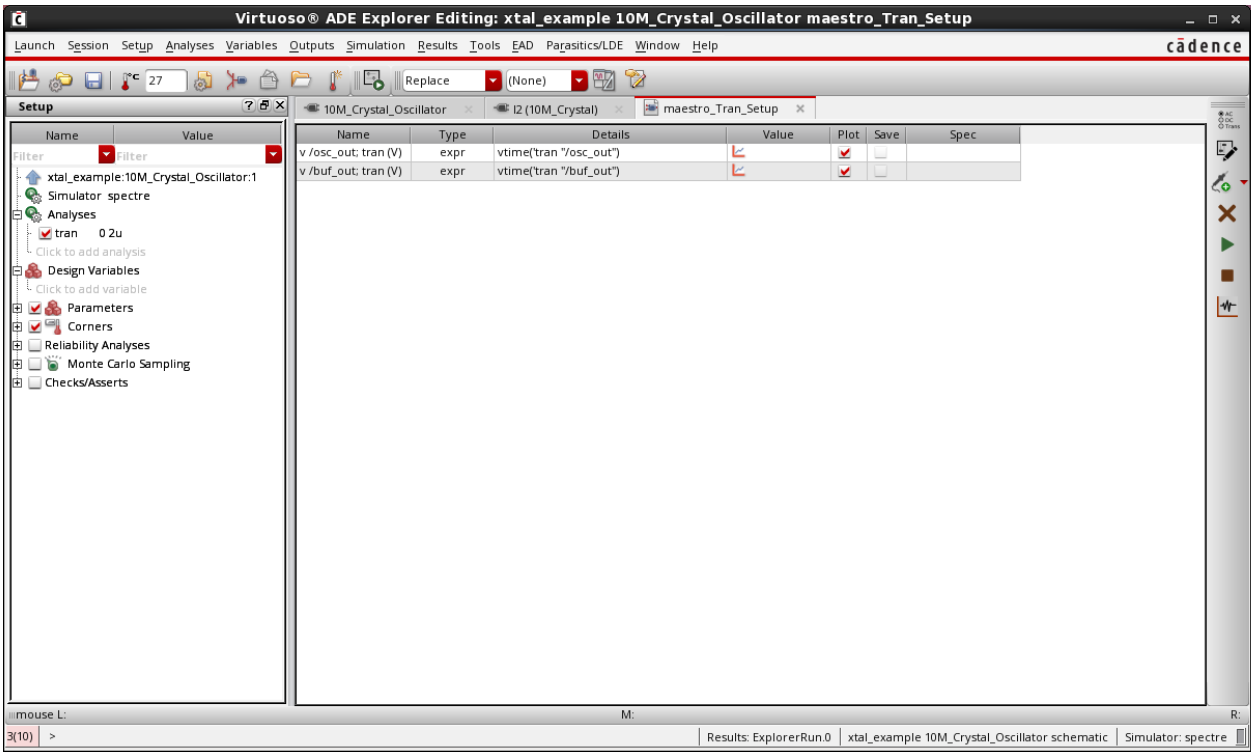
15. In the **Open ADE Explorer View** window, select the **maestro_Tran_Setup** view from the **View** drop-down list and click **OK** to close the form.

Figure 10 - Open maestro_Tran_Setup view from the Open ADE Explorer View form



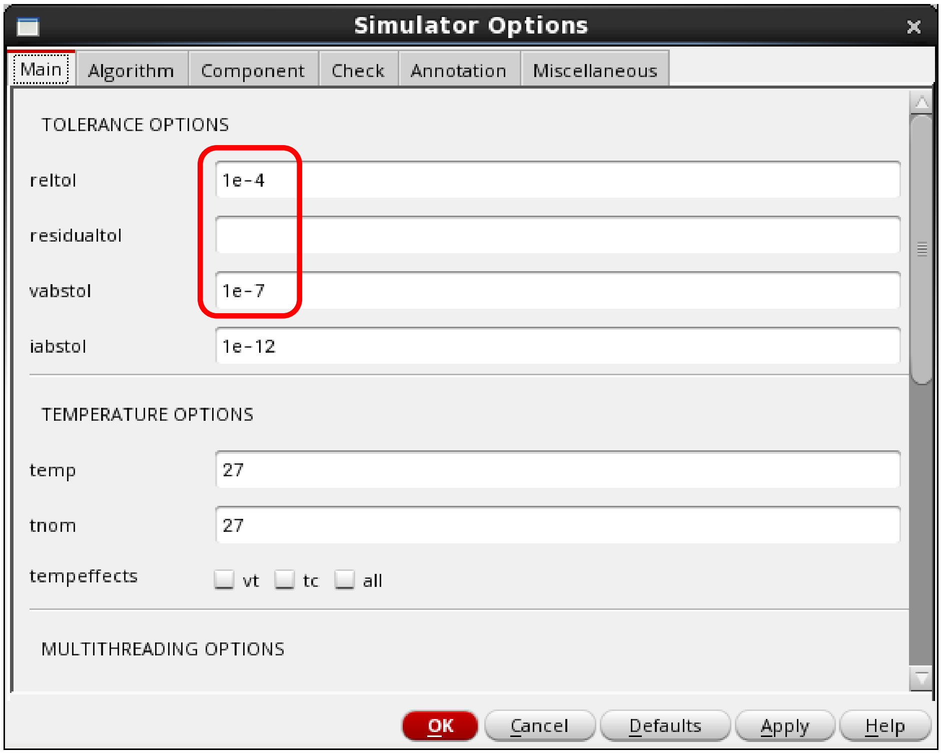
16. An ADE Explorer window with the **maestro_Tran_Setup** view gets opened in a new tab.

Figure 11 - Opening maestro_Tran_Setup view in ADE Explorer



17. In the ADE Explorer window, open the **Simulator Options** form (**Simulation > Options > Analog**) and note that **reltol** and **vabstol** have been changed from the default values. This is suggested to make the simulation more accurate than the default settings. Once reviewed, click **OK** to close the form.

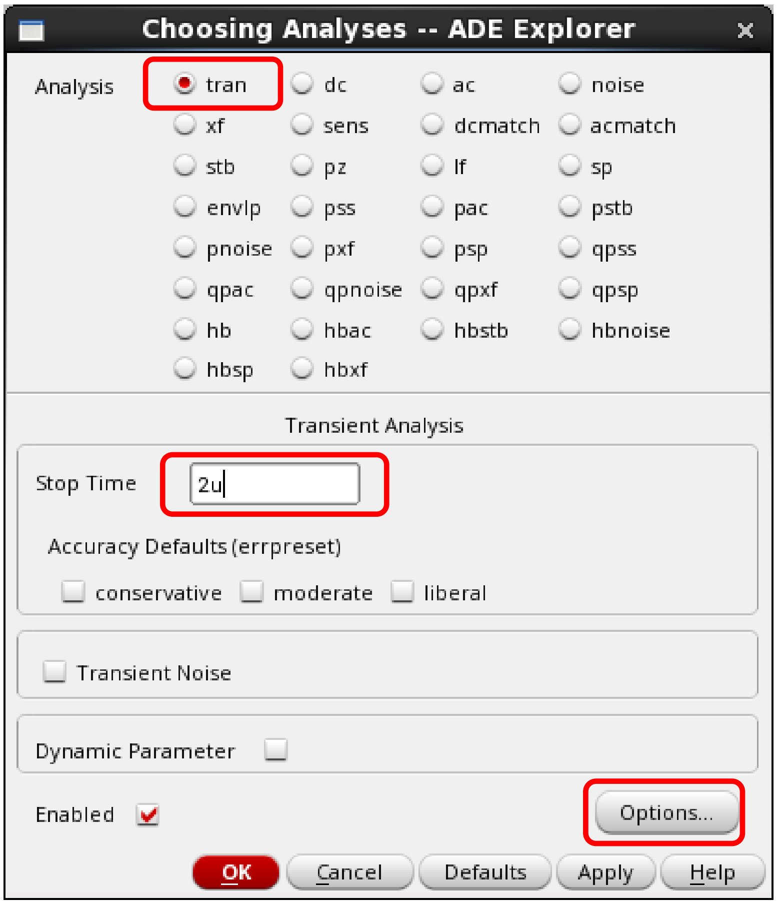
Figure 12 – Transient analysis - Simulator Options form



18. In the ADE Explorer window, double-click on the **tran** analysis line in the **Analyses** section of the Setup Assistant. This will open the **Choosing Analyses** form with **tran** analysis settings.

    The frequency of the example crystal is 10MHz. The stop time is set to about 20 periods of the oscillator frequency.

Figure 13 - Choosing Analyses form - Setting up transient analysis



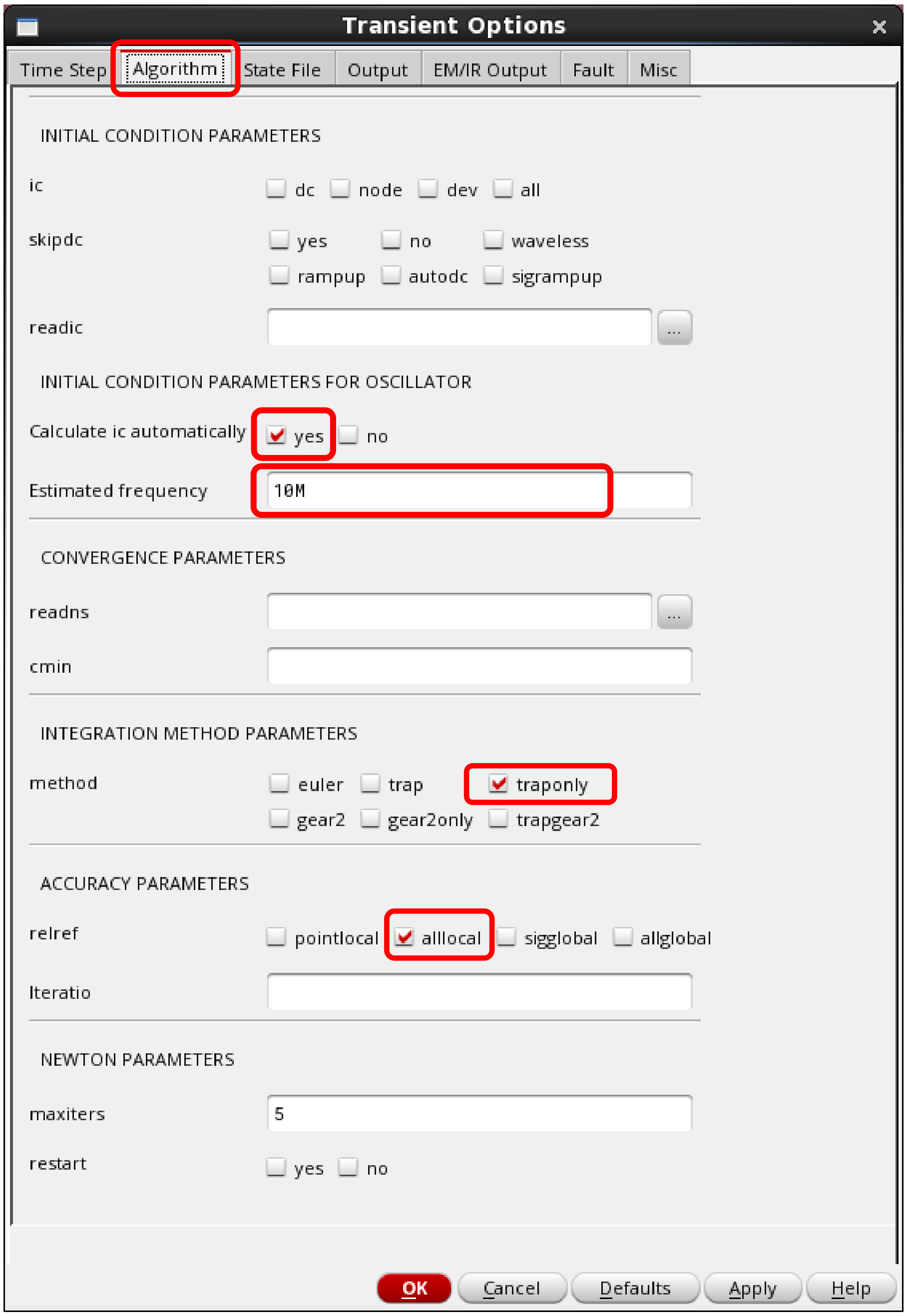
19. Click the **Options** button at the bottom of the form.

20. In the **Transient Options** form which gets opened, click the **Algorithm** tab. The window might need to be resized.

You will see that the **Calculate ic automatically** option is set to **yes**. The oscillator frequency for this example is 10MHz; so, the **Estimated frequency** option is set to **10MHz**.

You will also see that the **method** is set to **traponly** and the **relref** is set to **alllocal**.

Figure 14 - Transient analysis – Transient Options form – Setting method and relref parameters



Traponly is suggested for the integration method because this method does not numerically emphasize or numerically damp the oscillations in the circuit. Alllocal is suggested because there are large amplitude signals in the High Q resonator.

Note that in the transient analysis, each timepoint is an iterated solution. The iterations stop when the convergence criteria are met. The convergence criteria for voltages is as follows:

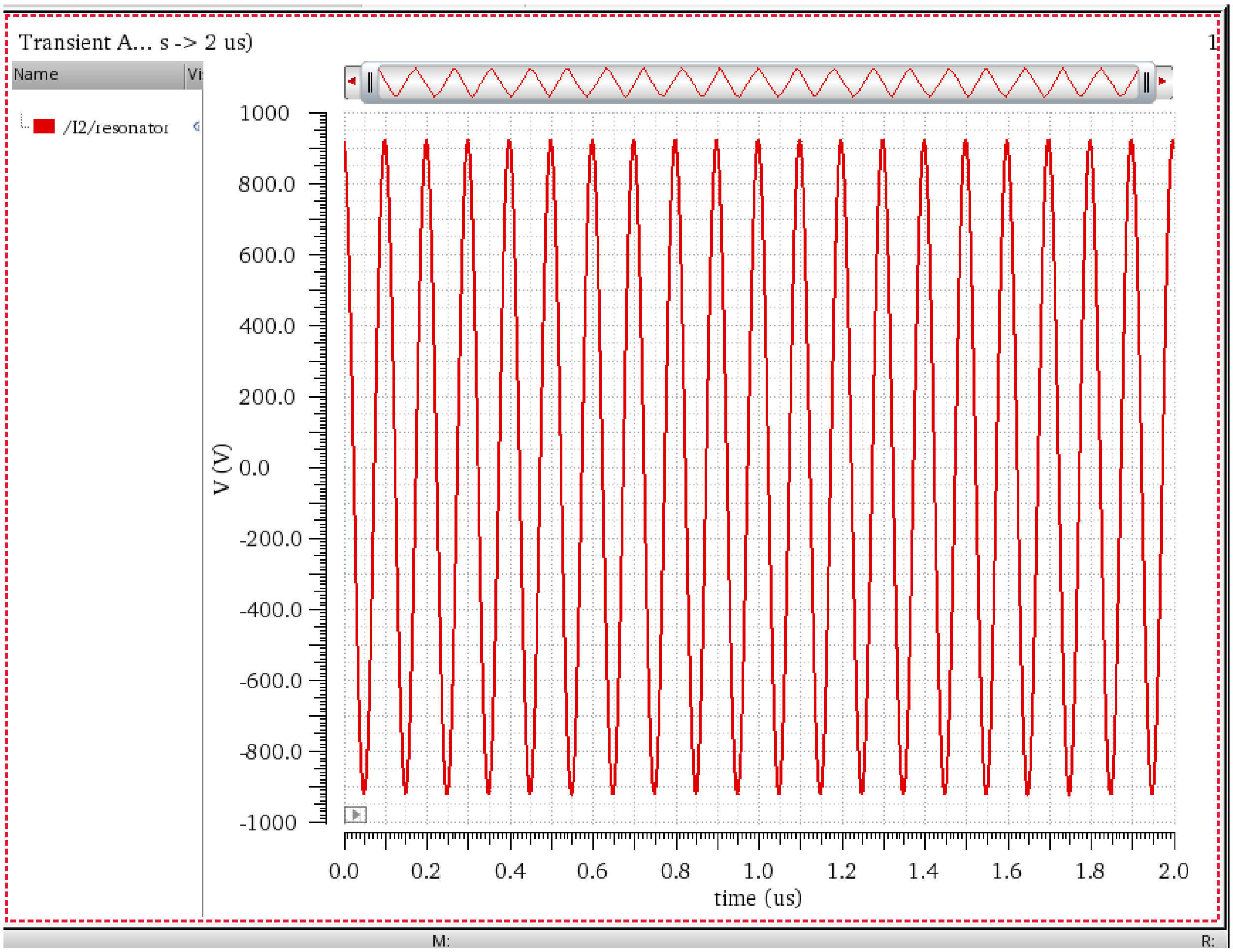
- reltol * largest voltage on the net up to the current time + vabstol (when alllocal is set)

- reltol * largest voltage anywhere in the circuit up to the current time + vabstol (when the default sigglobal selection is used)

Because sigglobal takes the largest voltage in the circuit, the accuracy on the low-voltage nets would be seriously degraded because of the large amplitude signals in the resonator.

For reference only, shown below is a plot of the net called **resonator** in the crystal motional equivalent circuit. Normal voltages at this net are in the 500V to 2KV peak range. You can verify this after the transient simulation runs.

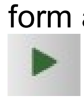
Figure 15 - Transient analysis response of resonator net signal



The waveform starts at a peak in the resonator. This is due to the estimate of the oscillating condition at time zero.

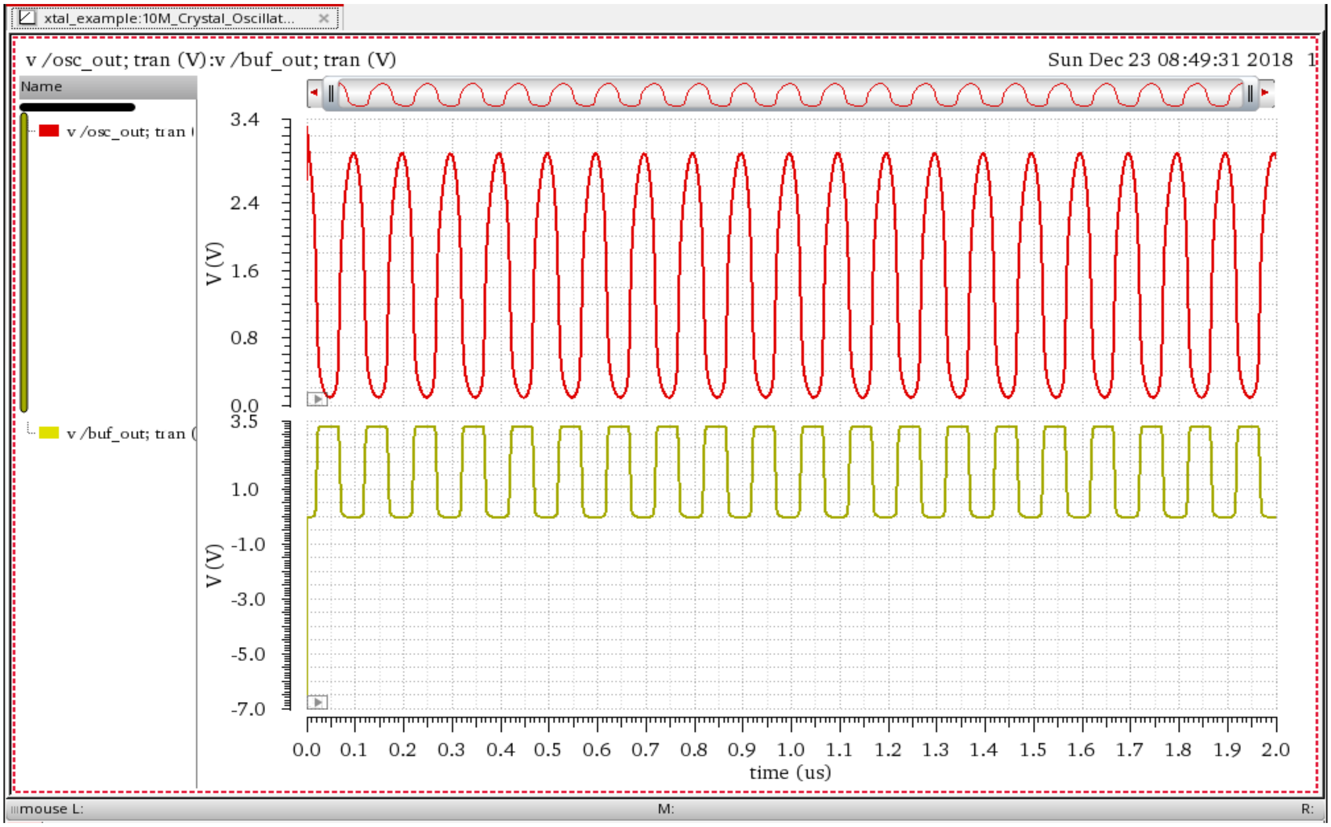
Note: The background of the plot window is changed to white by adding the following line to the `.cdsinit` file:

```
envSetVal("viva.graphFrame" "background" 'string "white")
```

21. Click **OK** to close the **Transient Options** form and the **Choosing Analyses** form. Then, run the simulation by clicking  in ADE Explorer. At the end of the simulation, the output signals of the two gates should plot.

