Departamento de Electrónica, Sistemas e Informática

Maestría en Diseño Electrónico

ITESO

Simulation of 32.768 KHz Oscillator using external Quartz Crystal Optimized Equivalent Circuit

Final Assignment Report

Author: José Gerardo Villalobos Rosales

Instructor: Dr. J. Ernesto Rayas Sánchez

Class: Computer Aided Design

May/2005

ITESO

Abstract

This Application report is intended to demonstrate the use of combined design tools MatLab – WinSpice to simulate electronic circuits. The use of Crystal equivalent circuit optimized values in the real circuit may not be practical due to the fact some of the Crystal parameters are related with temperature and some other to the quartz properties. This Application has been developed considering the ambient temperature of 25 ºC.

Introduction

This Report provides a method to simulate an accurate clock generator with few components. The circuit has two basic stages: The Crystal equivalent Circuit and the Resonator Circuit. The use of Matlab calling Winspice provides the accurate values for internal Crystal parameters through the functions Max, Optimset and Fminsearch. The electrical model used for quartz crystal equivalent circuit is in **Figure 1.** L1 is the quartz internal mass. C1 is the Mechanical elasticity and R1 is the internal resistance dissipated in heat. The dielectric of the quartz physically separates the two terminals Co is the load capacitance normally generated by the dielectric and component terminals. Together, these components are a reasonably accurate electrical model for the behavior of the crystal. Values for these component models are usually available from the crystal manufacturer. The initial values for equivalent circuit in this applications are given as follow:R1→ (Reference book) 11.2 KΩ, Co→ (Reference book) 1.84pF.
L1 and C1 →

$$f_c = \frac{1}{2\pi\sqrt{LC}} \qquad\qquad Q = \frac{f_c}{BW} = \frac{2\pi f_c L}{R}$$
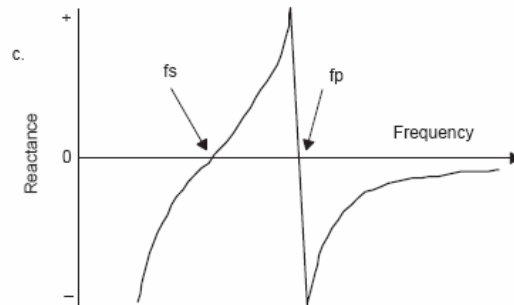
Where: BW = Band Width (1Hz)
       fc = Resonance frequency (32.768KHz)
       Q = Quality Factor

From formulation above we have for 32.768KHz: L1 = 1785H, C1 = 1.3234e-14F
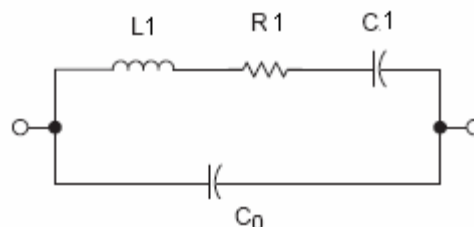
Figure 1.- Typical quartz crystal equivalent circuit

## Crystal   Behavior

Figure 2.- *Crystal reactance response*

Like LC-resonator, crystals have an impedance minimum at a frequency determined by L1 and C1. This is the series-resonant frequency (fs). The presence of C0 also introduces an impedance maximum at a frequency determined by L1 and C0. This frequency is the parallel-resonant frequency (fp). ***Figure 2*** shows the reactance of the crystal with frequency. The reactance (and consequently the phase) is zero at the series-resonant frequency (fs) because at this frequency the reactances of L1 and C1 cancel each other. At this frequency, the total impedance of the crystal is equal to the resistance R1.
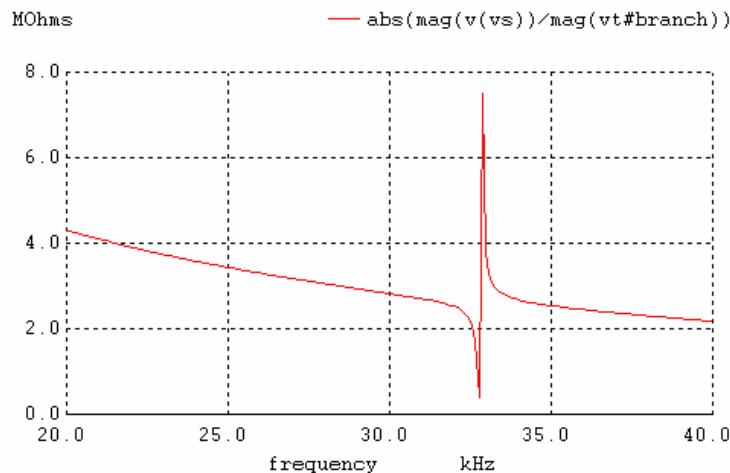
Figure 3.- *WinSpice Crystal impedance response for equivalent circuit with initial values*