22. After splitting the strips, the plot appears as shown in Fig 16.

Figure 16 – Plot of osc_out net voltage and buf_out net voltage signal after transient analysis

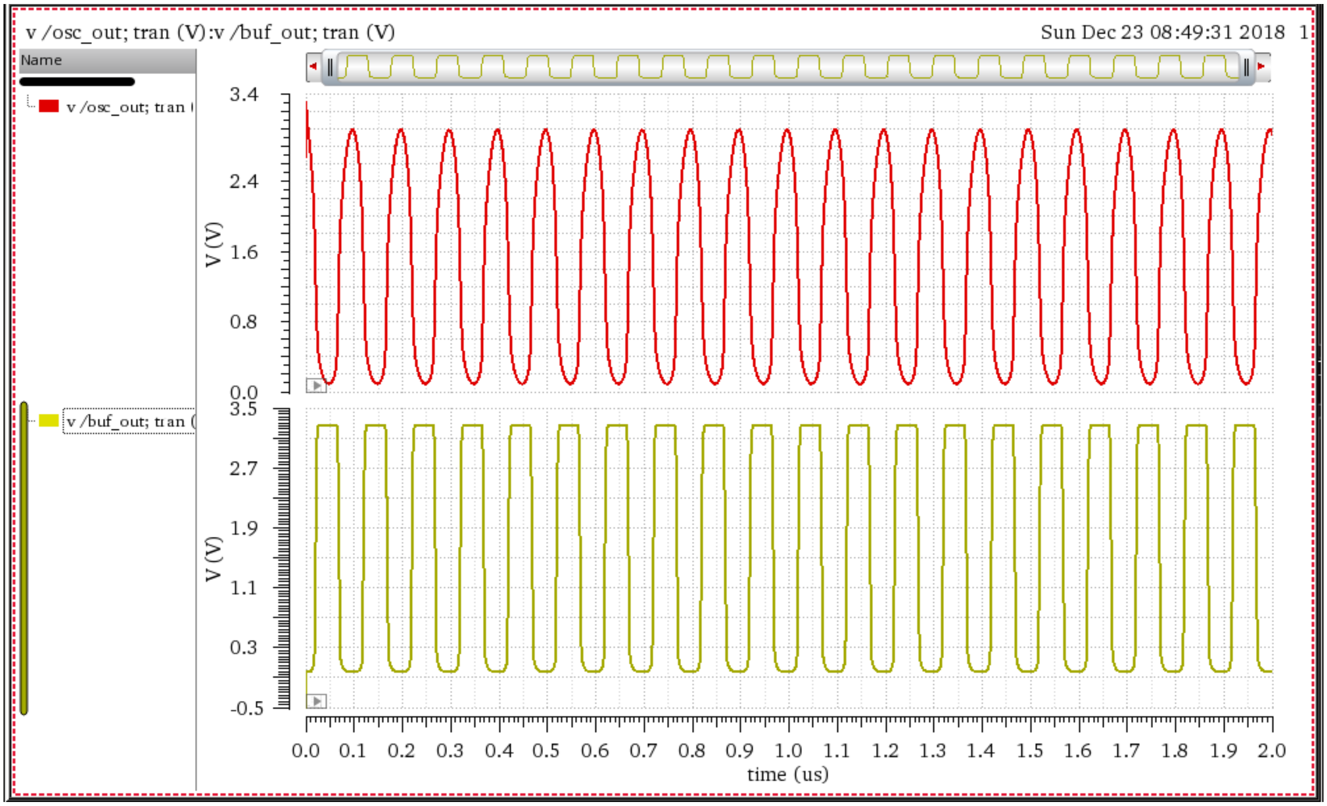


23. Change the scale of y-axis of `buf_out` signal to **-0.5V – 3.5V**.

Figure 17 - Changing Y-axis limits

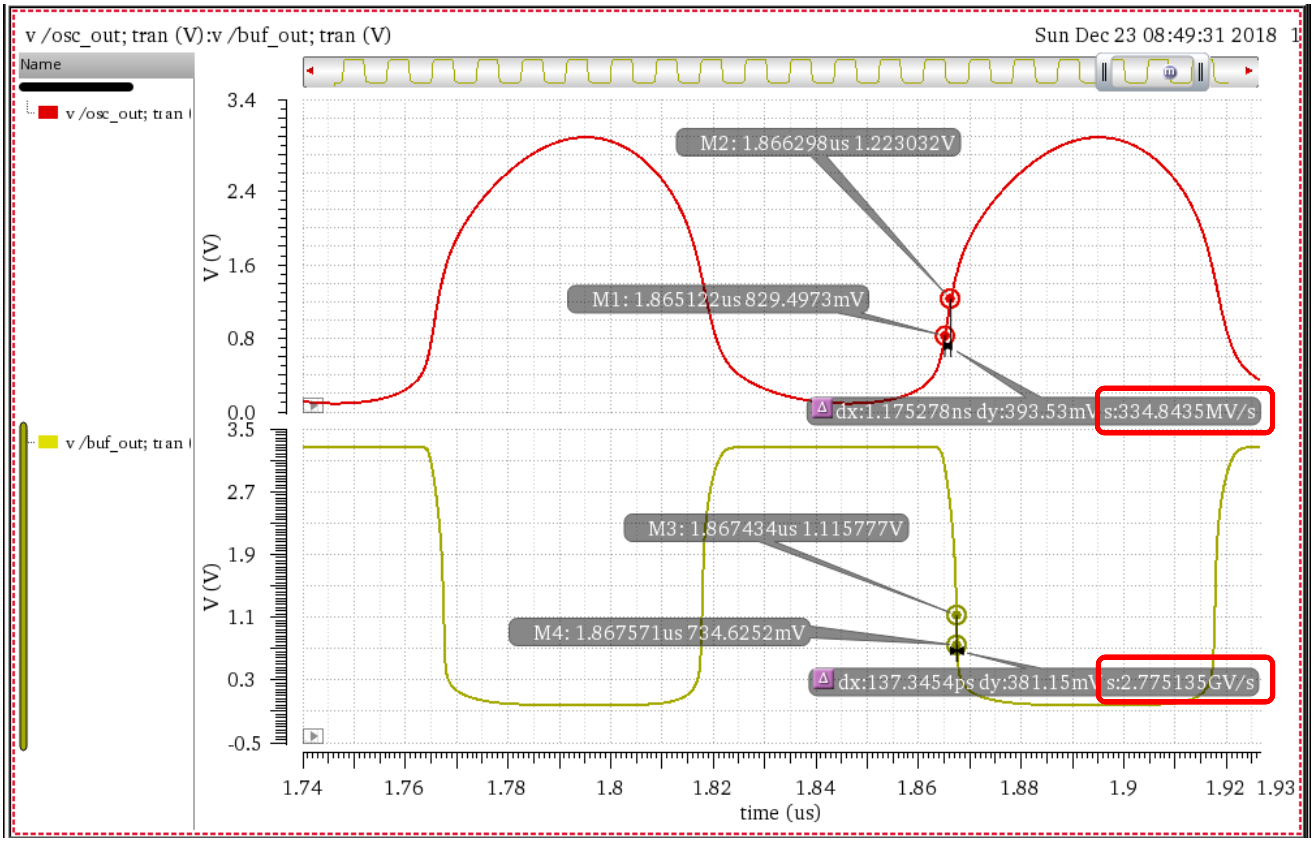
The plot should look as shown in Fig 18.

Figure 18 - Plot of osc_out net voltage and buf_out net voltage signal after manual scaling of y-axis of buf_out signal plot



24. Now, zoom in near the end and determine the largest slew rates for both signals.

Figure 19 – Determining largest SlewRate of osc_out and buf_out net voltage signals



In the above waveforms, both the rising and falling edges were analyzed. The edge with the largest slew rate is shown. The slew rate is displayed as the bottom number in the delta readouts, as highlighted above.

25. To position a marker, point your cursor near the location where you want the marker and type **m**. To get a delta marker, select a marker on the screen, then move to the next place you want to measure, and type **Shift+d**. Both marker locations can be moved holding the left mouse button down over a marker and moving the cursor.

26. Determine the 0% to 100% rise/fall time for both signals. For the `osc_out` signal, it is 3.3/334.85e6, and for the `buf_out` signal, it is 3.3/2.78e9.

    (9.86e-9 and 1.19e-9, respectively)

27. Determine the estimated number of harmonics by dividing the period by the risetime.

    For the **osc_out** signal: 1e-7/9.86e-9 = 10.14. Round up to 11.

    For the **buf_out** signal: 1e-7/1.19e-9 = 84.

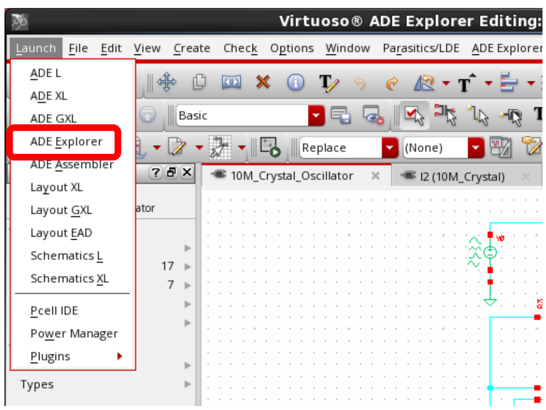
28. Close the ViVA XL Waveform window by going to **File > Close All Windows** in that window.

29. Close the ADE Explorer window by going to the **maestro_Tran_Setup** tab and clicking **Session > Quit** in that window. Click '**Yes**' when asked to save the ADE Explorer setup.

Setting Up Harmonic Balance Simulation

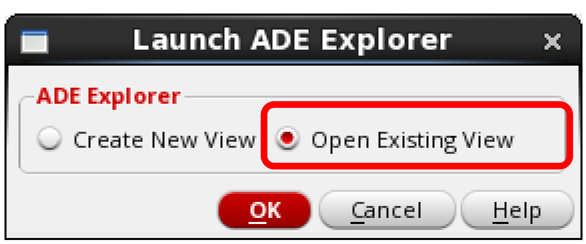
30. Go to the **10M_Crystal_Oscillator** schematic tab, and then click **Launch > ADE Explorer**.

Figure 20 - Open ADE Explorer from the schematic window



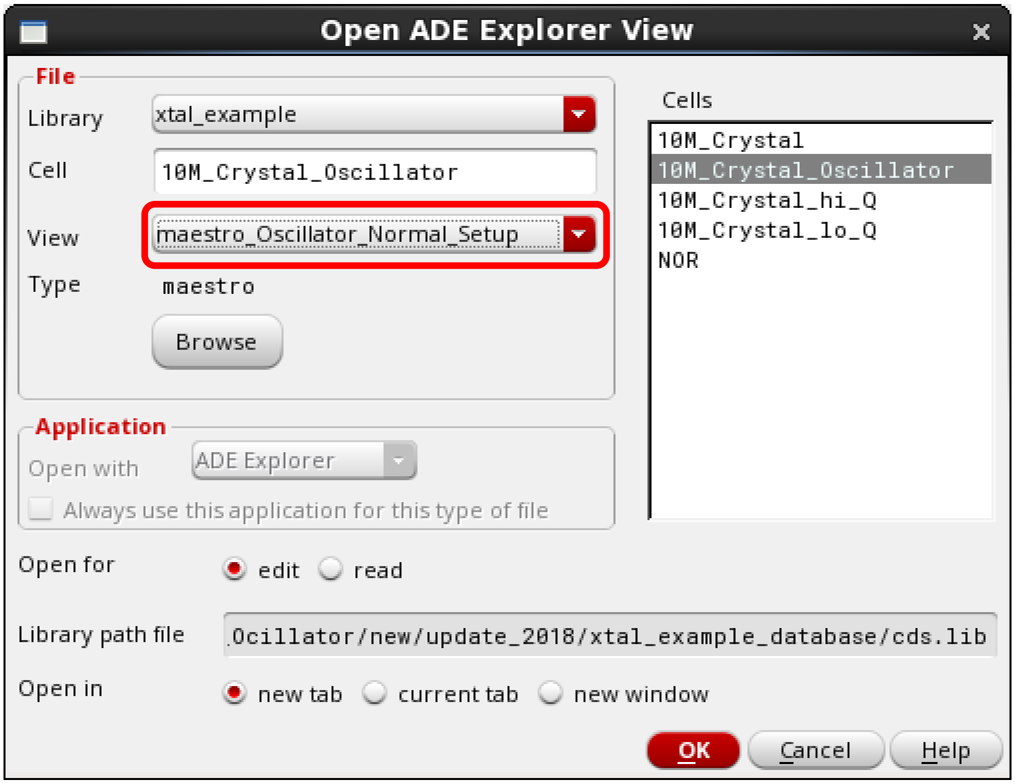
31. In the **Launch ADE Explorer** dialog box, select "**Open Existing View**" if not already selected and click **OK**.

Figure 21 - Open an existing maestro view in ADE Explorer



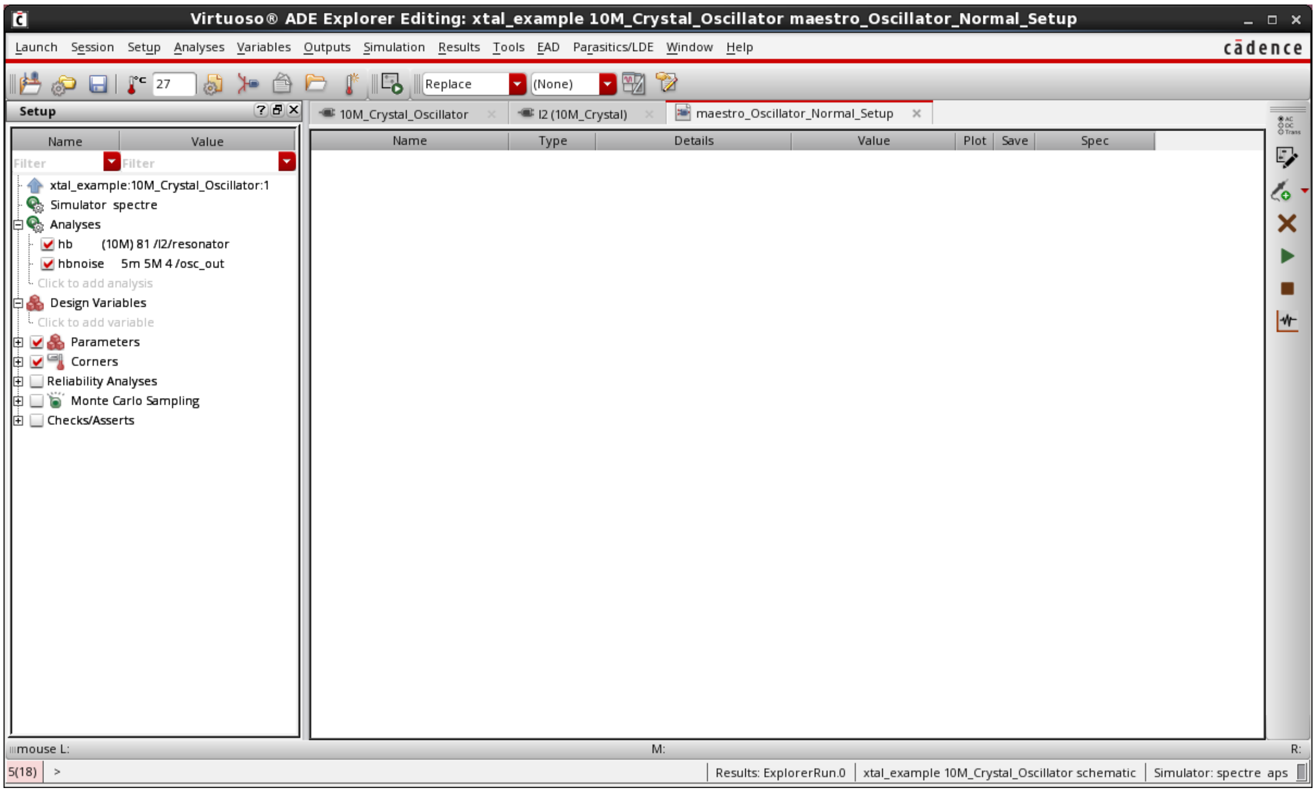
32. In the **Open ADE Explorer View** window, select the **maestro_Oscillator_Normal_Setup** view and click **OK** to close the form.

Figure 22 - Open maestro_Oscillator_Normal_Setup view from the Open ADE Explorer View form



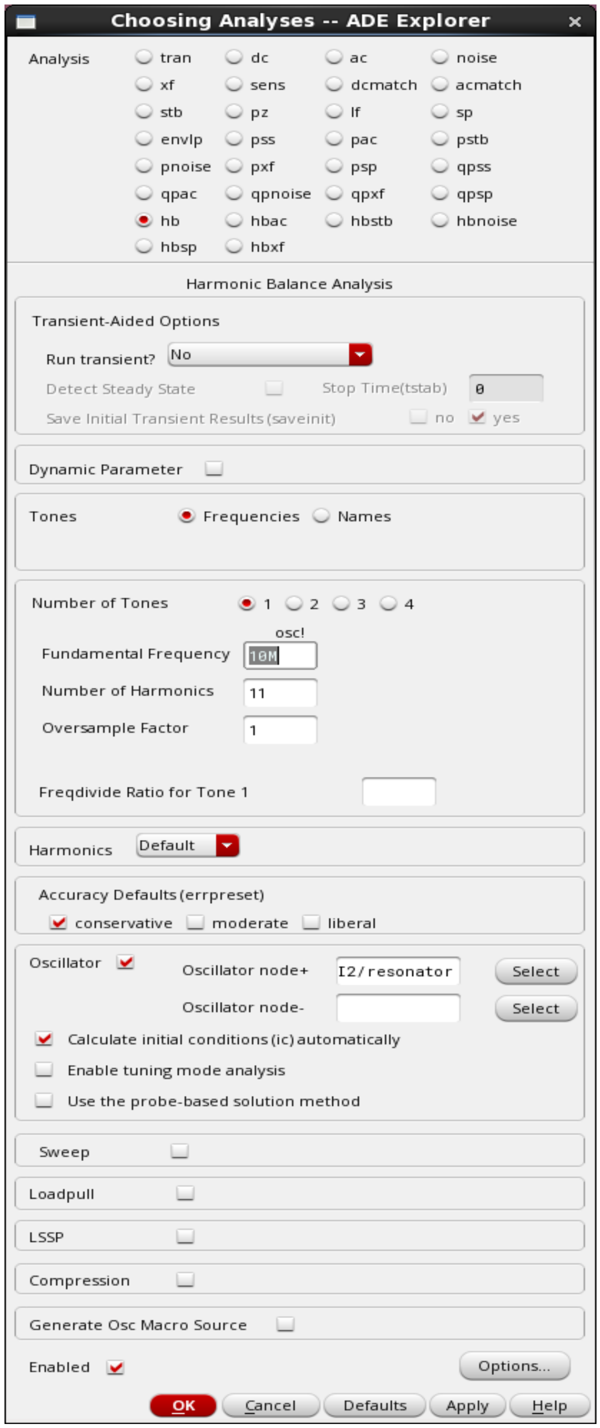
33. An ADE Explorer window with the **maestro_Oscillator_Normal_Setup** view gets opened in a new tab.

Figure 23 - maestro_Oscillator_Normal_Setup view in ADE Explorer in new tab



34. In the ADE Explorer window, double-click on the **hb** analysis line in the **Analyses** section of the Setup Assistant. This will open the **Choosing Analyses** form with **hb** analysis setup.

Figure 24 - Choosing Analyses form - Setting up of harmonic balance (hb) analysis



Analyzing the Harmonic Balance Analysis Settings

35. Analyze the settings of the hb analysis.

    a. **Run transient** is set to **No**. This causes hb to start without running a tstab transient analysis.

    b. **Fundamental Frequency** is set to the oscillator frequency.

    c. The **Number of Harmonics** field is set to **11**, which we calculated for the `osc_out` net. If the transient analysis from the preceding section is not available, start with an estimate of the number of harmonics that might be needed. The number of harmonics needed will be verified shortly.

    d. **Oversample Factor** is set to **1**. When the estimation procedure is used, oversample can usually be set to 1.

    e. Conservative accuracy is always recommended for any oscillator.

    f. The oscillator node is not critical. It should have a signal on that net.

    g. It is suggested that you select the **Calculate initial conditions (ic) automatically** option for any Crystal Oscillator. In earlier releases, this was the **Calculate oscillator initial conditions** option, which had two settings: **Default** and **Linear**. The **Linear** selection is the equivalent to the new setting.

       This option causes Spectre to run a variation of the stability analysis after running the DC solution. It estimates the amplitudes and phases on all the nets for the first harmonic. When **Run transient** is set to “**no**”, this estimate is used as the starting point of the harmonic balance analysis for the first harmonic and the DC solution is used for the DC harmonic. The harmonic balance algorithm goes on to solve for the specified number of harmonics of the nonlinear circuit. In many cases, this is a good enough starting point that allows the harmonic balance analysis to converge. Because this mode completely bypasses any transient analysis in the tstab interval, it runs quickly as well.

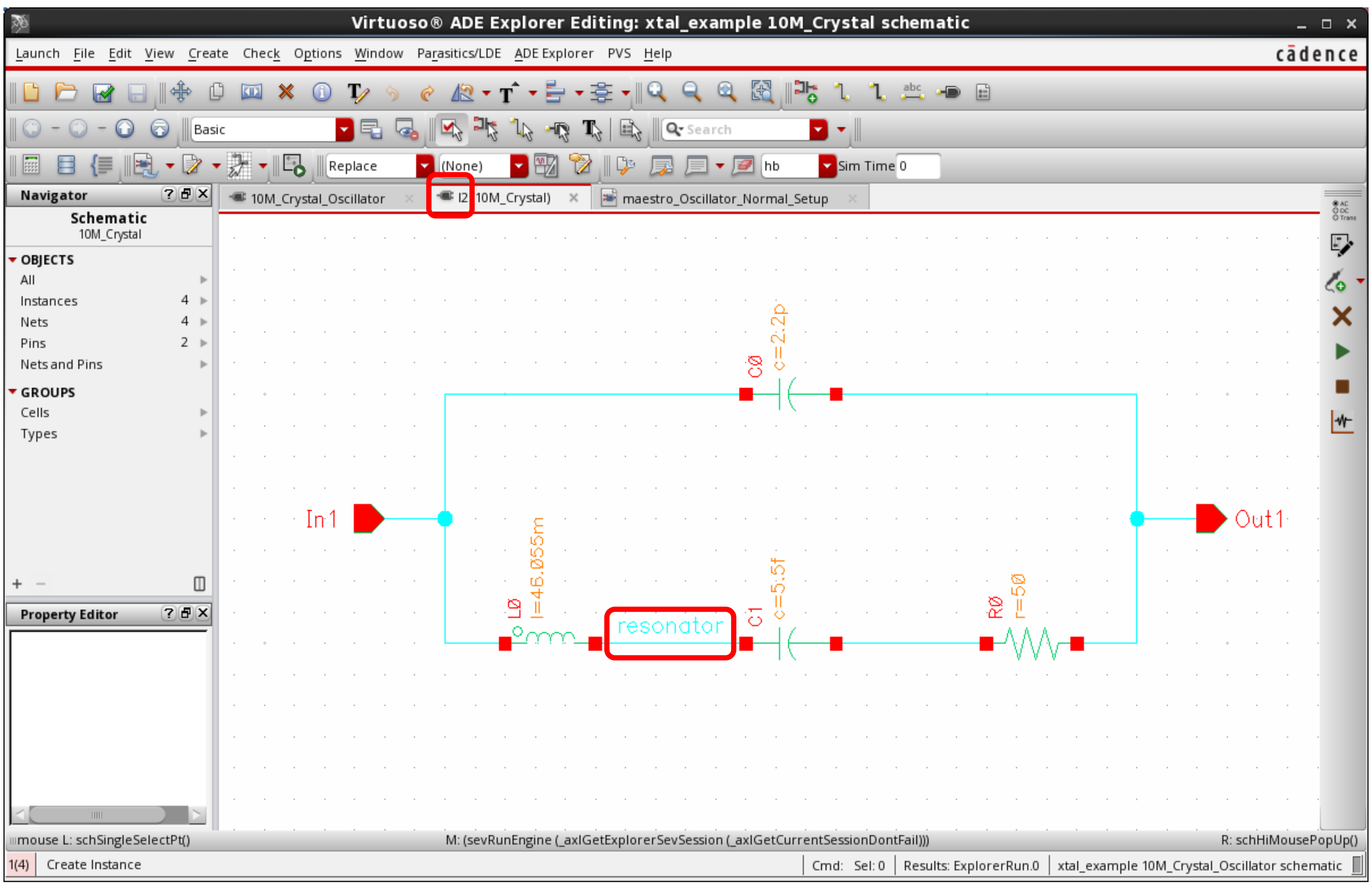
Setting Up Harmonic Balance Analysis Options

36. Click the **Options** button at the bottom of the hb **Choosing Analyses** form.

37. Click the **Convergence** tab.

38. Although Spectre has an algorithm for picking the pinnode automatically, for Crystal Oscillators, it is usually better to specify the pinnode yourself. The best choice is the net that is the junction between the inductor and the capacitor of the motional equivalent circuit as shown below.

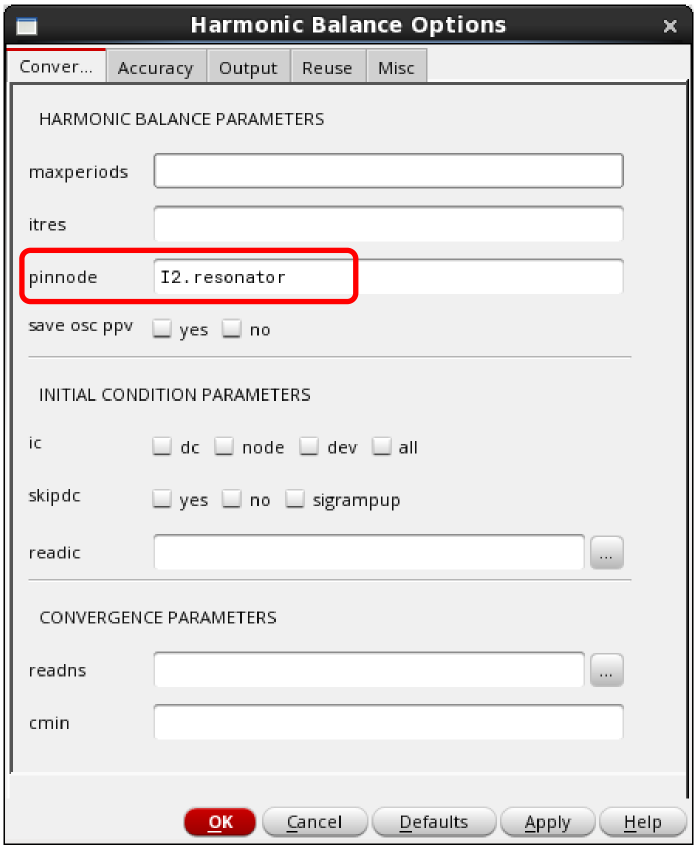
Figure 25 – Setting up pinnode option in the Convergence tab of Harmonic Balance Options form by selecting resonator net between the inductor and the capacitor of the motional equivalent circuit of Crystal Oscillator



39. The tab at the top of the crystal motional equivalent circuit displays the instance name of the crystal symbol. Specify the hierarchical path to the resonator net in the **pinnode** option. In this case, the setting for the **pinnode** would be **I2.resonator**.

    This is shown in the options form below.

Figure 26 - Harmonic Balance Options



If convergence is not attained in the hb analysis, consider raising **maxperiods** to the 300 to 500 range. The default allows 100 iterations before reporting non-convergence. High Q circuits may require more iterations in order to allow convergence to be attained.

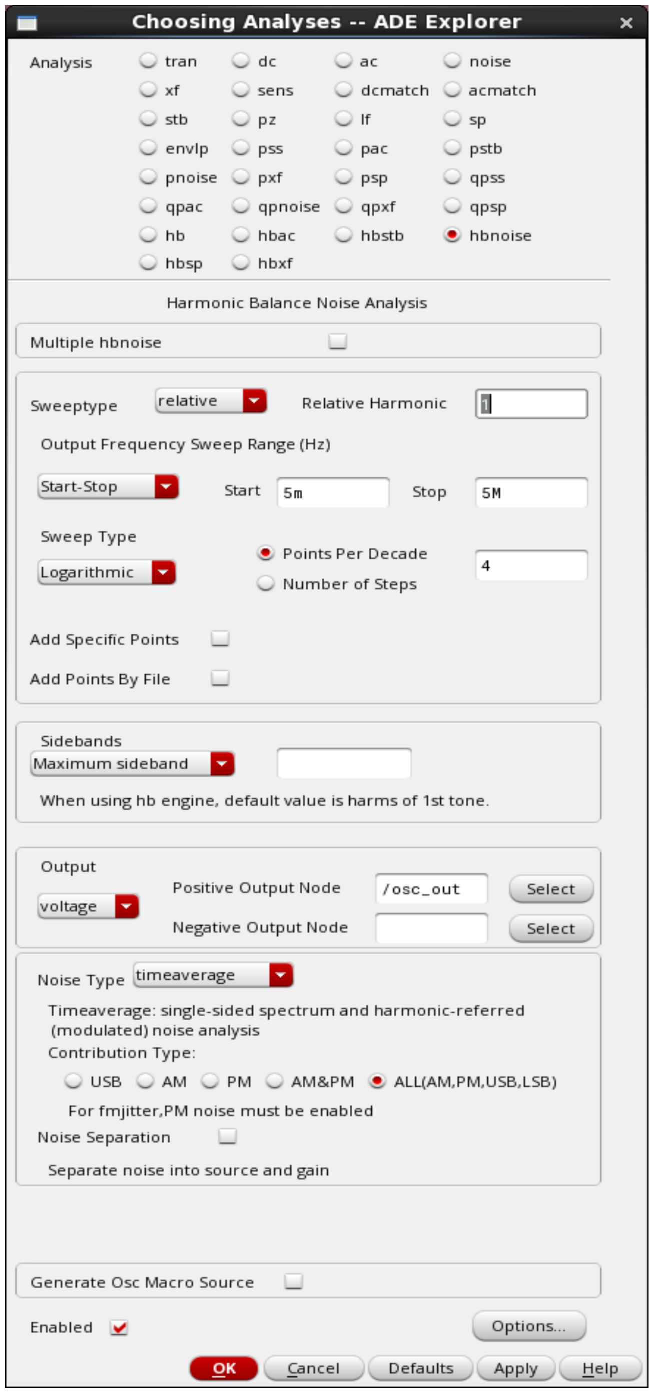
Take the defaults for all the other options.

40. Click **OK** in the **Harmonic Balance Options** form.

Setting Up the Harmonic Balance Noise (hbnoise) Analysis

41. In the **Choosing Analyses** form, select **hbnoise**.

Figure 27 - Choosing Analyses form - Setting up of hbnoise analysis



Analyzing the Harmonic Balance Noise Analysis Settings

The default **Sweeptype** is **relative**. Since a log sweep just above the output frequency is desired, the relative harmonic number is used to specify which harmonic is the output harmonic. Most Crystal Oscillators use the first harmonic.

42. In **relative** sweeptype, the sweep range specifies frequency offsets above the output frequency to be measured. Set an appropriate range for your circuit.

43. **Logarithmic** is usually desired. Generally, it is better to select **Logarithmic** manually, and then specify three to five points per decade, than it is to use the default of **Automatic**. **Automatic** will always run 100 total noise points, which is usually more than required. Specifying a log sweep manually usually saves simulation time.

44. Leave the **Maximum sideband** field blank. This causes noise folding near all the harmonics in the hb large-signal analysis to be included in the hbnoise analysis.

45. Usually, an oscillator has an output net in the circuit; so, the **Output** value should normally be set to **voltage**. If the output is a current, add an iprobe in series with the output of the oscillator in the schematic and set the **Output** selection to **Probe**. In that case, set the instance name of the iprobe in the **Probe** field.

46. Use the **Select** button to the right of the **Positive Output Node** field, and then select the net in the schematic for the noise output. This circuit is single-ended. In any of the fields where there is a negative net in any analysis in SpectreRF, if the **Negative Output Node** field is left blank, the global ground net will be used as the reference net.

47. Generally, the PM component of phase noise is desired. To make this measurement in hbnoise, select **timeaverage** for the **Noise Type** field and then, as **Contribution Type**, either select **PM** or **ALL(AM,PM,USB,LSB)**.

48. Click **OK** to close the **Choosing Analyses** form.

49. Click on the  icon in the ADE Explorer window and run the analysis.

50. Once the analysis is finished, check the Spectre output window (`spectre.out` file).

    There are several things to check in the Spectre output window.

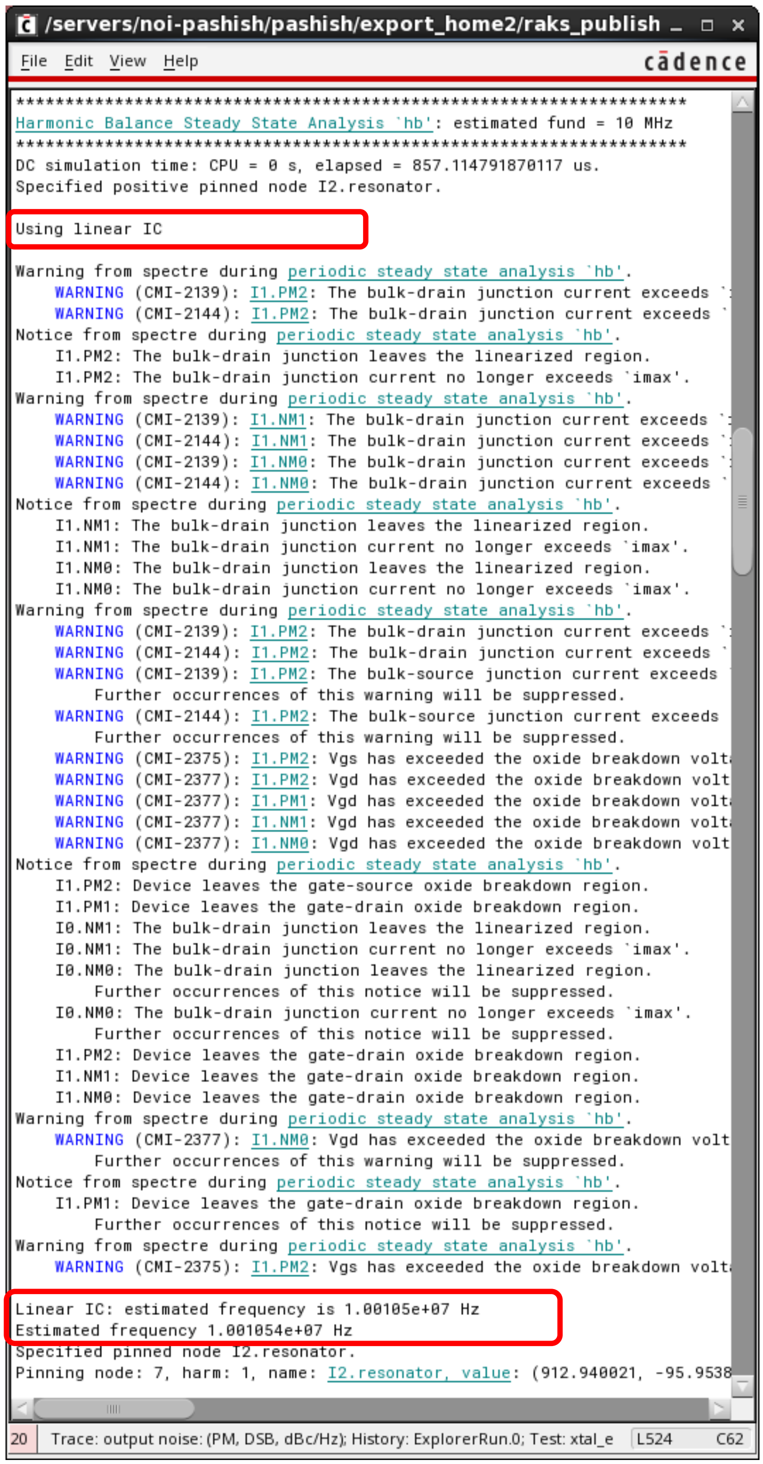
    a. Just below the circuit inventory, the pinnode is displayed. When you run the circuit for the first time, make sure that the schematic net is displayed and there are no warnings indicating that the net does not exist.

b. Just below that is a message that the linear initial condition is being calculated. The warning messages below the Linear IC statement can always be ignored.

c. The first time the simulation runs, or any time you get non-convergence, check that the frequency determined by the Linear IC is correct for your circuit.

d. If the Linear IC fails, try a different pinnode inside the resonator, or change the frequency estimate in the hb **Choosing Analyses** form by about 10%.

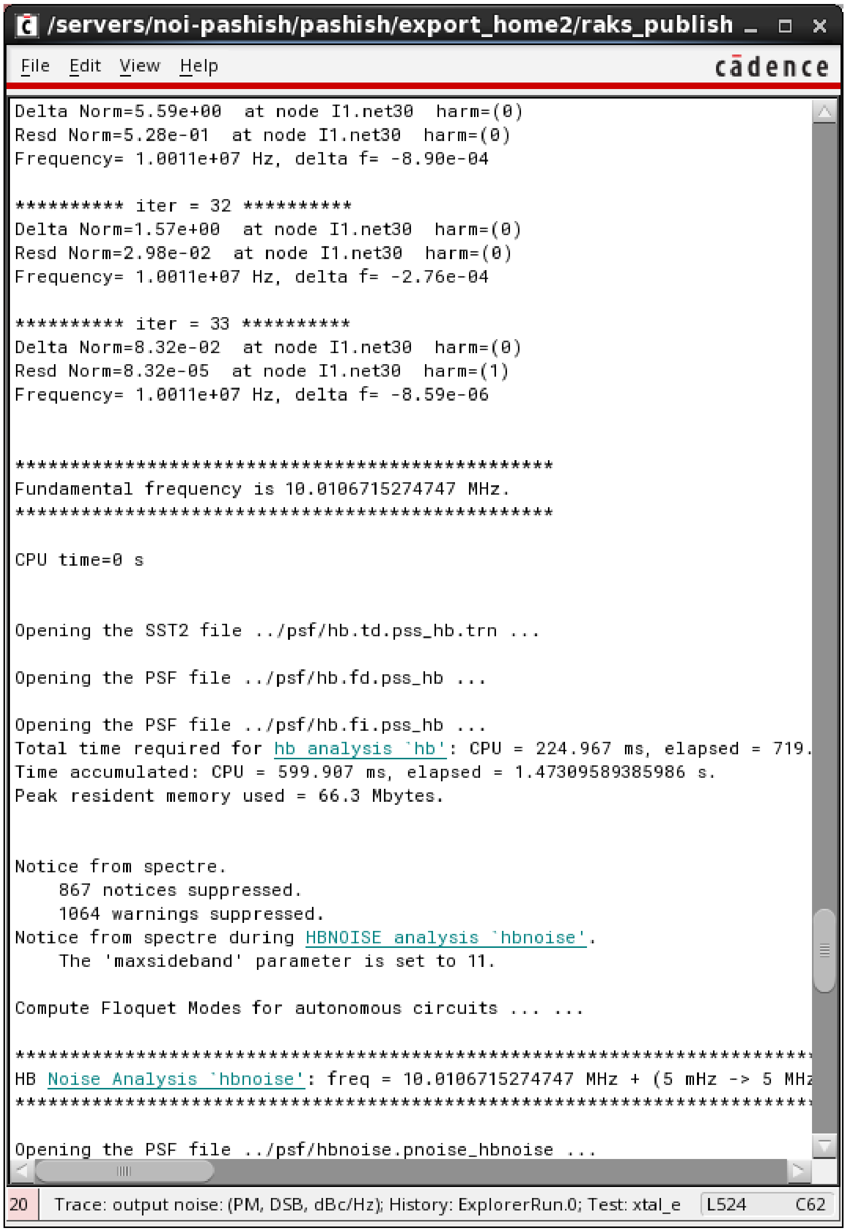
Figure 28 – Analyzing Spectre output log file (spectre.out) – hb analysis output



```
/servers/noi-pashish/pashish/export_home2/raks_publish
File  Edit  View  Help                                                cādence
***********************************************************************
Harmonic Balance Steady State Analysis `hb': estimated fund = 10 MHz
***********************************************************************
DC simulation time: CPU = 0 s, elapsed = 857.114791870117 us.
Specified positive pinned node I2.resonator.

Using linear IC

Warning from spectre during periodic steady state analysis `hb'.
    WARNING (CMI-2139): I1.PM2: The bulk-drain junction current exceeds `
    WARNING (CMI-2144): I1.PM2: The bulk-drain junction current exceeds `
Notice from spectre during periodic steady state analysis `hb'.
    I1.PM2: The bulk-drain junction leaves the linearized region.
    I1.PM2: The bulk-drain junction current no longer exceeds `imax'.
Warning from spectre during periodic steady state analysis `hb'.
    WARNING (CMI-2139): I1.NM1: The bulk-drain junction current exceeds `
    WARNING (CMI-2144): I1.NM1: The bulk-drain junction current exceeds `
    WARNING (CMI-2139): I1.NM0: The bulk-drain junction current exceeds `
    WARNING (CMI-2144): I1.NM0: The bulk-drain junction current exceeds `
Notice from spectre during periodic steady state analysis `hb'.
    I1.NM1: The bulk-drain junction leaves the linearized region.
    I1.NM1: The bulk-drain junction current no longer exceeds `imax'.
    I1.NM0: The bulk-drain junction leaves the linearized region.
    I1.NM0: The bulk-drain junction current no longer exceeds `imax'.
Warning from spectre during periodic steady state analysis `hb'.
    WARNING (CMI-2139): I1.PM2: The bulk-drain junction current exceeds `
    WARNING (CMI-2144): I1.PM2: The bulk-drain junction current exceeds `
    WARNING (CMI-2139): I1.PM2: The bulk-source junction current exceeds
        Further occurrences of this warning will be suppressed.
    WARNING (CMI-2144): I1.PM2: The bulk-source junction current exceeds
        Further occurrences of this warning will be suppressed.
    WARNING (CMI-2375): I1.PM2: Vgs has exceeded the oxide breakdown volt
    WARNING (CMI-2377): I1.PM2: Vgd has exceeded the oxide breakdown volt
    WARNING (CMI-2377): I1.PM1: Vgd has exceeded the oxide breakdown volt
    WARNING (CMI-2377): I1.NM1: Vgd has exceeded the oxide breakdown volt
    WARNING (CMI-2377): I1.NM0: Vgd has exceeded the oxide breakdown volt
Notice from spectre during periodic steady state analysis `hb'.
    I1.PM2: Device leaves the gate-source oxide breakdown region.
    I1.PM1: Device leaves the gate-drain oxide breakdown region.
    I0.NM1: The bulk-drain junction leaves the linearized region.
    I0.NM1: The bulk-drain junction current no longer exceeds `imax'.
    I0.NM0: The bulk-drain junction leaves the linearized region.
        Further occurrences of this notice will be suppressed.
    I0.NM0: The bulk-drain junction current no longer exceeds `imax'.
        Further occurrences of this notice will be suppressed.
    I1.PM2: Device leaves the gate-drain oxide breakdown region.
    I1.NM1: Device leaves the gate-drain oxide breakdown region.
    I1.NM0: Device leaves the gate-drain oxide breakdown region.
Warning from spectre during periodic steady state analysis `hb'.
    WARNING (CMI-2377): I1.NM0: Vgd has exceeded the oxide breakdown volt
        Further occurrences of this warning will be suppressed.
Notice from spectre during periodic steady state analysis `hb'.
    I1.PM1: Device leaves the gate-drain oxide breakdown region.
        Further occurrences of this notice will be suppressed.
Warning from spectre during periodic steady state analysis `hb'.
    WARNING (CMI-2375): I1.PM2: Vgs has exceeded the oxide breakdown volt

Linear IC: estimated frequency is 1.00105e+07 Hz
Estimated frequency 1.001054e+07 Hz
Specified pinned node I2.resonator.
Pinning node: 7, harm: 1, name: I2.resonator, value: (912.940021, -95.9538
20  Trace: output noise: (PM, DSB, dBc/Hz); History: ExplorerRun.0; Test: xtal_e   L524   C62
```

e. Then, hb runs and reports progress.

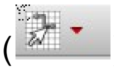
Figure 29 - Analyzing Spectre output log file (spectre.out) – hb analysis output



f. For the last several iterations, the Delta Norm and the Resd Norm should be steadily decreasing, and the delta frequency should be some small fraction of 1 Hz.

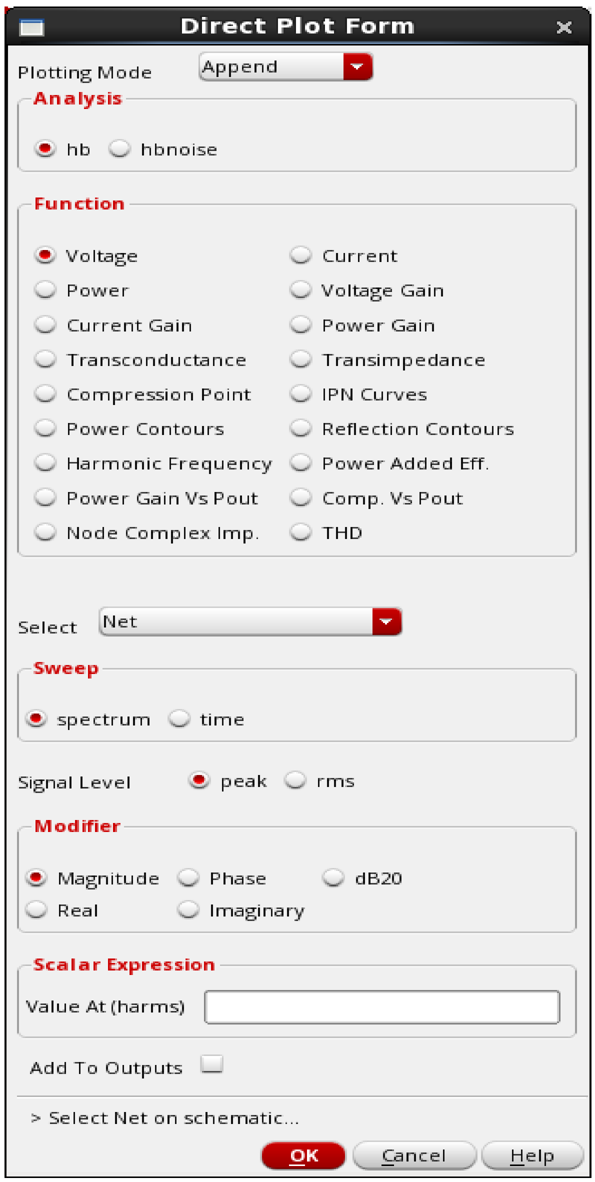
g. When the **Noise Type** is set to **timeaverage** and **ALL**, hbnoise runs frequencies above and below the output harmonic; so, you will see positive and negative frequencies in the hbnoise output file.

51. When the analysis completes, open the **Direct Plot Form** using one of the following two methods:

   a. From the **10M_Crystal_Oscillator** schematic tab, select the red arrow to the right of the Direct Plot icon () and then select **Main Form**.

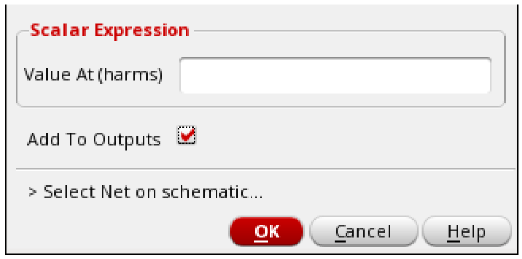
   b. From the **maestro_Oscillator_Normal_Setup** tab in the ADE Explorer window, select **Results > Direct Plot > Main Form**. The **Direct Plot Form** is as shown below.