Crystal Equivalent Circuit Optimization

Running WinSpice from Matlab from 32700Hz to 32800Hz, the crystal response for minimum impedance is about 32746Hz. At this point, we are using the estimated values obtained from the formulas in the introduction of this document. The response very close (just 22Hz below) of the nominal crystal resonance value.
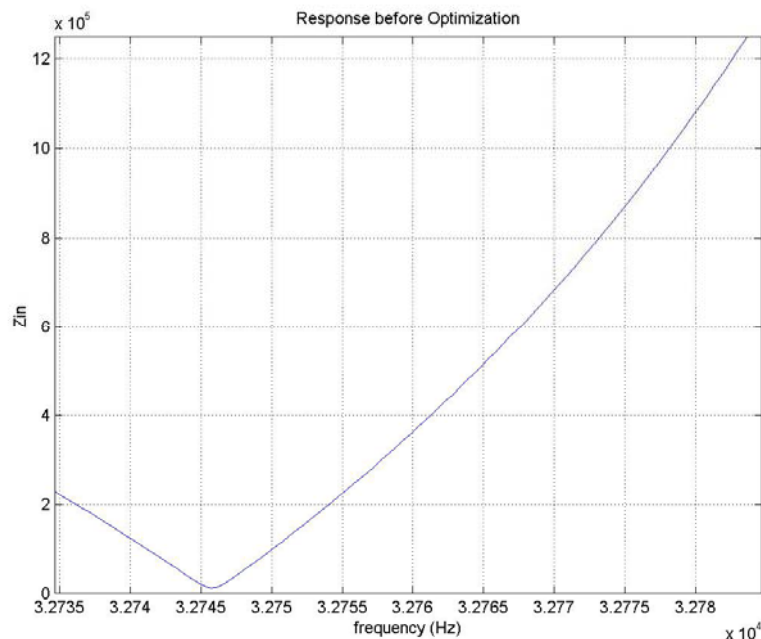
Figure 4.- *Quartz Crystal Equivalent Circuit response running WinSpice From Matlab*

In order to have a crystal Equivalent Circuit Response closer to the nominal value the Objective Functions are formulated as follow:

```
%Specifications
%|Zin| <= 11.2e3 for 32767.5Hz <= f <= 32768.5Hz
%|Zin| >= 500e3 for f<= 32767.5Hz and f>=32768.5Hz
```

```
Zin_min = 500e3;
Zin_max = 11.2e3;
%Calculate Filter response and Error vector in  first band
%Zin  from 32700 to 32767Hz = 134 vector elements (32700Hz to 32800Hz =
200 readings)
for i = 1: 134  ; e1 = 1 - Zin(i)/Zin_min ;end
%Calculate Filter response and Error vector in  Rejection band
% e2 for 32767.5Hz to 32768.5Hz are 3 readings Zin elements(135 to 137)
  for i = 135: 137; e2 = Zin(i)/Zin_max -1; end
%Calculate Filter response and Error vector in  first band
%Zin  from 32768.5Hz to 32800Hz = 63 vector elements (32700Hz to
32800Hz = 200 readings)
for i = 138: 200;e3 = 1 - Zin(i)/Zin_min;end
u = max([e1;e2;e3])
```
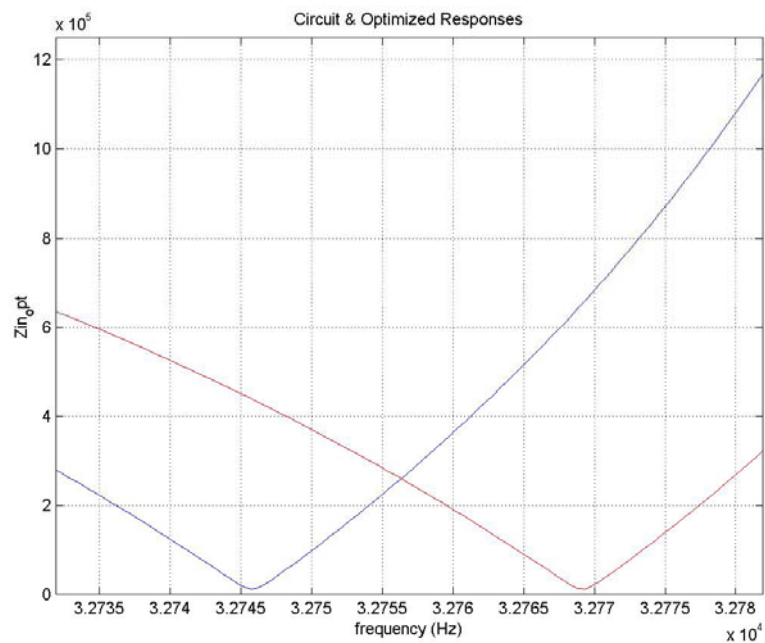