Figure 30 - Direct Plot Form - Plotting hb analysis results



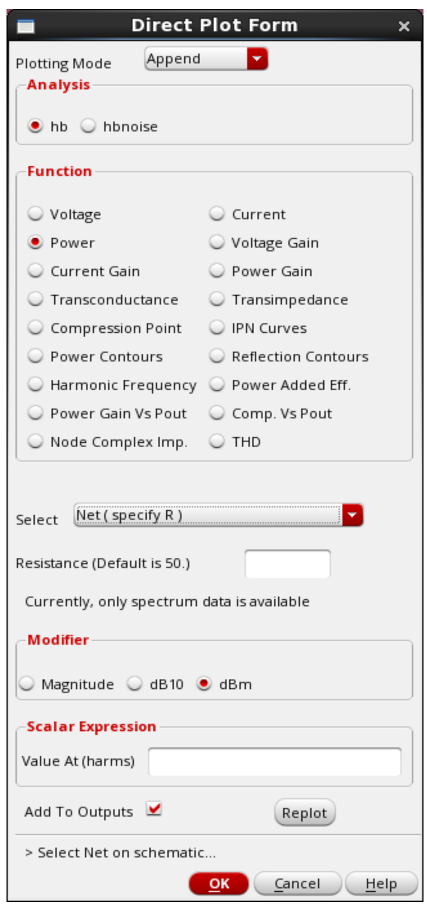
52. If you want to have the selected curves plot automatically after you make changes to your circuit and re-simulate, click the **Add To Outputs** checkbox at the bottom of the form.

Figure 31 - Enable 'Add to Outputs' checkbox



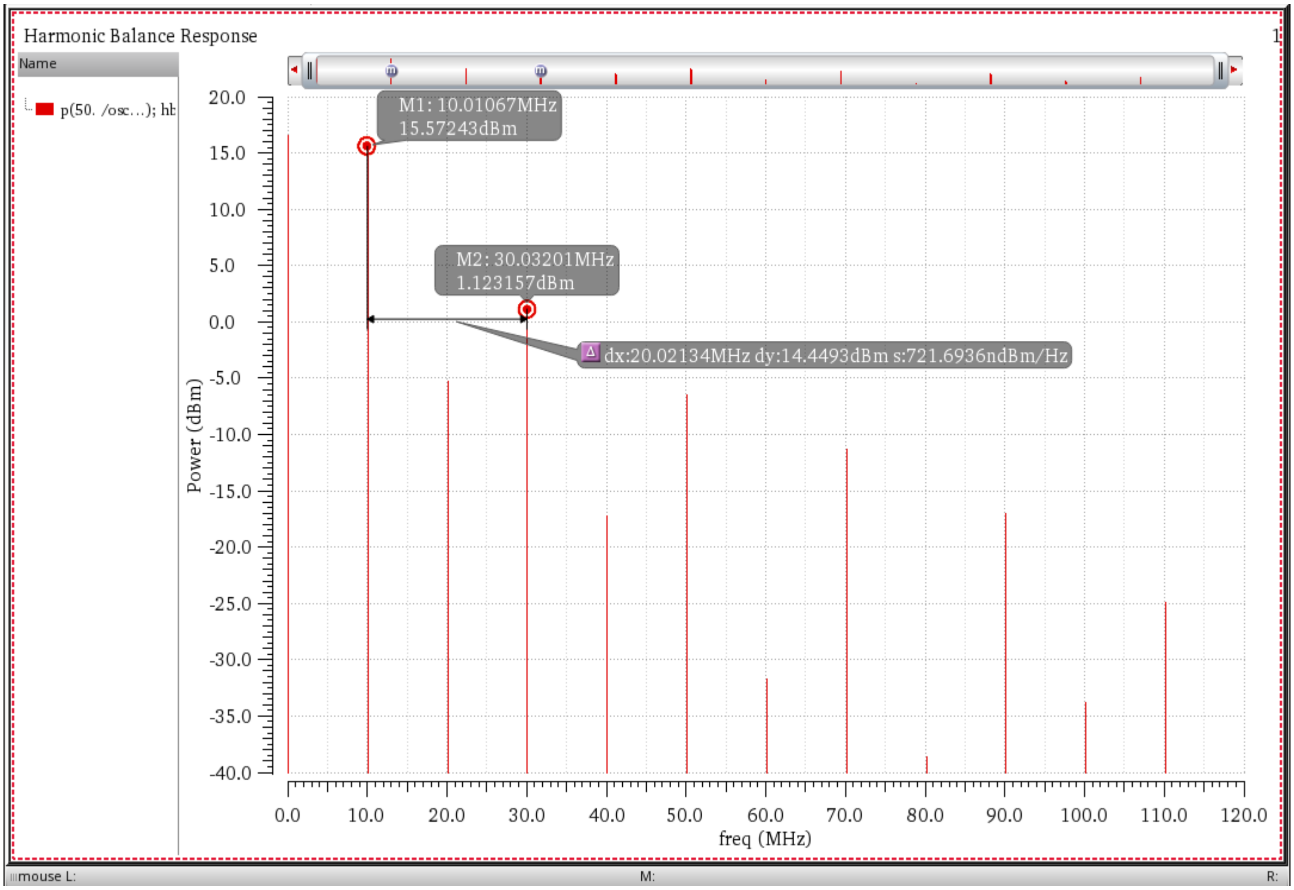
53. To plot the hb results, select **hb** at the top of the form (if not already selected). Many formats are possible. In this example, power based on the voltage on a net with an assumed load resistor is plotted. Set up the **Direct Plot Form** as shown in Fig 32.

Figure 32 - Plot power at osc_out net



54. Select the **osc_out** net from the schematic. The result is as shown in Fig 33.

Figure 33 - osc_out net voltage output spectrum from hb analysis

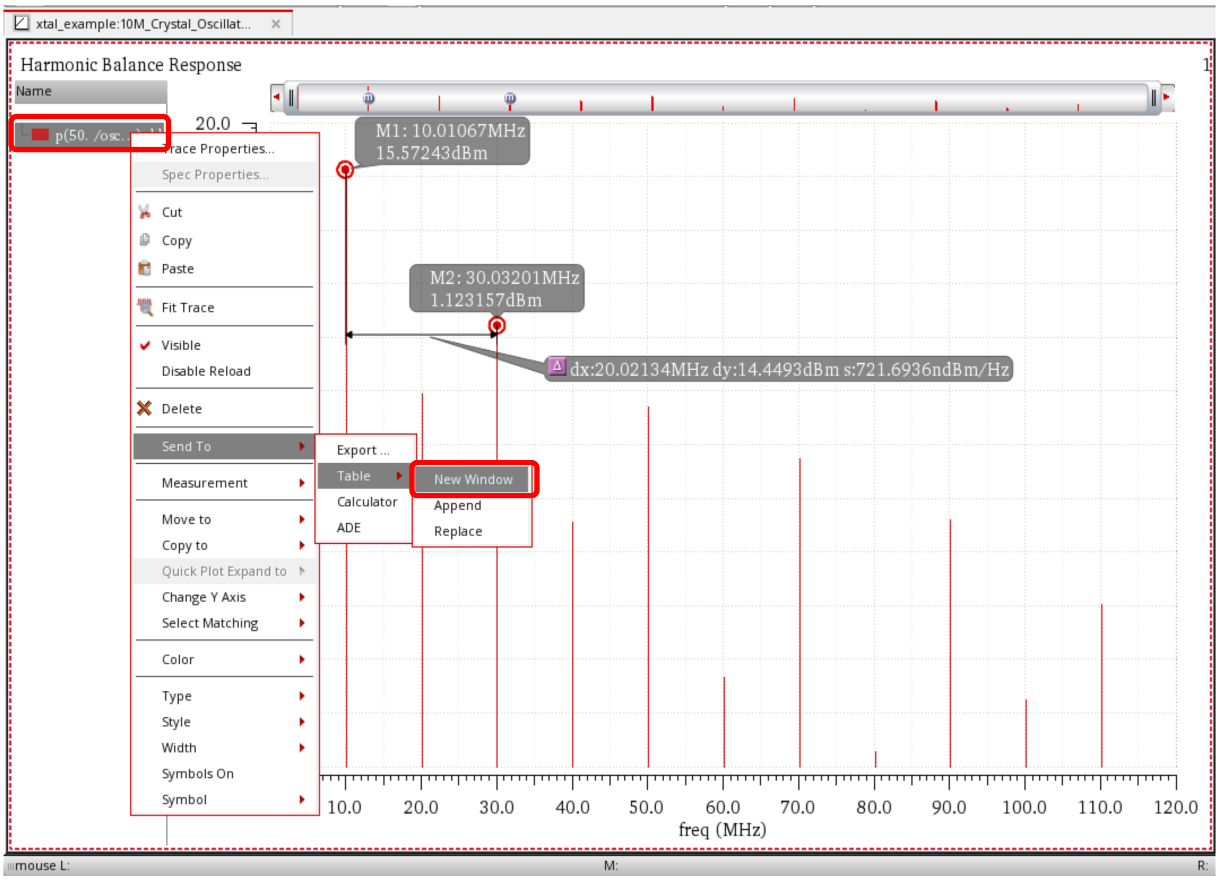


55. One way to determine the frequency more exactly is to send the spectral results to a table, and then set more digits for the frequency. To do this, perform the following steps:

    a. Select the trace legend. The legend will highlight.

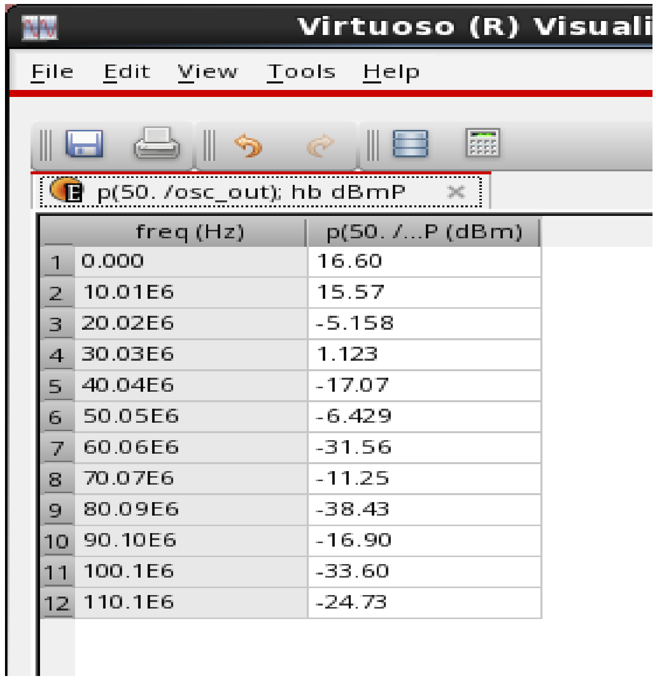
    b. Right-click and release. Move to **Send To > Table > New Window**.

Figure 34 - hb analysis results – Send To a Table in New Window



The table window appears as shown in Fig 35.

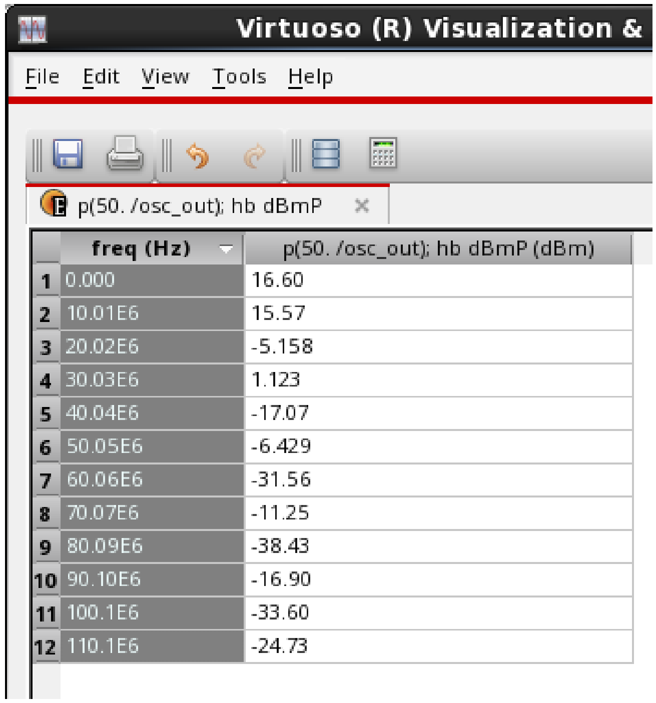
Figure 35 – osc_out voltage spectrum output in ViVA table



	freq (Hz)	p(50. /...P (dBm)
1	0.000	16.60
2	10.01E6	15.57
3	20.02E6	-5.158
4	30.03E6	1.123
5	40.04E6	-17.07
6	50.05E6	-6.429
7	60.06E6	-31.56
8	70.07E6	-11.25
9	80.09E6	-38.43
10	90.10E6	-16.90
11	100.1E6	-33.60
12	110.1E6	-24.73

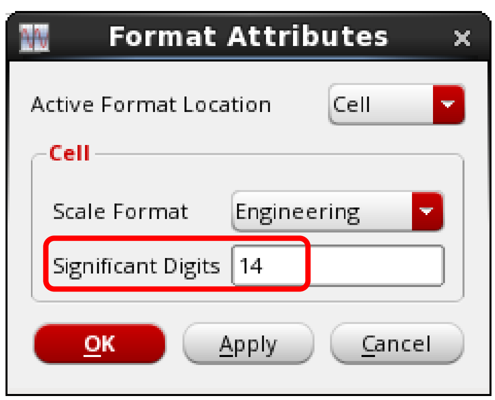
56. Select the frequency column. It will highlight as shown in Fig 36.

Figure 36 - Selecting the frequency column in Viva table



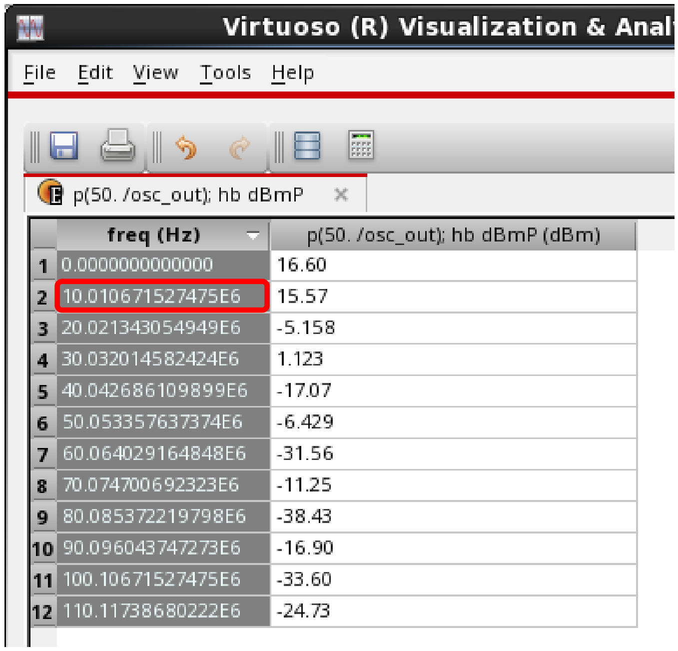
57. In the table window, select **View > Format**. In the **Format Attributes** window, type **14** in the **Significant Digits** field.

Figure 37 – Setting Format Attributes – Significant Digits is set to 14



58. Select **OK**. The frequency field updates with 14 digits as shown in Fig 38.

Figure 38 – Frequency column showing 14 significant digits in ViVA table



	freq (Hz)	p(50. /osc_out); hb dBmP (dBm)
1	0.0000000000000	16.60
2	10.010671527475E6	15.57
3	20.021343054949E6	-5.158
4	30.032014582424E6	1.123
5	40.042686109899E6	-17.07
6	50.053357637374E6	-6.429
7	60.064029164848E6	-31.56
8	70.074700692323E6	-11.25
9	80.085372219798E6	-38.43
10	90.096043747273E6	-16.90
11	100.10671527475E6	-33.60
12	110.11738680222E6	-24.73

The frequency of the oscillator is shown with more resolution.

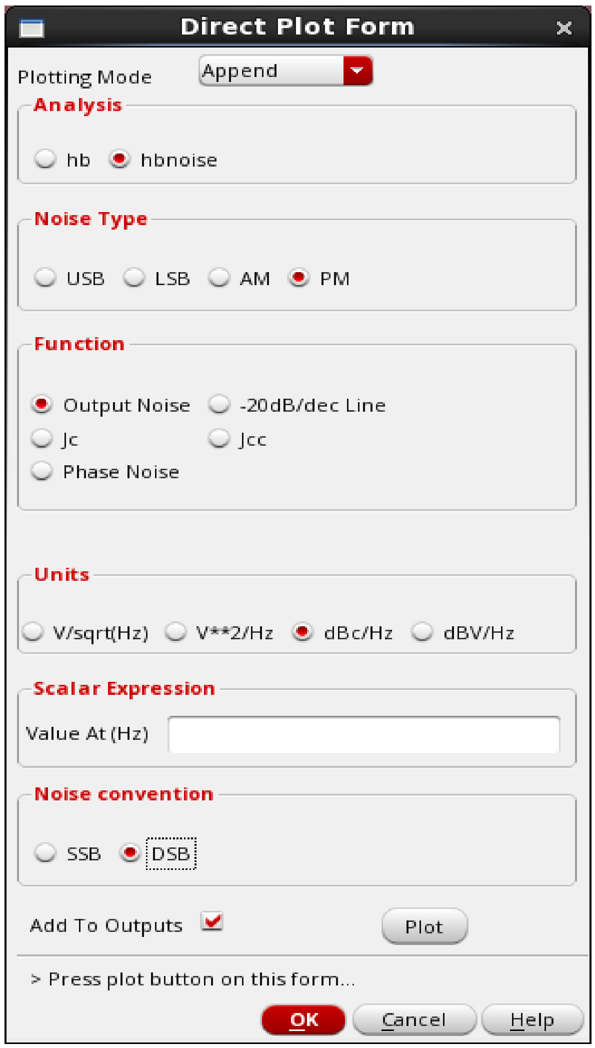
59. Close the table window and the waveform window.

There is a second method coming up shortly.

60. To plot the PM component of phase noise, select hbnoise results.

61. Select the **PM** component as **Noise Type** and then **dBc** as Modifier. Also, set the **Noise convention** as **DSB**.

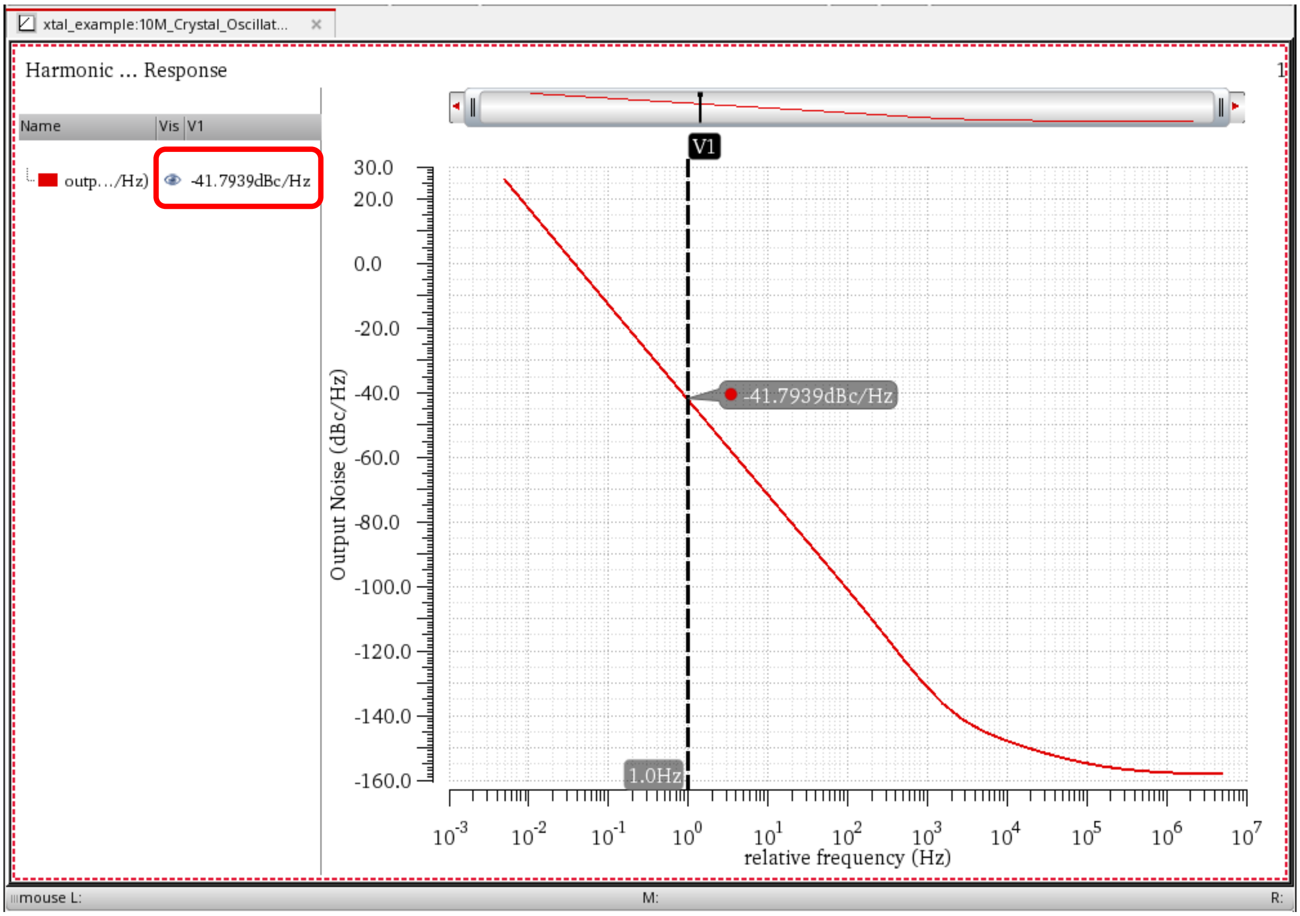
Figure 39 - Direct Plot Form - Plotting hbnoise results



62. Click **Plot**.

    The result is shown in Fig 40.

Figure 40 - Plotting PM noise of oscillator



63. The phase noise curve has a vertical marker placed at 1Hz. To place a vertical marker, move your mouse cursor near the desired frequency on the trace and type **v**. To exactly place the marker, select it and then the frequency at the lower left of the marker becomes editable.

    The readout to the right of the legend is the value at the vertical marker. This is highlighted in the red box above.

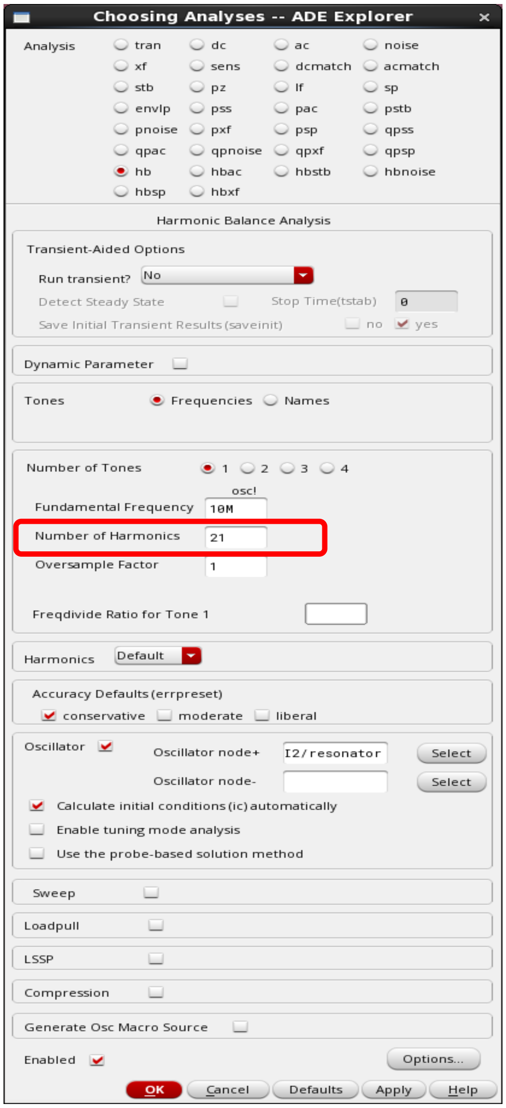
64. Click **OK** to close the **Direct Plot Form**.

Verifying If the Harmonics Set Are Enough

Check to see if you have enough harmonics. Many Crystal Oscillators operate at frequencies much lower than the F_t of the devices. There is the potential of noise at higher frequencies folding down to the output frequency; so, checking to see if there are enough harmonics is strongly suggested.

65. Go to the **maestro_Oscillator_Normal_Setup** tab in the ADE Explorer window and double-click on the **hb** analysis line in the **Analyses** section of the Setup Assistant. Increase the number of harmonics by about double. In this example, the number of harmonics was changed to **21**.

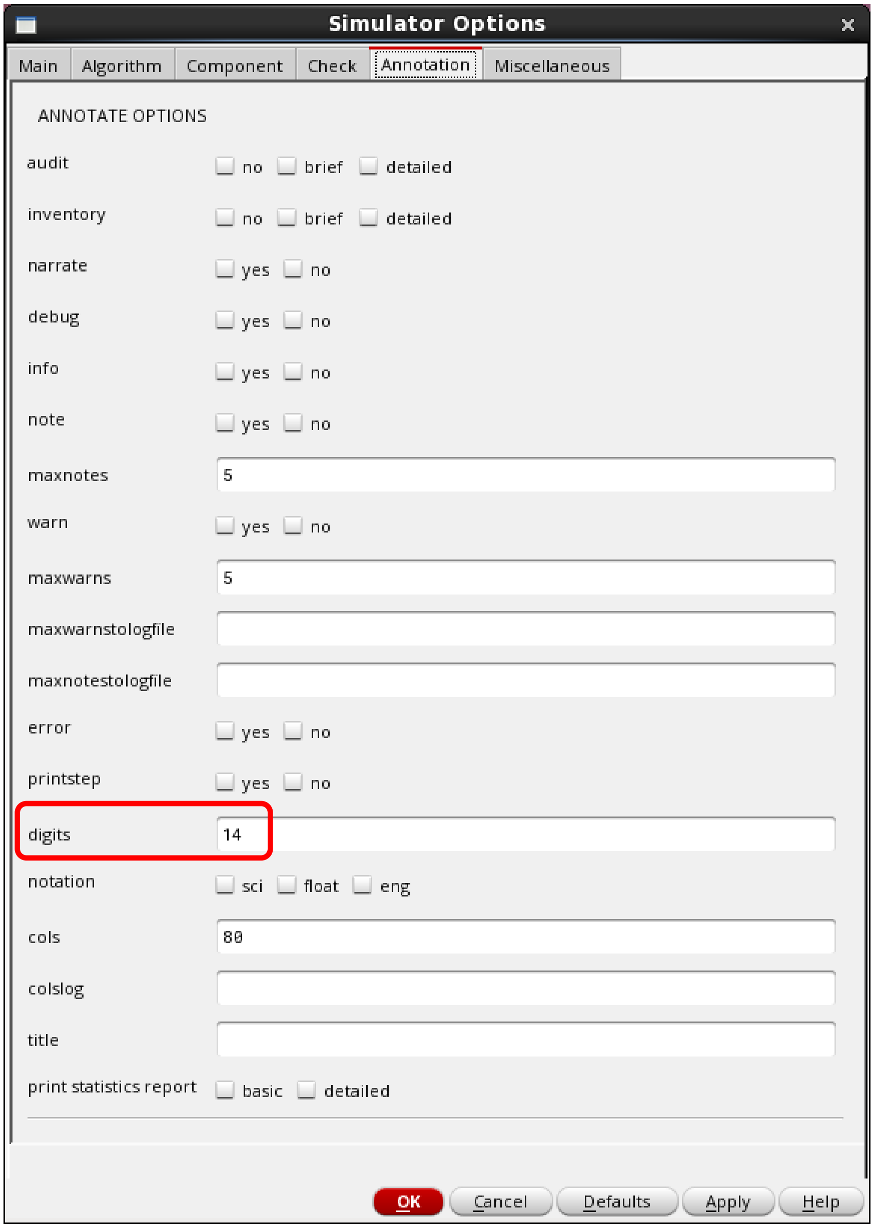
Figure 41 - Choosing Analyses form - Setting up hb analysis with number of harmonics set to 21



66. Click **OK** in the **Choosing Analyses** form.

67. To get more digits in the Spectre output window, open the **Simulator Options** form by going to **Simulation > Options > Analog**. Then, select the **Annotation** tab. Type **14** in the **digits** field (if not set already).

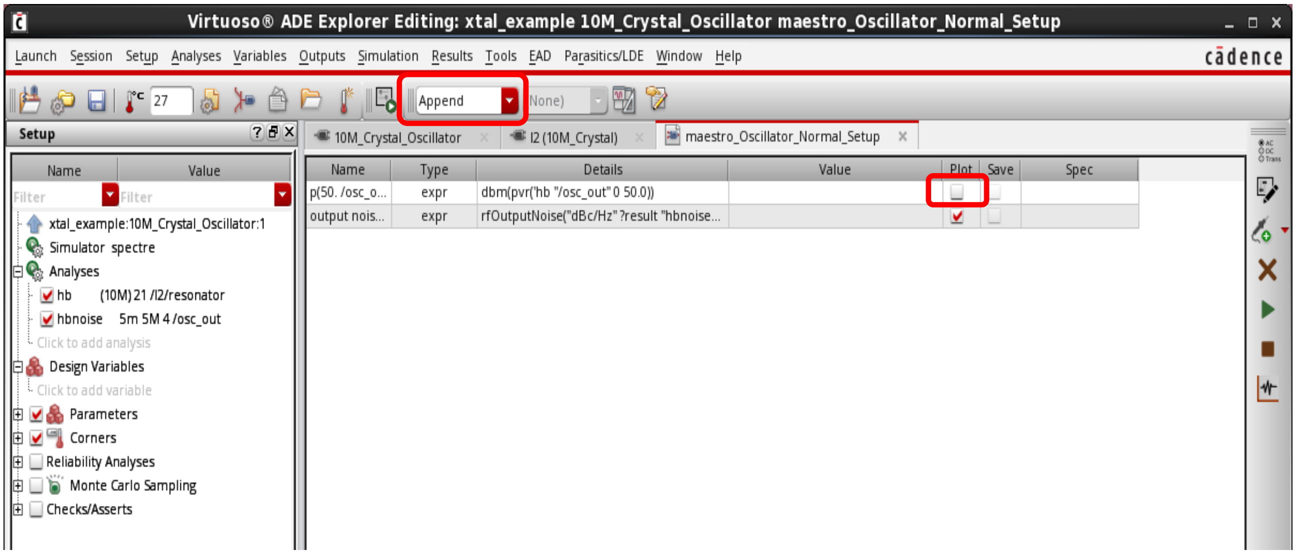
Figure 42 - hb analysis - Simulator Options



68. Click **OK** to close the **Simulator Options** form.

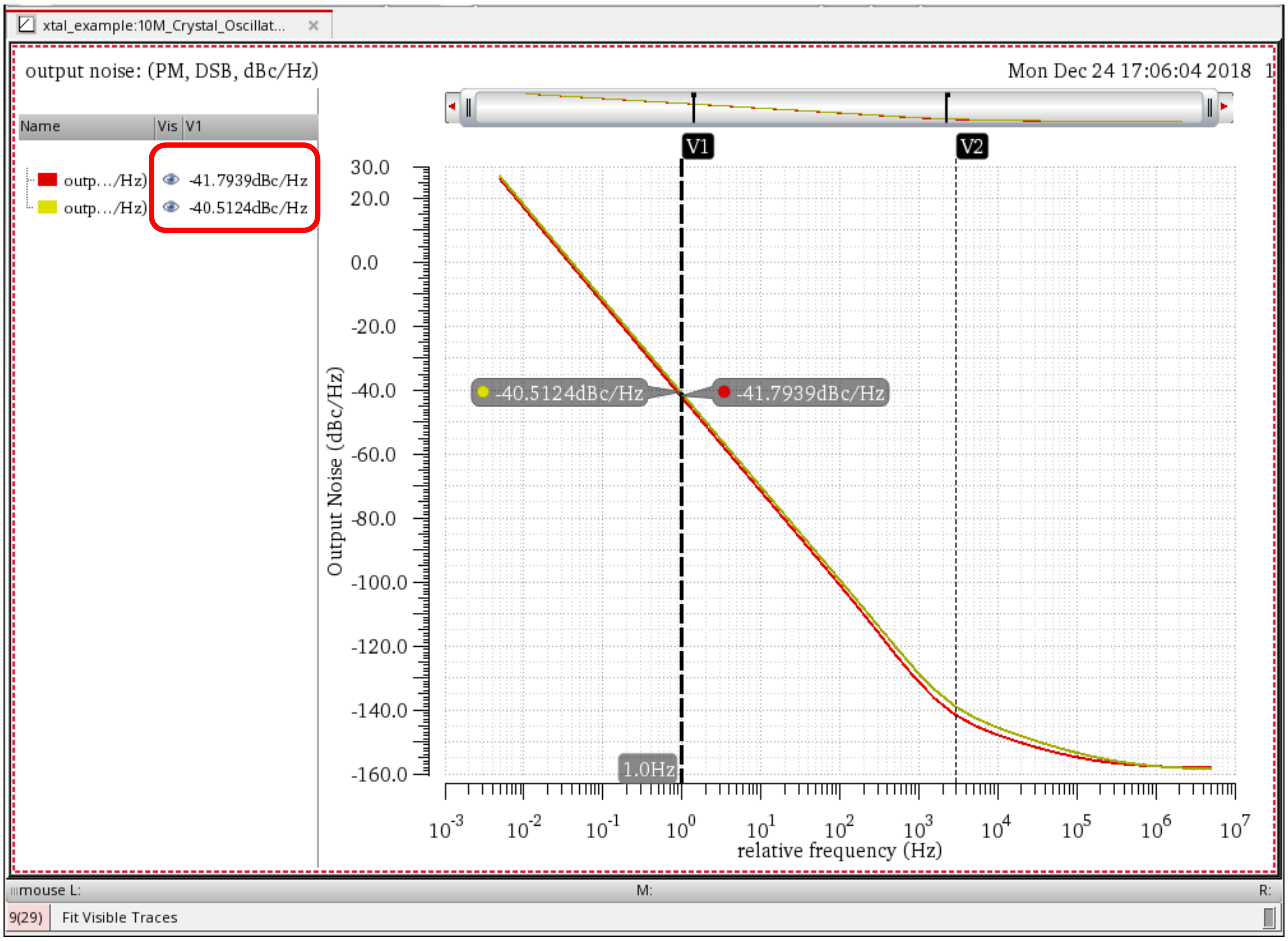
69. To plot the new result over the existing result, set the Plotting Mode in the ADE Explorer window to **Append** mode. Since the focus is on noise, the hb large-signal result has been deselected in the outputs section.

Figure 43 - ADE Explorer window when the Plotting mode is set to Append and hb large-signal results deselected in the outputs section



70. Run the simulation by clicking the  icon and compare the noise results.

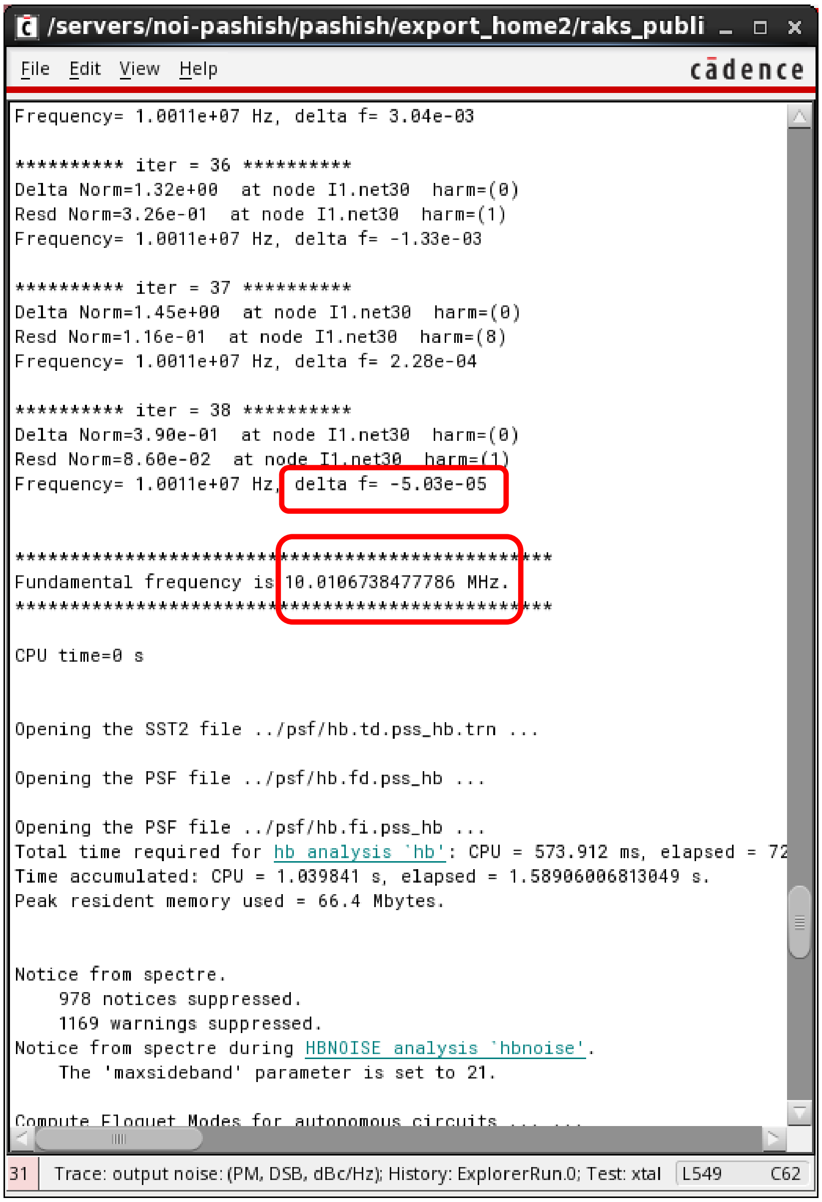
Figure 44 - PM noise plot of Crystal Oscillator



The noise result at 1Hz changed considerably. Also, the noise above about 100Hz is visibly different. An additional vertical marker was placed at 3 KHz. The readout to the right of the legend will read the result of the selected vertical marker.

71. Note the frequency in the Spectre output window. Go to the bottom of the hb output and read the frequency. This is highlighted below.

Figure 45 - Analyzing Spectre output log file (spectre.out) – hb analysis simulation output

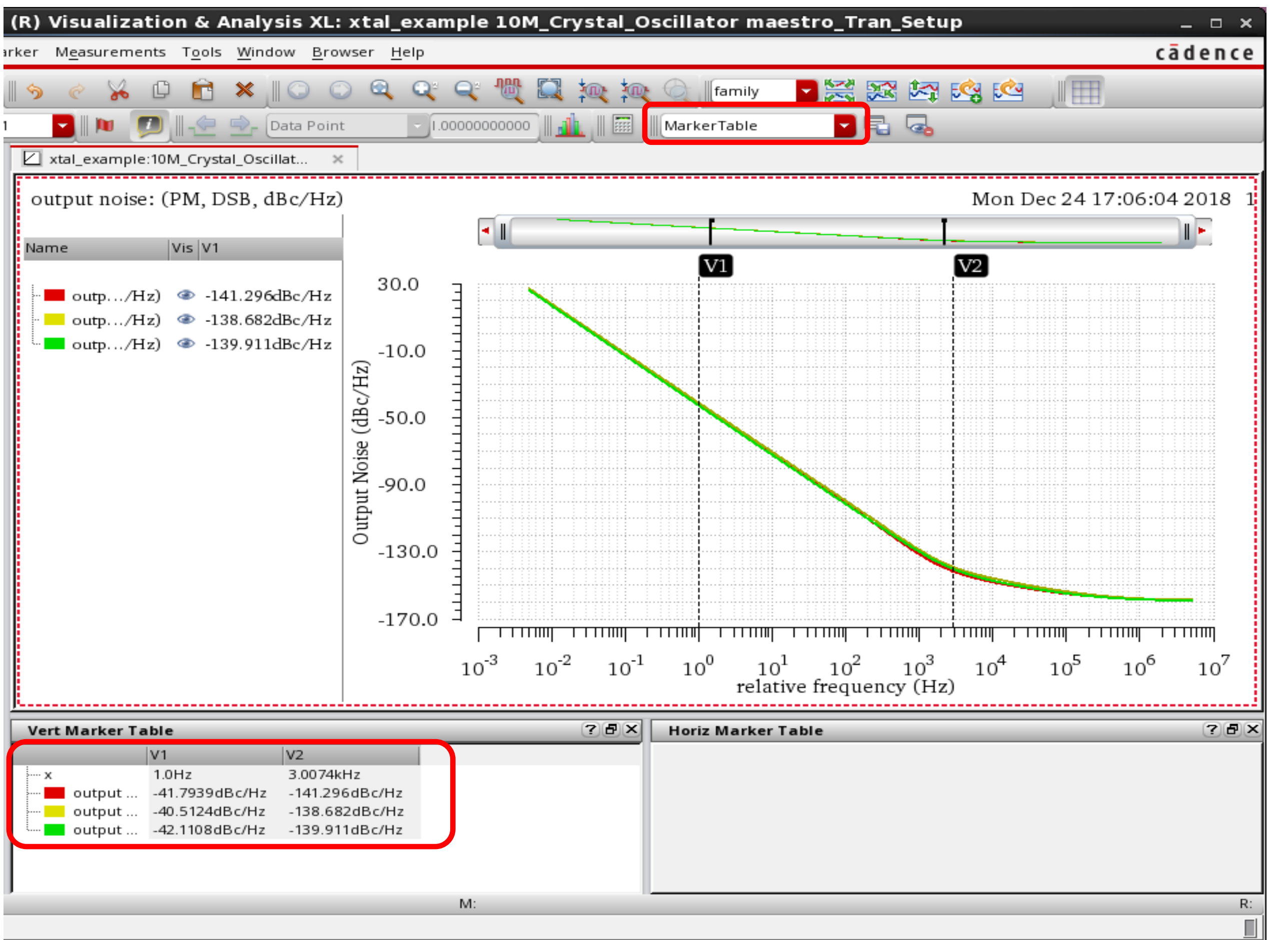


Note that the fundamental frequency is around 10MHz, and the delta f on the last iteration is 5.03e-05 Hz. This gives an actual resolution of 5.03e-05/1e7 or 5.03e-12. This will change with different numbers of harmonics because the solution is slightly different. The frequency will also be very slightly different.

72. Now, roughly double the number of harmonics to 41 and rerun the simulation. An odd number of harmonics is chosen because, in this case, the odd harmonics are emphasized in the large-signal result.

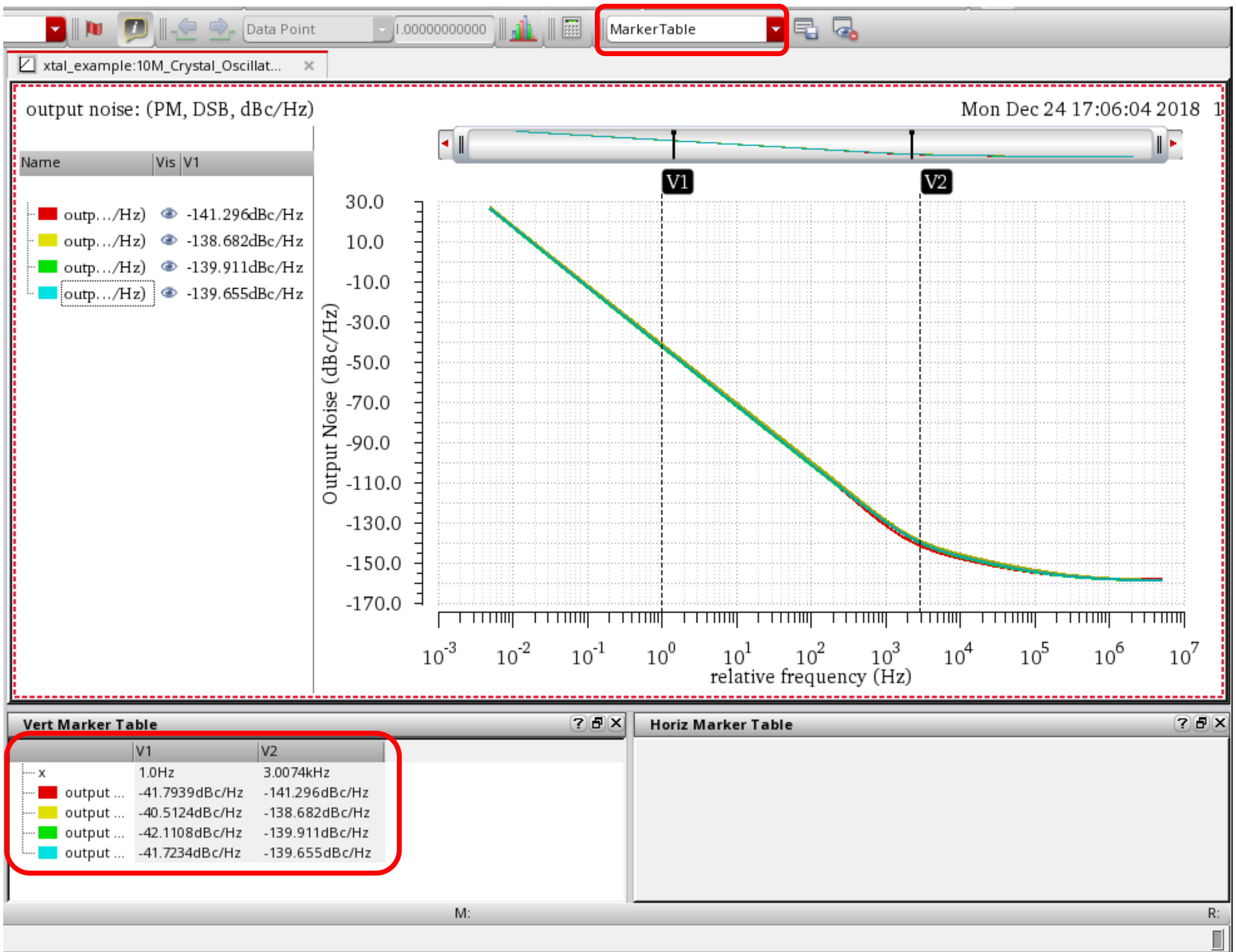
73. When the noise result plots, select the **MarkerTable** Workspace in the ViVA window. This shows the values of both the markers at the bottom of the waveform tool.