Figure 5.- *Optimized and initial responses of quartz crystal equivalent*
Circuit. Optimized response is closer to nominal value

Optimized Values for R1, L1 and C1 are close to the initial values due to the fact the initial response is just 20Hz below of nominal resonance frequency.
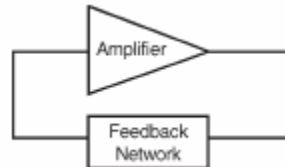L1opt =1.787860225698107e+003
C1opt =1.319448159138599e-014
R1opt =1.144826457397890e+004

Oscillator Simulation

Crystal controlled oscillators may be considered as consisting of an amplifier and a feedback network that selects a part of the amplifier output and returns it to the amplifier input. A generalized depiction of such a circuit is shown below.


Figure 5.- *Generalized form of Crystal Controller Oscillator*

This circuit used in this application is a single inverter, with two capacitors in the feedback loop. These capacitors comprise the load capacitance and together with the crystal unit, establish the frequency at which the oscillator will operate. As the value of the load capacitance is changed, so is the output frequency of the oscillator. ***Figure 6*** shows single inverter oscillator with quartz crystal and the same oscillator using the quartz crystal equivalent circuit. The equivalent circuit with Optimized values of L1 , C1 and R1 obtained in previous section was used for Spice Simulation.

(a)   *(b)*
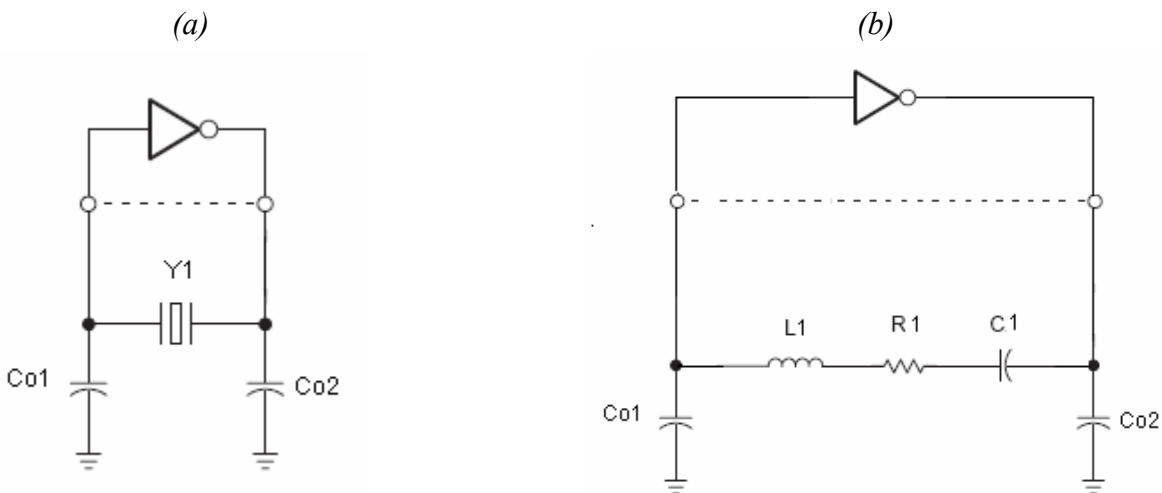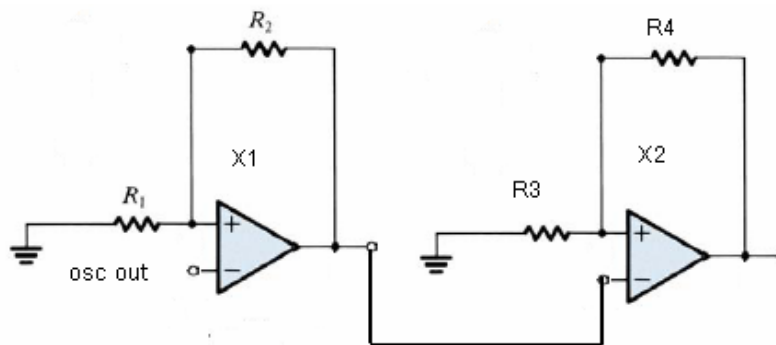

Figure 6.- *Single inverter quartz crystal oscillator (a) . Single inverter quartz crystal equivalent circuit (b)*

 Channel P and Channel N MOSFET transistors provide the required circuit to perform the inverter section of Simulation.
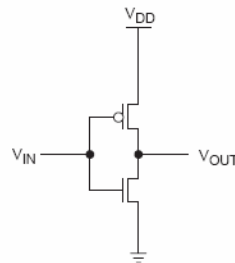
Figure 7.- *Single inverter complementary Channel P and Channel N MOSFET transistor*