Figure 46 – PM noise plot of Crystal Oscillator – MarkerTable Workspace in ViVA window (when number of harmonics is set to 41 in hb analysis)



74. The results for both markers are still changing; so, double the harmonics to 81 and rerun the simulation.

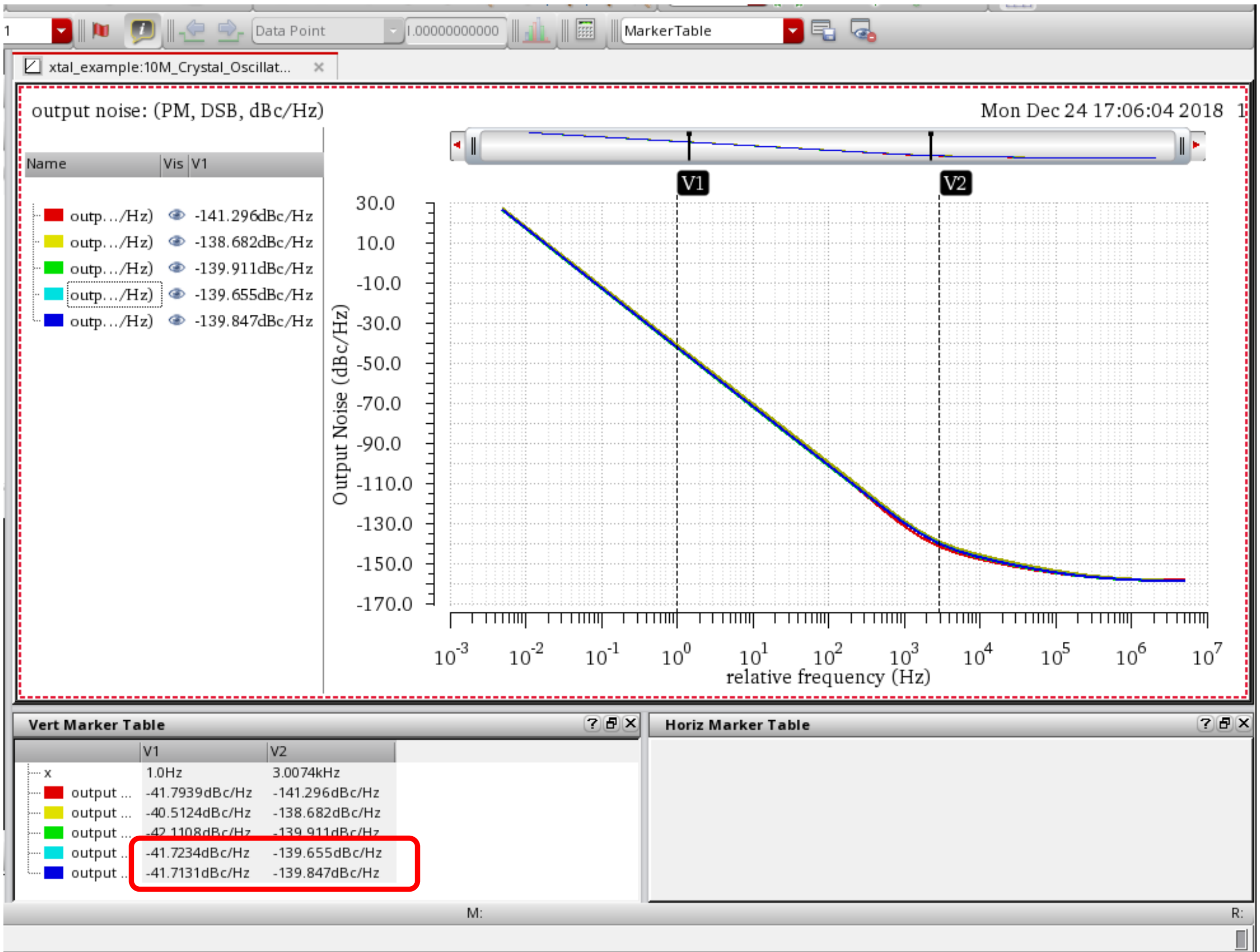
Figure 47 - PM noise plot of Crystal Oscillator – MarkerTable Workspace in ViVA window (when number of harmonics is set to 81 in hb analysis)



	V1	V2
x	1.0Hz	3.0074kHz
output ...	-41.7939dBc/Hz	-141.296dBc/Hz
output ...	-40.5124dBc/Hz	-138.682dBc/Hz
output ...	-42.1108dBc/Hz	-139.911dBc/Hz
output ...	-41.7234dBc/Hz	-139.655dBc/Hz

75. The results are still changing; so, rerun the simulation with 161 harmonics.

Figure 48 - PM noise plot of Crystal Oscillator – MarkerTable Workspace in ViVA window (when number of harmonics is set to 161 in hb analysis)



76. The result is stable at 81 and 161 harmonics; so, 81 harmonics are enough. If desired, you can try 321 harmonics; you will see that the result is almost the same and the runtime is much longer.

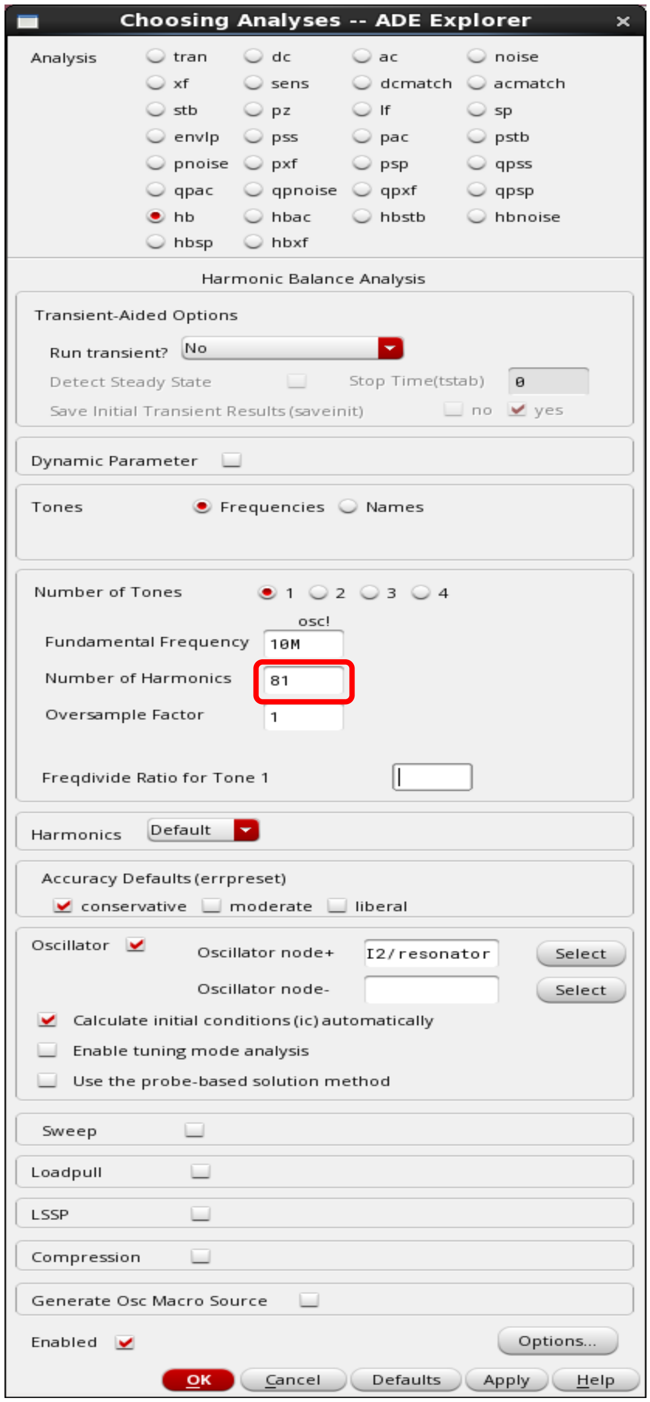
77. Close the waveform tool when you are satisfied that 81 harmonics are enough.

Investigating the Effect of Running the Transient in the tstab Interval

Using the estimate of the oscillator frequency as the starting point of the harmonic balance simulation without running tstab is a viable strategy for many oscillators. However, for nonlinear oscillators, the starting point provided by the initial condition calculation is not close enough to allow convergence. In this case, try setting a small number of cycles in tstab, and running again. Running that small number of cycles and using the last cycle's fft as the starting point for the harmonic balance iterations can improve the convergence of the oscillator. Sometimes, this also saves the overall simulation time by reducing the number of frequency domain iterations in harmonic balance. This is not a given. Some oscillators will only converge when tstab is specified, and some oscillators will only converge when tstab is not run. This section will investigate this strategy.

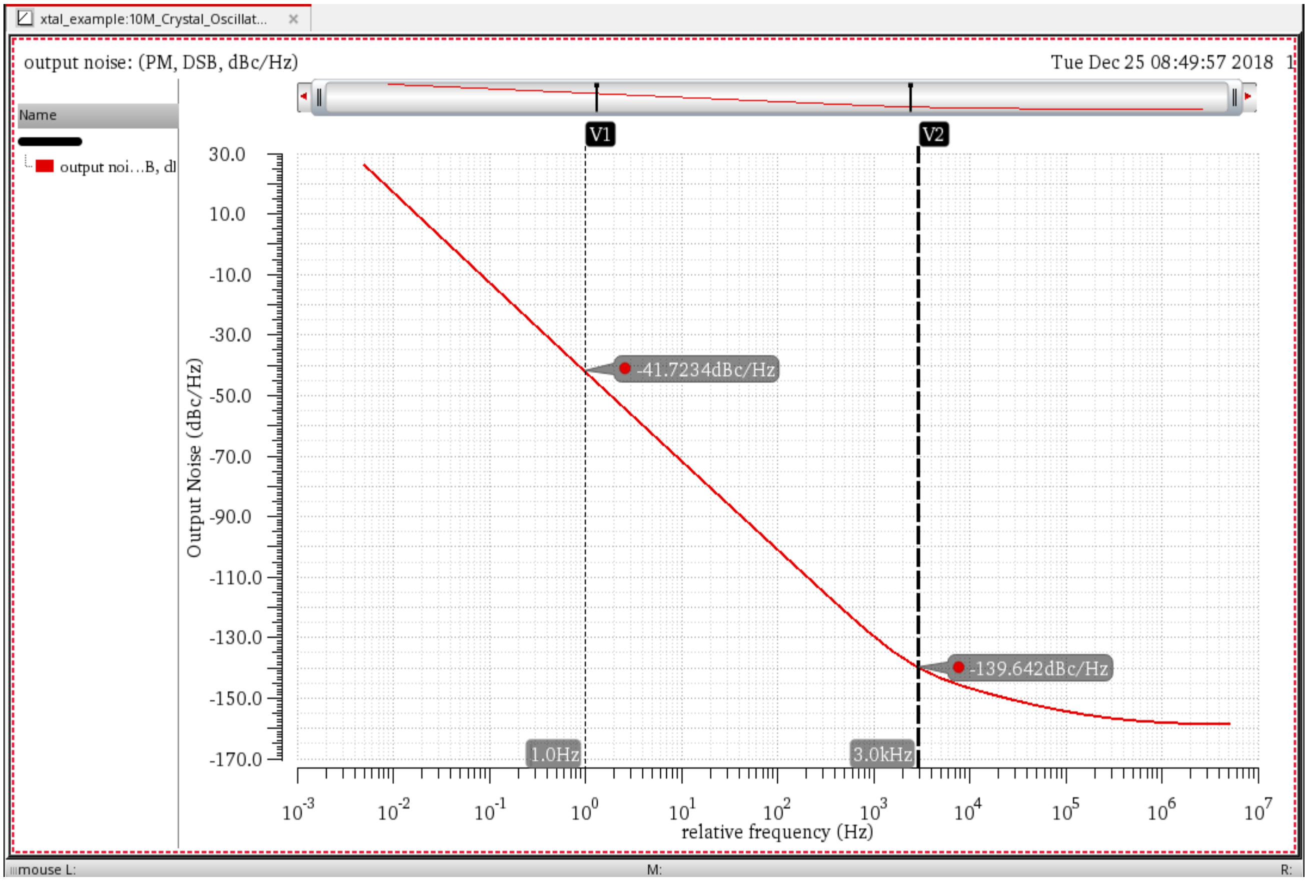
78. In the ADE Explorer window, double-click on the **hb** analysis line in the **Analyses** section of the Setup Assistant, and set the **Number of Harmonics** to **81**.

Figure 49 - Choosing Analyses form - Setting up hb analysis with maxharms set to 81



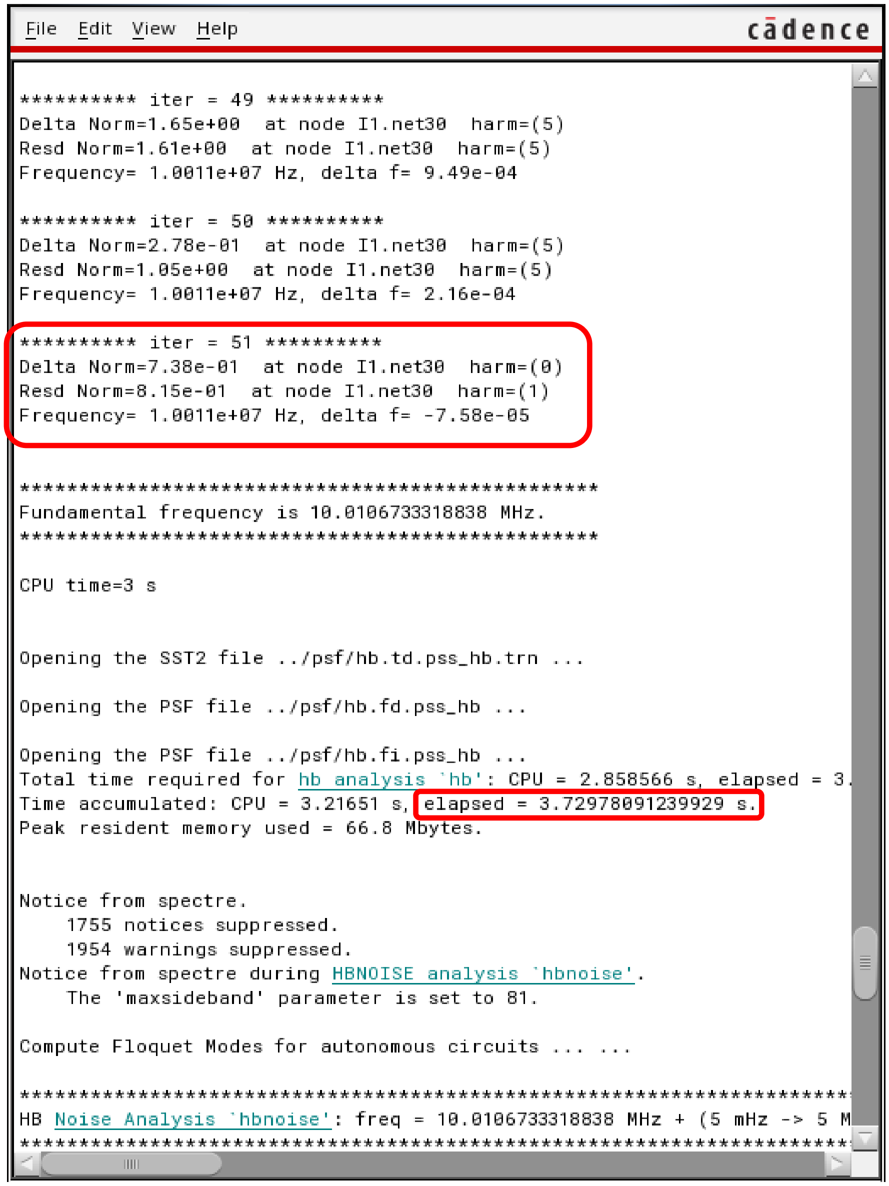
79. Run the simulation. The noise plots.

Figure 50 - PM noise plot of Crystal Oscillator (when Number of Harmonics is set to 81)



80. Note the Spectre output window. For the harmonic balance simulation, 51 iterations are required, and the elapsed time is 3.73 seconds.

Figure 51 - Analyzing Spectre output log file (spectre.out) – hb analysis simulation output (Observing the total number of iterations and the elapsed time for hb simulation)



81. In the ADE Explorer window, double-click on the **hb** analysis line in the **Analyses** section of the Setup Assistant.

    a. Now, set the **Run transient** choice to **Yes**.

    b. Uncheck the **Detect Steady State** checkbox.

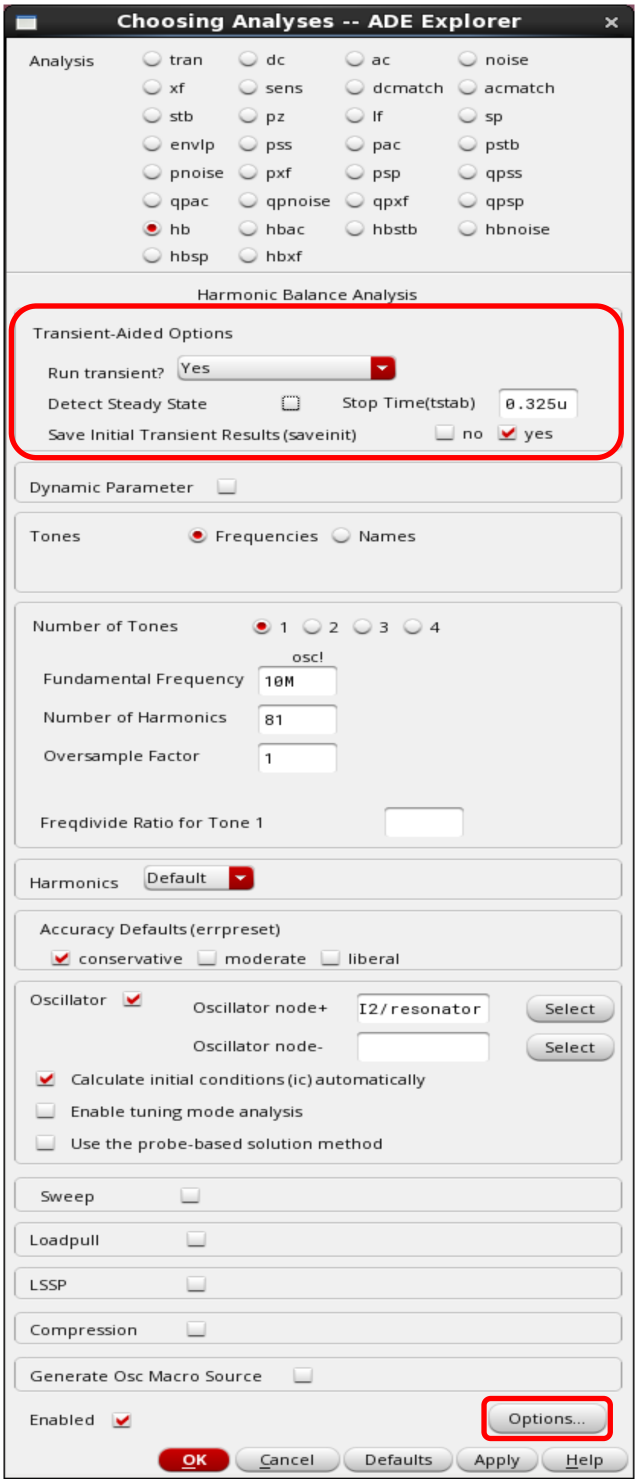
       Since the crystal has a very high Q, steady-state will be detected almost immediately. Turn this off to make sure that the full time specified is actually run. This also eliminates the overhead of checking for steady-state.

    c. The frequency of this oscillator is 10MHz. The period is 0.1usec. Set **Stop Time (tstab)** to **0.325u**, which will run 3.25 cycles of the waveform in tstab. Remember from the transient analysis that the waveform starts at the peak voltage in the resonator. If an integer number of cycles are run in tstab, the period calculation will be hard because the waveform starts and ends at a peak. An additional quarter cycle has been added to tstab to allow the waveform to be roughly in the middle. This makes the period calculation from the waveform much more accurate and improves the chances of convergence because the frequency is more accurate for the first iteration of the hb analysis.

    d. Select **yes** in the checkbox to the right of **Save Initial Transient Results (saveinit)**.

Your **Choosing Analyses** form should look like the one as shown below.

Figure 52 - Choosing Analyses form - Setting up hb analysis with tstab (Enabling Transient-Aided Options)

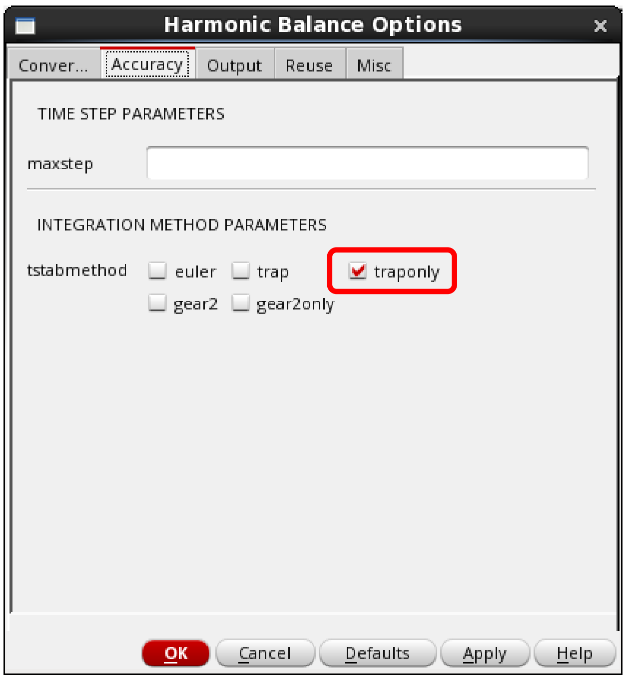


82. Click the **Options** button at the bottom of the form. This will bring up the **Harmonic Balance Options** form.

83. In the **Harmonic Balance Options** form, perform the following actions:

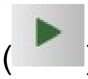
   - Click the **Accuracy** tab.
   - Set the **tstabmethod** option to **trap** or **traponly**. Selecting **traponly** selects an electrically neutral numerical integration method. Do not use any other method as this would introduce numerical damping in the tstab transient that could cause the oscillations in the resonator to be smaller than the actual amplitude, which would reduce the chances of convergence. The **euler** option numerically damps especially heavily, and should never be used for any oscillator.

Figure 53 - Harmonic Balance analysis - Harmonic Balance Options



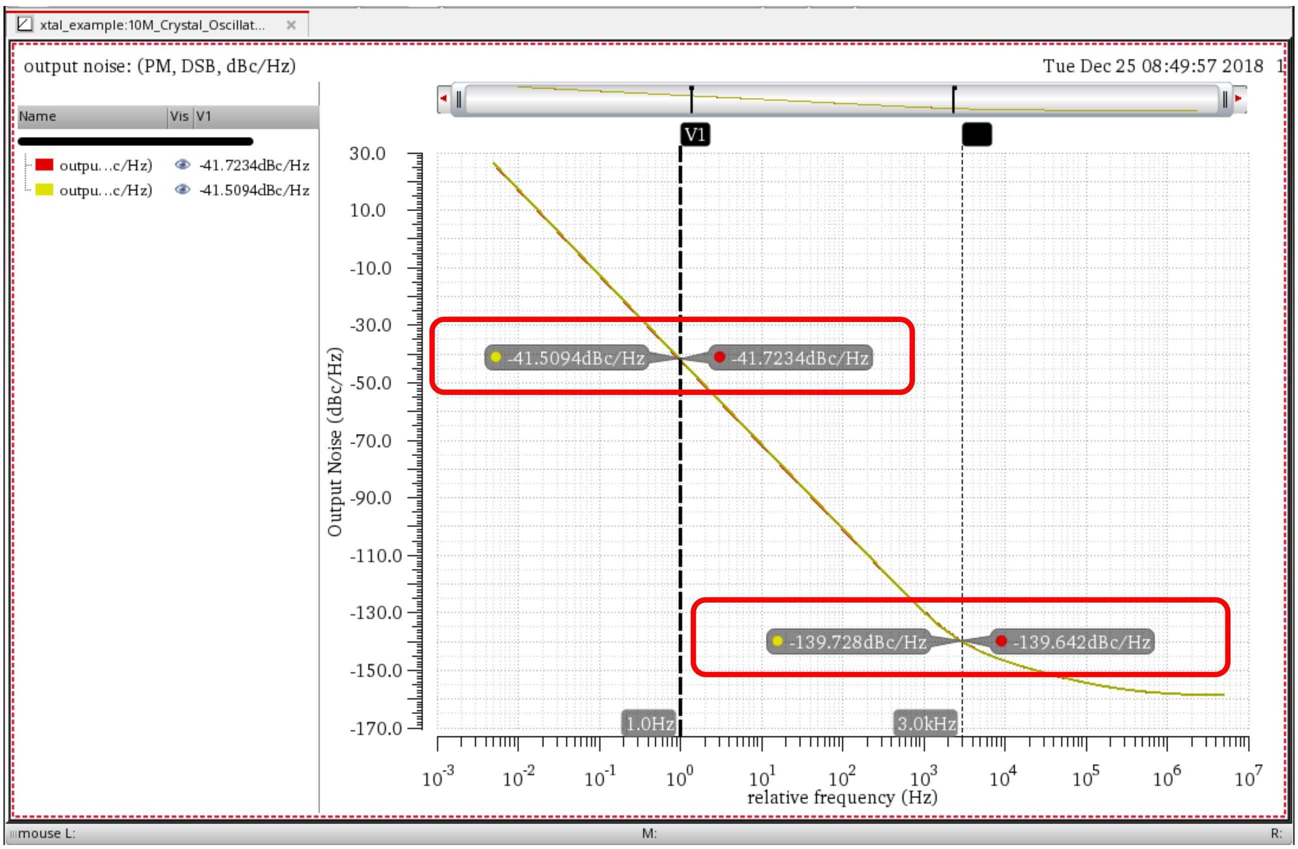
84. Click **OK** to close the **Harmonic Balance Options** form.

85. Click **OK** to close the **Choosing Analyses** form.

86. Hit the Run button () and run the analysis.

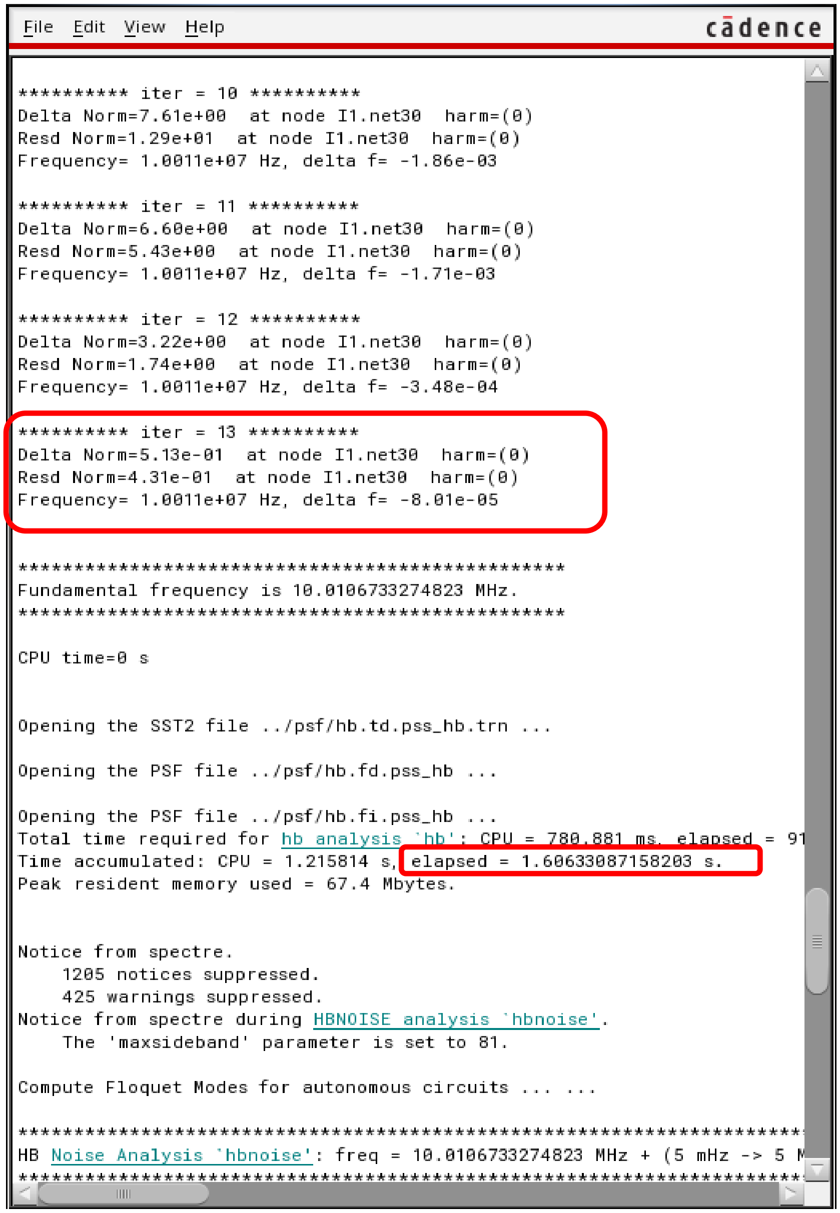
The noise result is almost identical.

Figure 54 - PM noise plot of Crystal Oscillator (with tstab set and when Number of Harmonics is set to 81)



87. Now, investigate the effect on the harmonic balance (hb) analysis in the Spectre output window. In this example, the number of iterations went from 51 to 13, and the elapsed time went from 3.73 seconds to 1.61 seconds. It is worth considering using this strategy because the convergence and runtime might improve.

Figure 55 - Analyzing Spectre output log file (spectre.out) – hb analysis simulation output (Observing the total number of iterations and the elapsed time for hb simulation)



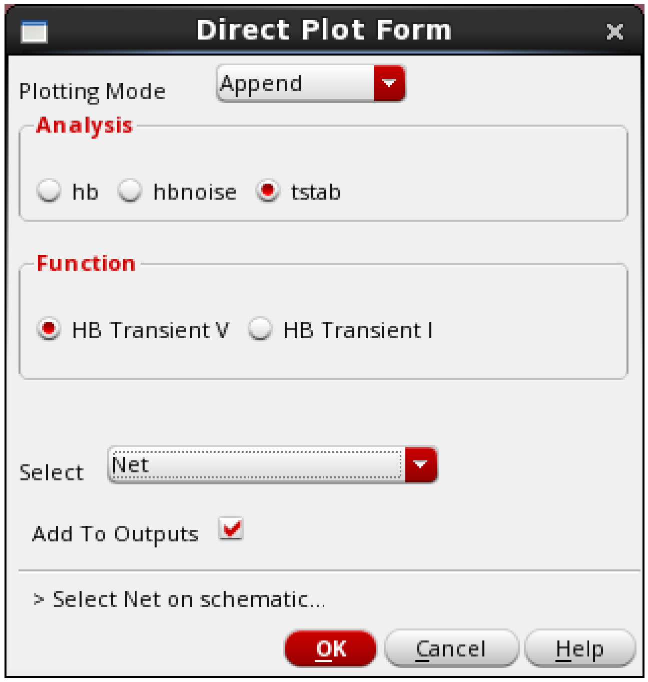
```
File  Edit  View  Help                                          cādence

********** iter = 10 **********
Delta Norm=7.61e+00  at node I1.net30  harm=(0)
Resd Norm=1.29e+01  at node I1.net30  harm=(0)
Frequency= 1.0011e+07 Hz, delta f= -1.86e-03

********** iter = 11 **********
Delta Norm=6.60e+00  at node I1.net30  harm=(0)
Resd Norm=5.43e+00  at node I1.net30  harm=(0)
Frequency= 1.0011e+07 Hz, delta f= -1.71e-03

********** iter = 12 **********
Delta Norm=3.22e+00  at node I1.net30  harm=(0)
Resd Norm=1.74e+00  at node I1.net30  harm=(0)
Frequency= 1.0011e+07 Hz, delta f= -3.48e-04

********** iter = 13 **********
Delta Norm=5.13e-01  at node I1.net30  harm=(0)
Resd Norm=4.31e-01  at node I1.net30  harm=(0)
Frequency= 1.0011e+07 Hz, delta f= -8.01e-05

**************************************************
Fundamental frequency is 10.0106733274823 MHz.
**************************************************

CPU time=0 s


Opening the SST2 file ../psf/hb.td.pss_hb.trn ...

Opening the PSF file ../psf/hb.fd.pss_hb ...

Opening the PSF file ../psf/hb.fi.pss_hb ...
Total time required for hb analysis `hb': CPU = 780.881 ms, elapsed = 91
Time accumulated: CPU = 1.215814 s, elapsed = 1.60633087158203 s.
Peak resident memory used = 67.4 Mbytes.


Notice from spectre.
    1205 notices suppressed.
    425 warnings suppressed.
Notice from spectre during HBNOISE analysis `hbnoise'.
    The 'maxsideband' parameter is set to 81.

Compute Floquet Modes for autonomous circuits ... ...

************************************************************************
HB Noise Analysis `hbnoise': freq = 10.0106733274823 MHz + (5 mHz -> 5 M
************************************************************************
```

88. Close the ViVA XL window.

89. Plot the Initial Transient Simulation results waveform.

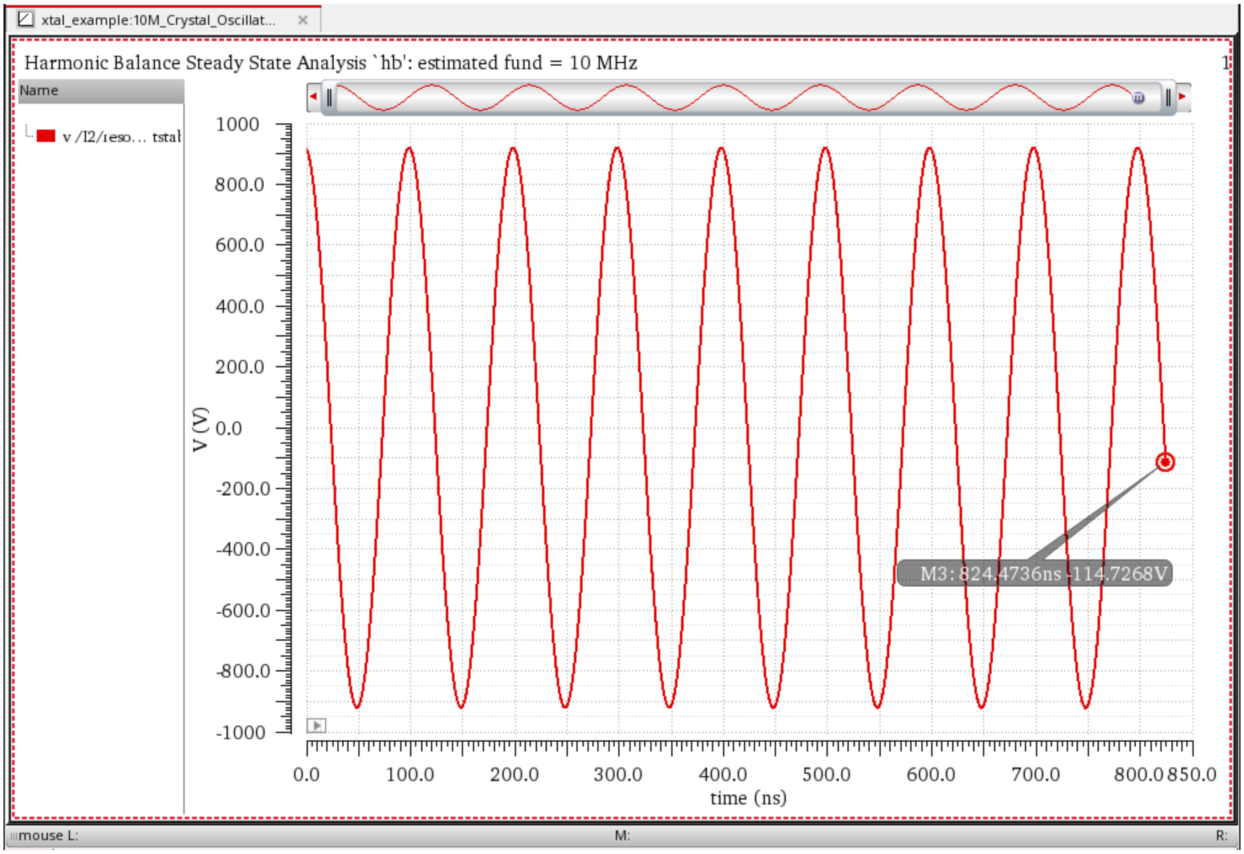
    a. Open the **Direct Plot Form** and select tstab results by selecting **tstab** as **Analysis**.

Figure 56 - Direct Plot Form – Plotting Initial Transient Simulation results



b. Select the **resonator** net from the schematic. The net is plotted below.

Figure 57 – Plotting the Initial Transient Simulation (tstab) results of resonator net in ViVA window



Why did Spectre actually run to 0.825usec when tstab was set to 0.325usec? First, tstab is run. This is 0.325usec. Then, one period of the frequency specified in the **Choosing Analyses** form is run. In this case, the frequency is set to 10M and that has a period of 0.1usec. Next, four periods of this frequency are run, and Spectre looks for a frequency divided output signal at all the nodes in the circuit. This adds the last 0.4usec to the waveform. In this example, there is no frequency division; so, Spectre then goes on to solve for one period of the oscillator waveform.

90. Click **OK** to close the **Direct Plot Form** and close the ViVA XL window.

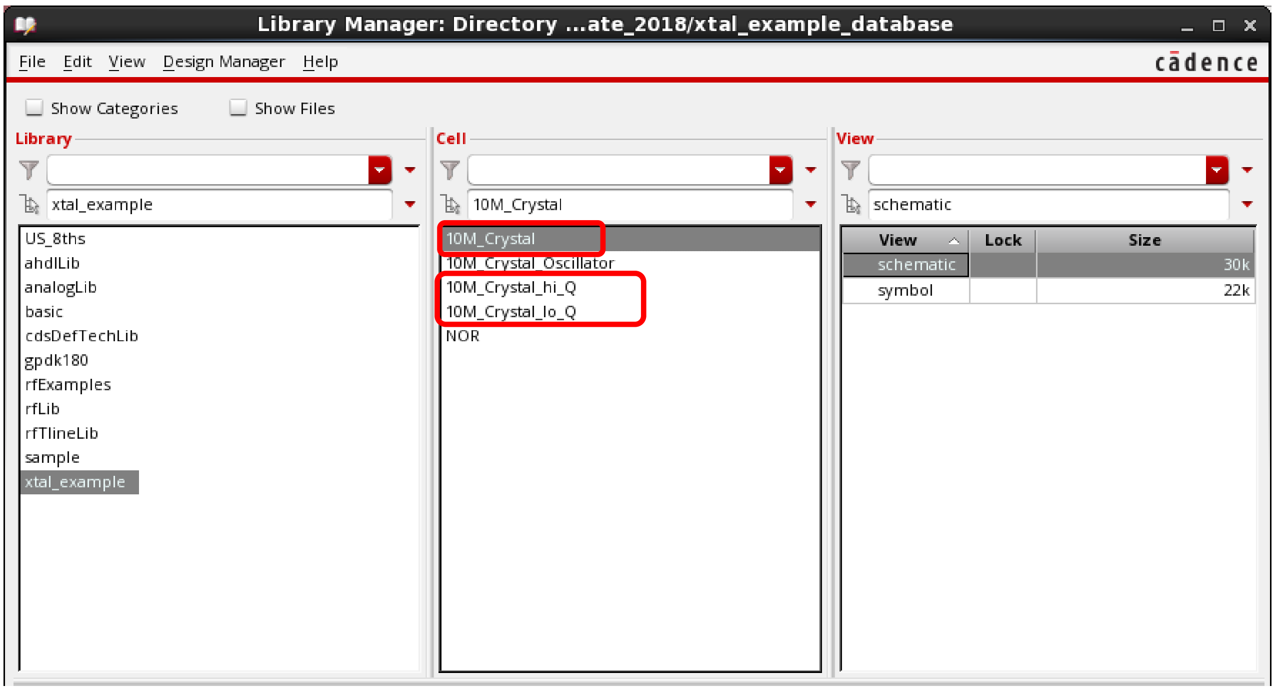
91. Deselect the hb tstab portion result of the I2.resonator net in the outputs section.

Investigating the Effects of Resonator Q on the Oscillator Phase Noise

92. From the Library Manager, open the schematic view of the following cells of the **`xtal_example`** library:

- 10M_Crystal
- 10M_Crystal_hi_Q
- 10M_Crystal_lo_Q

Figure 58 – Opening schematic view of 10M_Crystal, 10M_Crystal_hi_Q, and 10M_Crystal_lo_Q cells of xtal_example library from Library Manager



The circuits differ in that the value of the resonator inductors and capacitors vary by factors of 10 and retain the same resonant frequency.

Figure 59 – Schematic of 10M_Cystal Oscillator (l=46.055m C=5.5f)

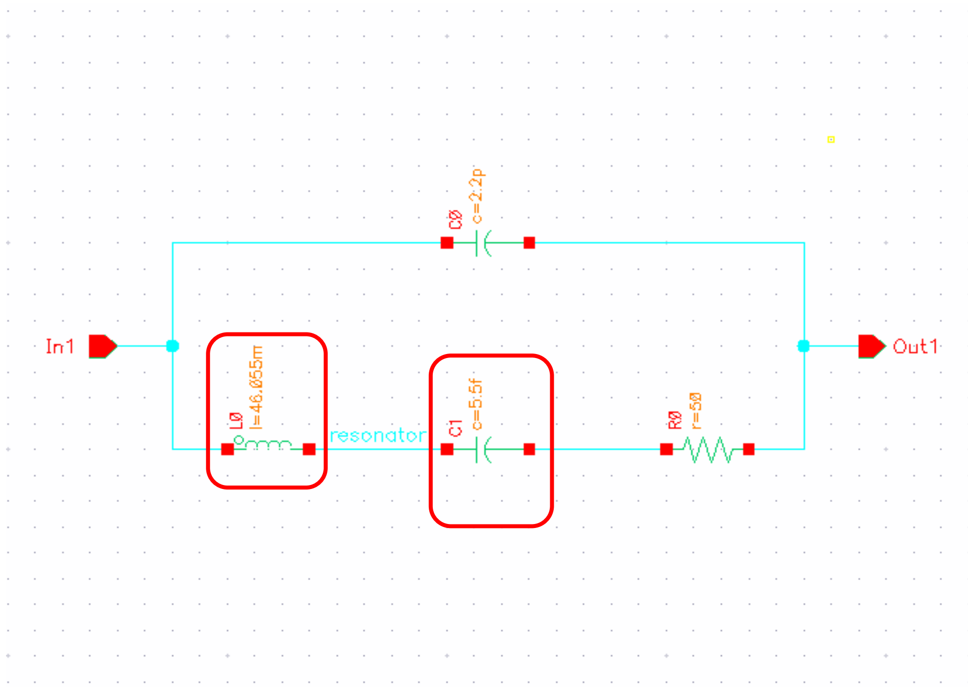


Figure 60 - Schematic of 10M_Cystal_hi_Q Oscillator (l=460.055m C=550a)

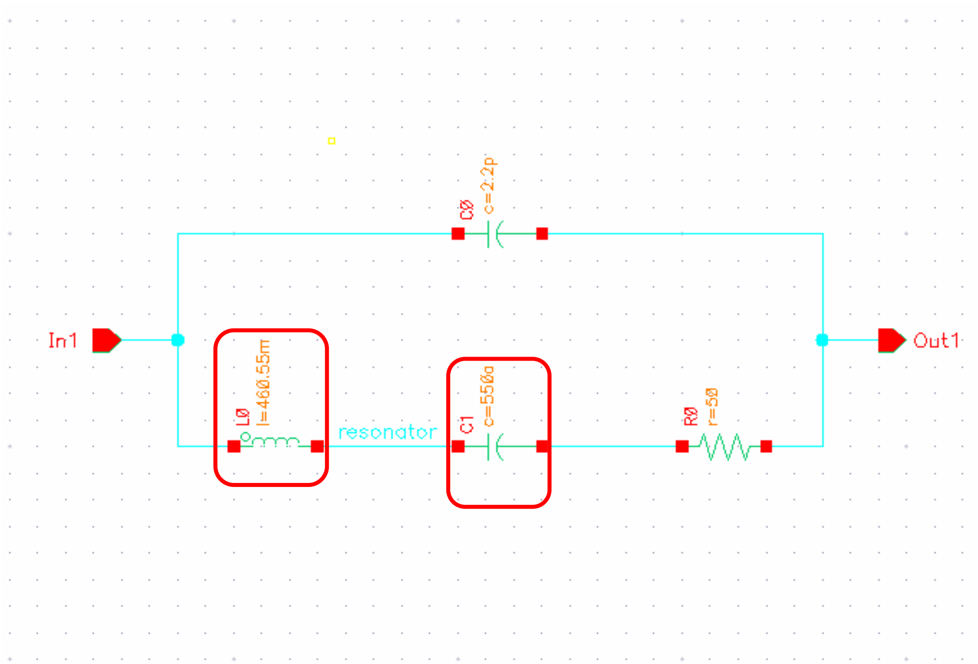
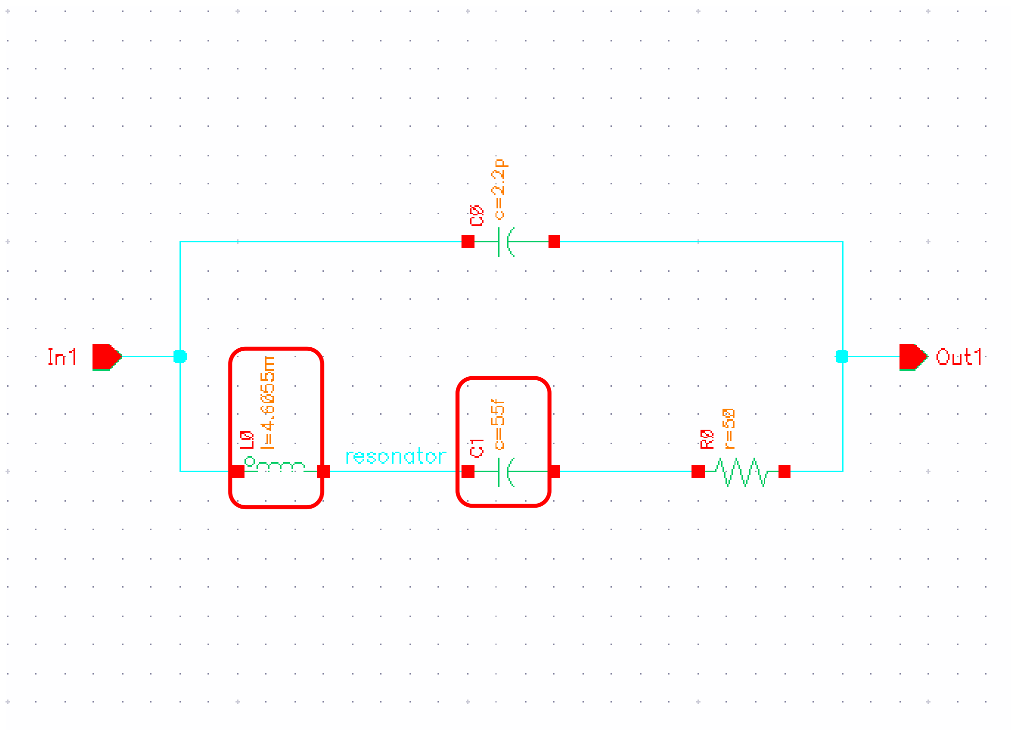


Figure 61 - Schematic of 10M_Cystal_lo_Q Oscillator (l=4.6055m C=55f)



93. After reviewing, close all the schematic windows (which are mentioned in Step 92).

Plotting the PM Noise from the 10M_Crystal Schematic Cell

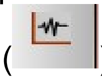
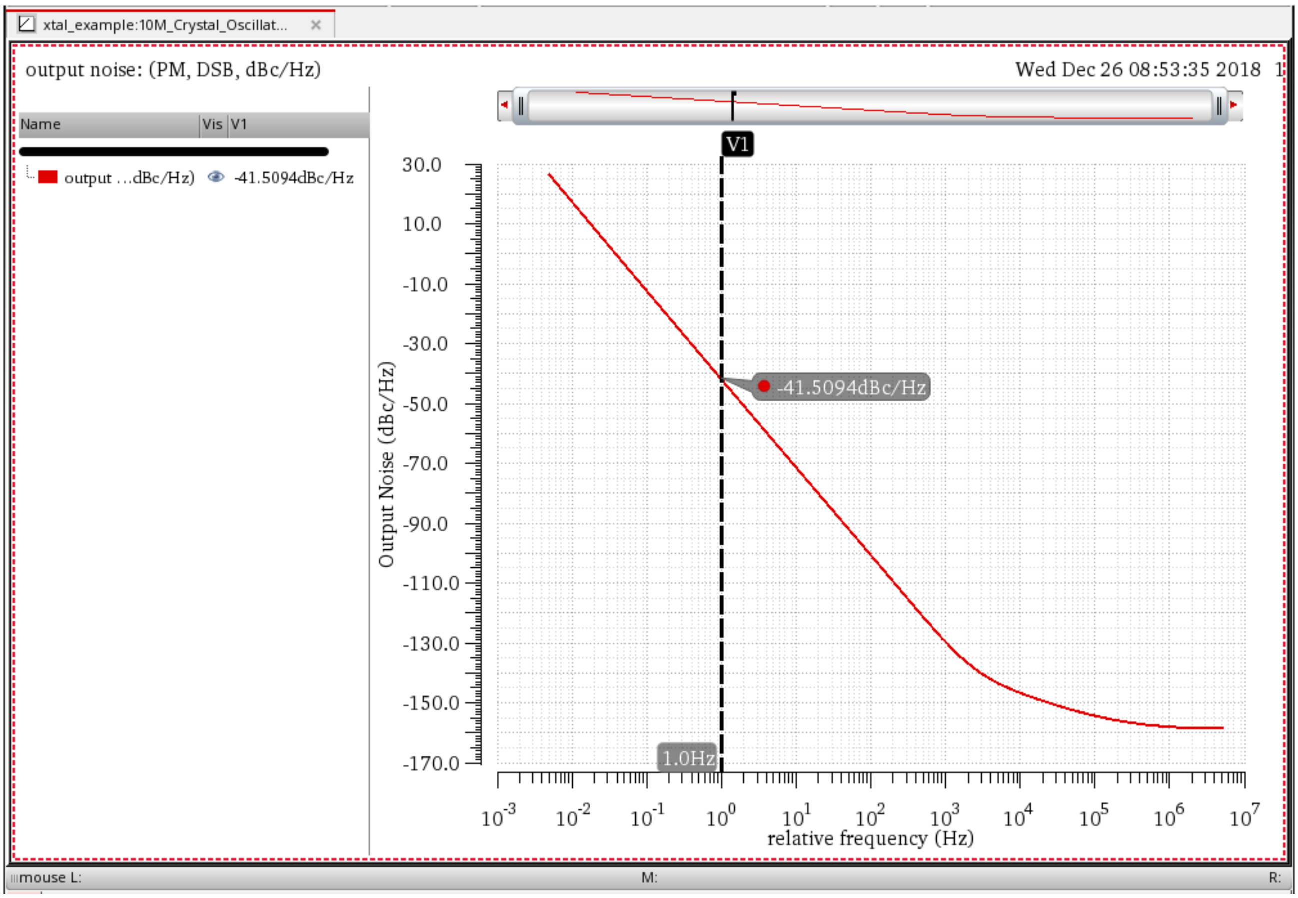
94. From the ADE Explorer window, plot the PM (DSB) noise result by clicking the Plot Outputs icon (). This is the result with the medium Q crystal (that is, from 10M_Crystal Oscillator).

Figure 62 - PM noise plot of medium Q crystal (that is, 10M_Cystal) Oscillator

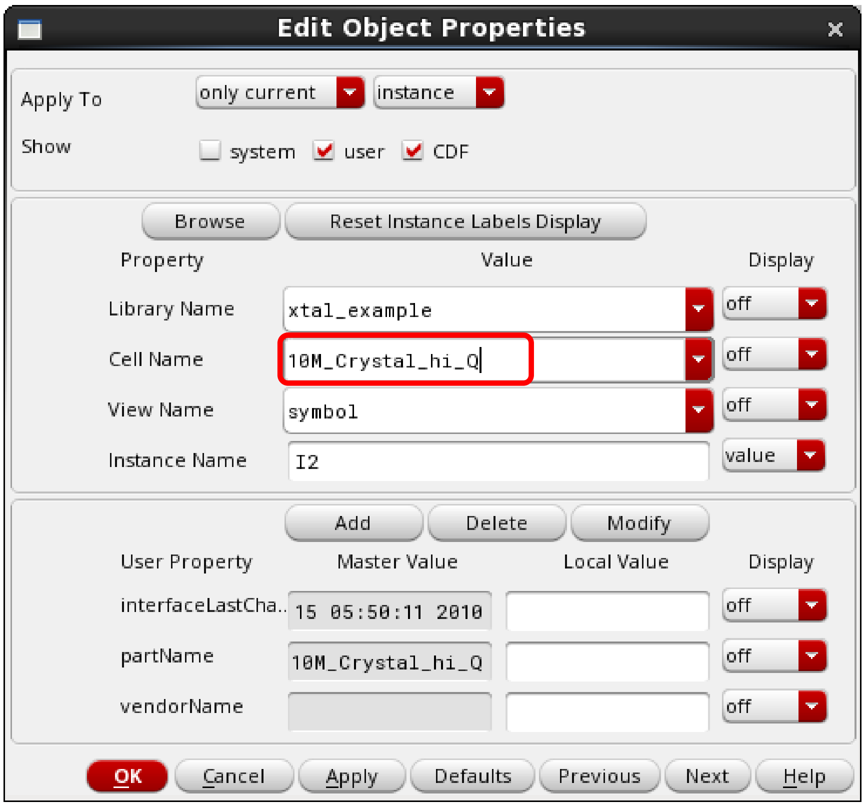


Plotting the PM Noise from the 10M_Crystal_hi_Q Schematic Cell

95. In the 10M_Crystal_Oscillator schematic window (consisting of the 10M_Crystal cell) which is already opened, select the crystal symbol and change the cell to **10M_Crystal_hi_Q**.

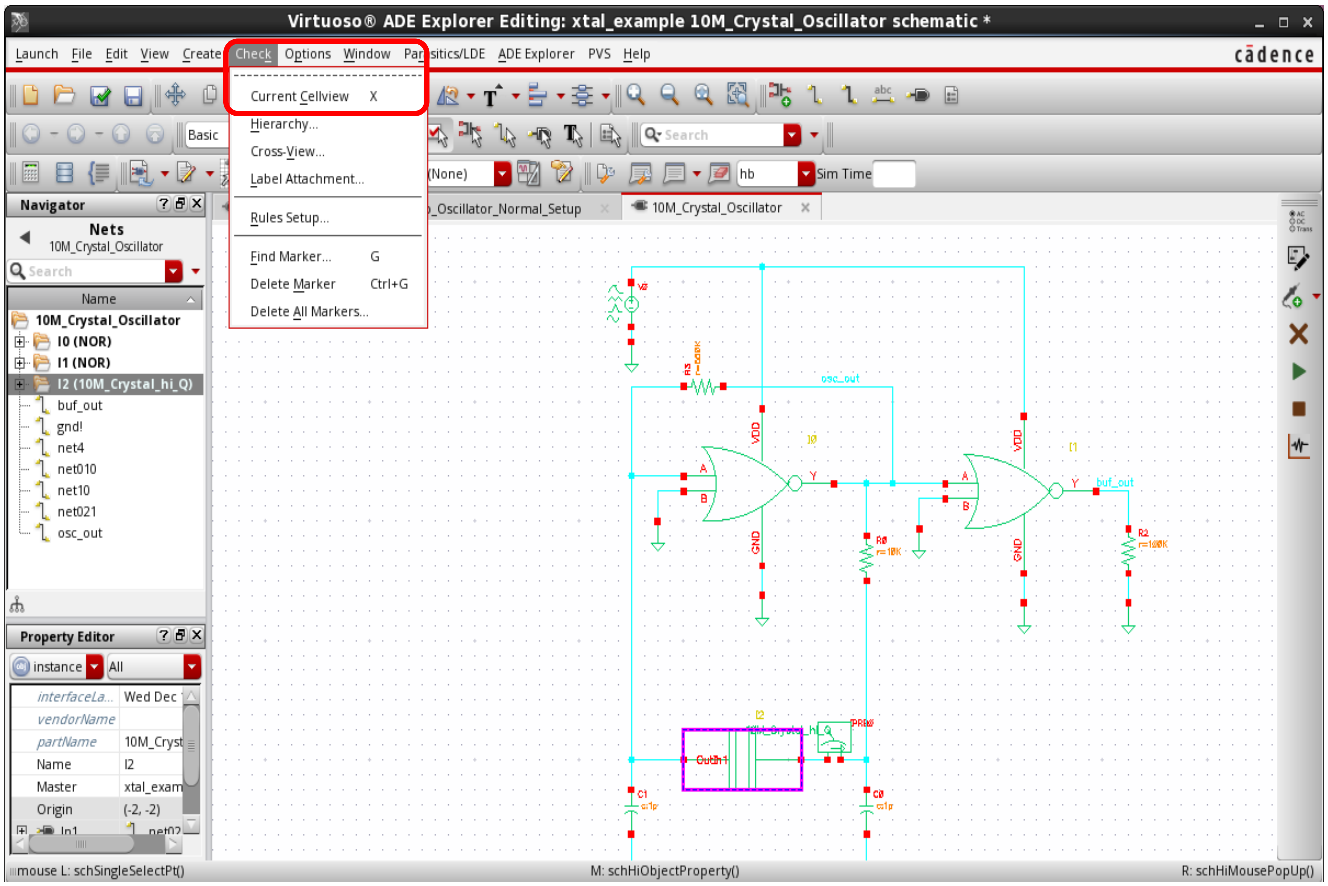
96. Click **OK** to close the **Edit Object Properties** form.

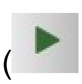
Figure 63 – Changing cell name to 10M_Crystal_hi_Q in Edit Object Properties form



97. In the schematic, select **Check > Current Cellview**. This will update the circuit to the new configuration without saving it to the disk.

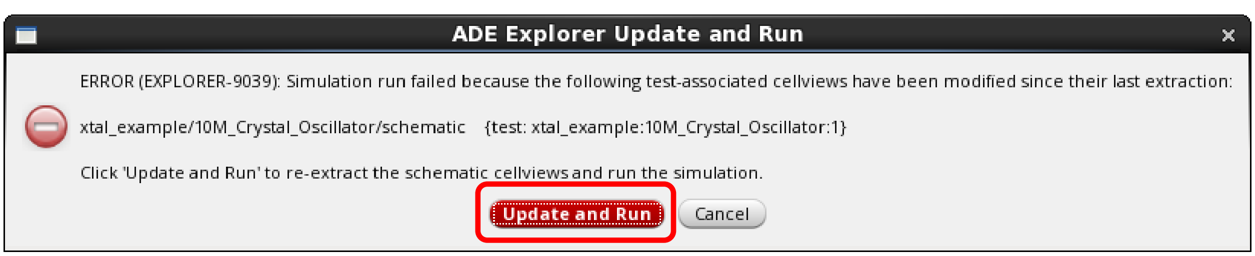
Figure 64 - Check Current Cellview



98. Click on the **Run Simulation** icon () from the **10M_Cystal_Oscillator** schematic tab and rerun the simulation. (You can also rerun the simulation from the **maestro_Oscillator_Normal_Setup** tab).

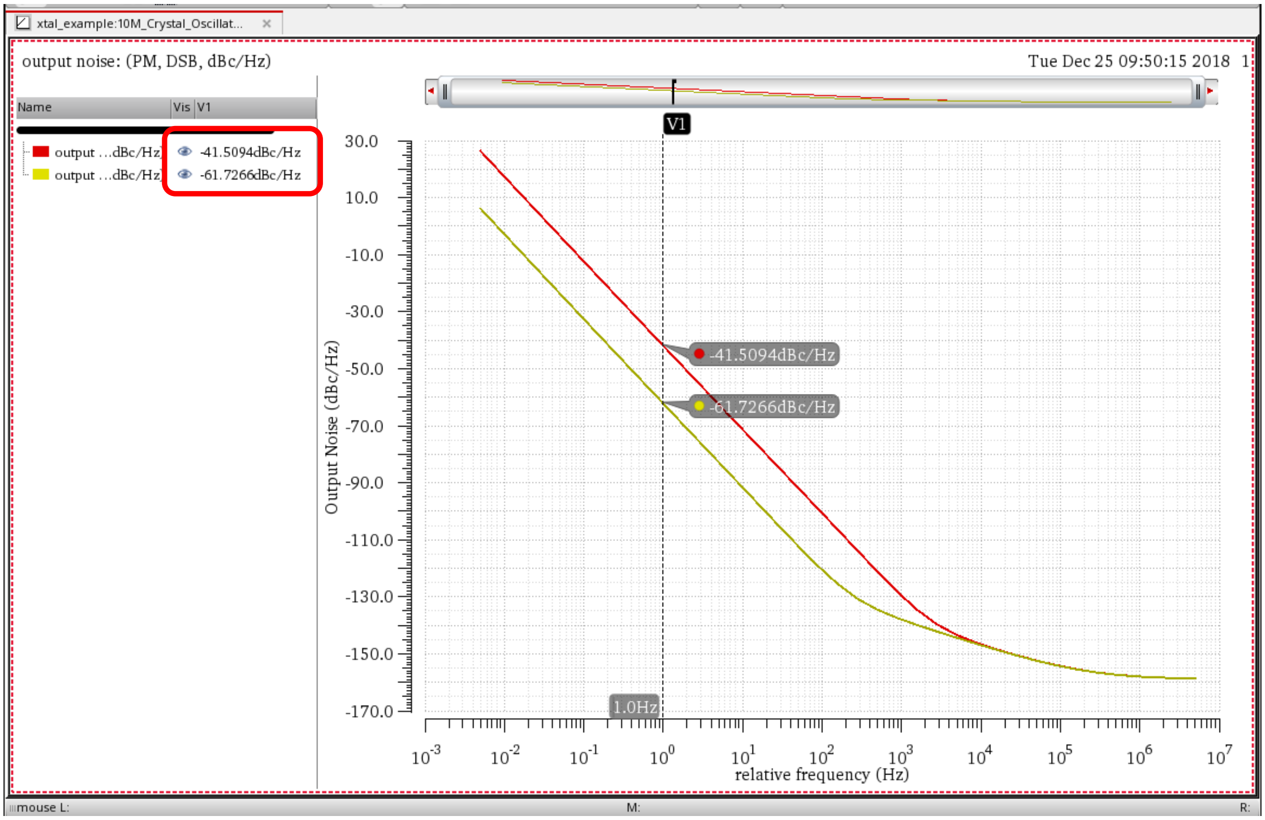
99. In the dialog box which appears, click on **Update and Run** as shown in Fig 65.

Figure 65 - ADE Explorer Update and Run dialog box



The simulation finishes, and the noise result plots after the run completes.

Figure 66 - PM noise plot of medium Q crystal and hi Q crystal (that is, 10M_Cystal and 10M_Crystal_hi_Q) Oscillators



The result is as expected. With a factor of 10 change in Q, the noise should change by about 20dB.

Plotting the PM Noise from the 10M_Crystal_lo_Q Schematic Cell

100. In the schematic window of the 10M_Crystal cell, which is already opened, select the crystal symbol and change the cell to **10M_Crystal_lo_Q**.

101. Click **OK** to close the **Edit Object Properties** form.

102. As shown above in Fig 64, in the schematic, select **Check > Current Cellview**. This will update the circuit to the new configuration without saving it to the disk.

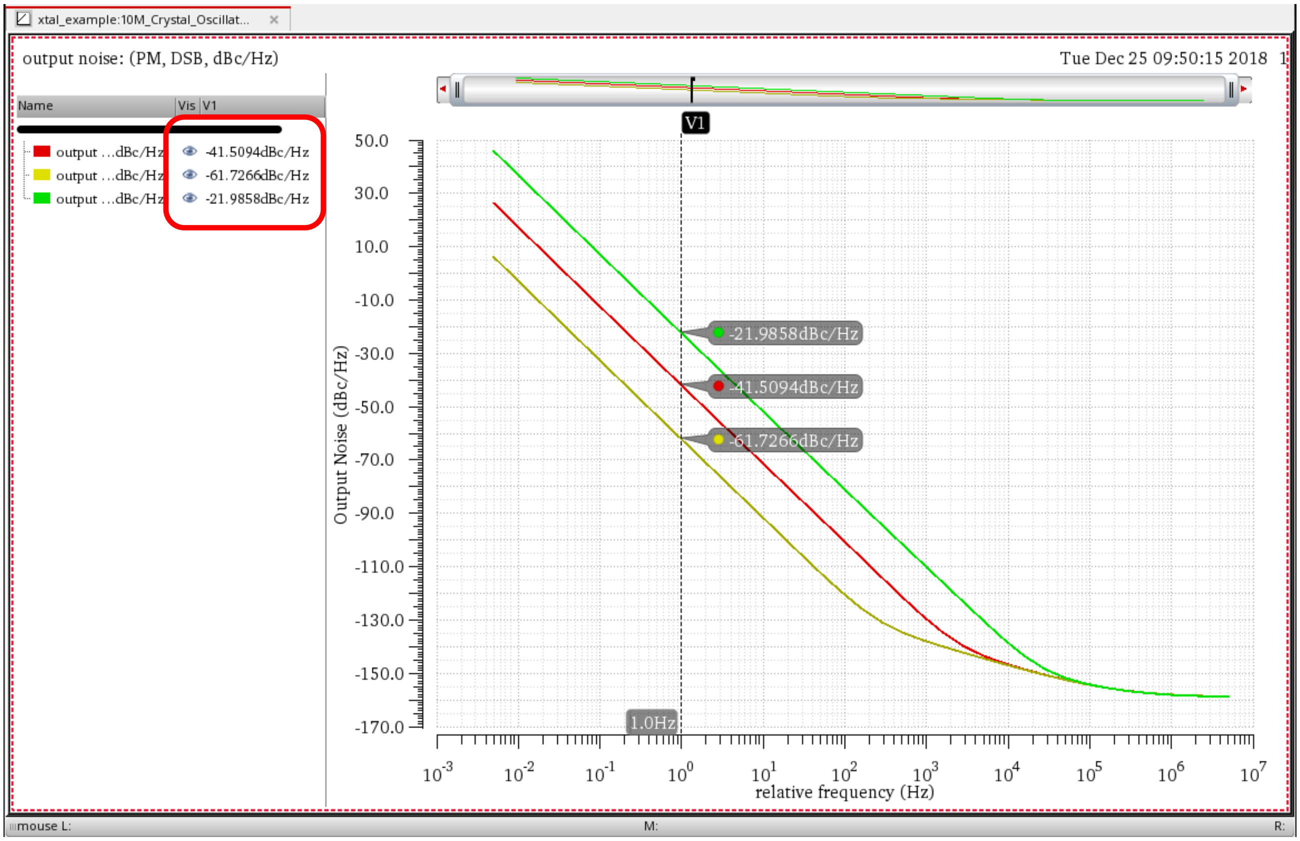
103. Click on the **Run Simulation** icon () from the **10M_Cystal_Oscillator** schematic tab and rerun the simulation. (You can also rerun the simulation from the **maestro_Oscillator_Normal_Setup** tab).

104. In the dialog box which appears, click on **Update and Run** as shown in Fig 67.

Figure 67 - ADE Explorer Update and Run dialog box

The simulation finishes, and the noise result plots after the run completes.

Figure 68 - PM noise plot of medium Q crystal, high Q crystal, and low Q crystal (that is, 10M_Cystal, 10M_Crystal_hi_Q, and 10M_Crystal_lo_Q) Oscillators

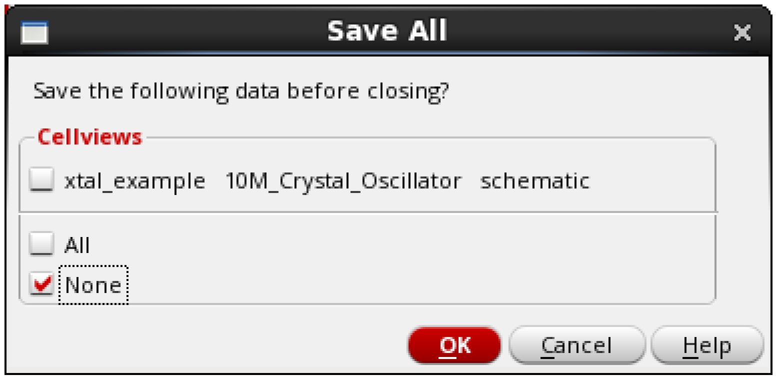


Now, the noise comes up almost 20dB.

This concludes the Crystal Oscillator RAK.

105. Quit from the Virtuoso session. When asked to save the schematic view, select **None** and click **OK**. This will allow you to preserve the integrity of the original database.

Figure 69 - Select None when asked to save the 10M_Crystal_Oscillator schematic view



Summary

The PM noise results of the Crystal Oscillator varies with the Q of the oscillator, as expected. The PM noise is higher for low Q crystal and reduces as the Q-factor of the crystal increases.

References

1. Oscillator Noise Analysis in SpectreRF Application Note
2. Spectre Circuit Simulator and Accelerated Parallel Simulator RF Analysis in ADE Explorer User Guide
3. Virtuoso Visualization and Analysis XL User Guide
4. Virtuoso Analog Design Environment Explorer User Guide
5. Spectre Classic Simulator, Spectre Accelerated Parallel Simulator (APS), and Spectre Extensive Partitioning Simulator (XPS) User Guide

Support

Cadence Support Portal provides access to support resources, including an extensive knowledge base, access to software updates for Cadence products, and the ability to interact with Cadence Customer Support. Visit https://support.cadence.com.

Feedback

Email comments, questions, and suggestions to content_feedback@cadence.com.