Figure 8.- *Circuit Conditioner simulated to amplify the oscillator's signal to Levels of 0 to 5v.R1=10kΩ, R2=1MEGΩ, R3 =100Ω and R4 = 600Ω The model is the QuasiIdealOpamp.*
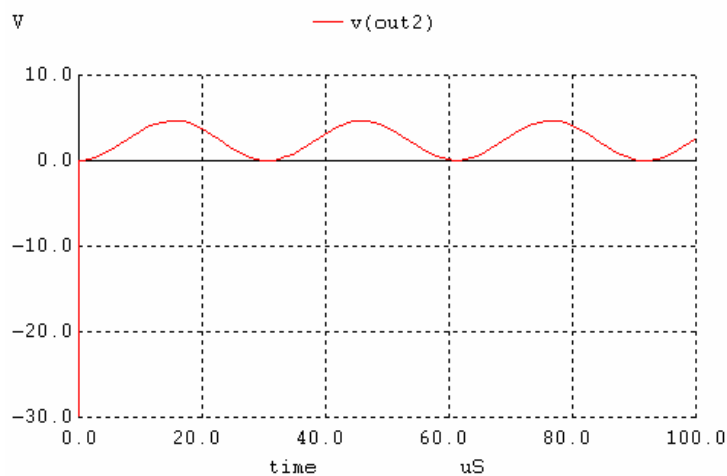
Figure 9.- *X2 Amplifier output in WinSpice. Expected period is 30.5uS.*

Program Code

```matlab
% crystal_control
L1 = 1785;
C1 = 1.3234e-14;
R1 = 11.2e3;
Cp = 1.84e-12;
InitFreq = 32.700e3;
FinalFreq =32.800e3;
Xo = [L1 C1 R1];
[f,Zin] = crystal_SPICE(Xo, InitFreq,FinalFreq);
plot(f,Zin)
title ('Response before Optimization')
xlabel ('frequency (Hz)');
ylabel ('Zin');
grid on
hold on
options = optimset('MaxIter',1000,'TolX',1e-3);
[Xopt,FVAL] = fminsearch('objective_crystal_SPICE',Xo,options);
[f,Zin] = crystal_SPICE(Xopt,InitFreq,FinalFreq);
plot(f,Zin,'r')
title ('Circuit & Optimized Responses')
xlabel('frequency (Hz)');
ylabel('Zin_opt');
grid on
hold on
L1opt = Xopt(1)
C1opt = Xopt(2)
R1opt = Xopt(3)
% rsonator2_SPICE
%Calculating spice response
function u = objective_crystal_SPICE(X)
%Specifications
%|Zin| <= 11.2e3 for 32767.5Hz <= f <= 32768.5Hz
%|Zin| >= 500e3 for f<= 32767.5Hz and f>=32768.5Hz
Zin_min = 500e3;
Zin_max = 11.2e3;
FL = 32.700e3;
FH = 32.800e3;
InitFreq = FL;
FinalFreq = FH;
[f,Zin] = crystal_SPICE(X,InitFreq,FinalFreq);% llega bien
%Calculate Filter response and Error vector in  first band
%Zin  from 32700 to 32767Hz = 134 vector elements (32700Hz to 32800Hz =
200 readings)
for i = 1: 134
   e1 = 1 - Zin(i)/Zin_min
end
%Calculate Filter response and Error vector in  Rejection band
% e2 for 32767.5Hz to 32768.5Hz are 3 readings Zin elements(135 to 137)
  for i = 135: 137
     e2 = Zin(i)/Zin_max -1
```

```matlab
    end


%Calculate Filter response and Error vector in  first band
%Zin  from 32768.5Hz to 32800Hz = 63 vector elements (32700Hz to
32800Hz = 200 readings)
for i = 138: 200
e3 = 1 - Zin(i)/Zin_min
end
 u = max([e1;e2;e3])
% Oscillator_SPICE
% This function provides input data to oscillator equivalent circuit
% simulated with WinSpice3, ask to calculate the corresponding
response,
% and returns the response.
% Usage: [p,Vo] = Oscillator_SPICE(X,InitPeriod,FinalPeriod)
% X = [L1 C1 R1]
% InitPeriod: Initial Period for TRAN analysis (zero)
% FinalPeriod: Final Period for TRAN analysis
% Vo: Output volyage
% p: Period for Simulation
%Cp: Terminals capacitance
function [p,Vout] = oscillator_SPICE(X,TStep,TStop)
Title = 'Oscillator';
%netlist block
nb{1} = '* Netlist';
nb{2} = 'vdd pwr  0   DC  5v';
nb{3} = 'vss  vref   0   DC  -8.5v';
nb{4} = 'M1   out  in pwr  pwr          pmos    L=10u  W=50u';
nb{5} ='M2   out   in  0  0        nmos    L=10u W=100u';
nb{6} = ['L1   in   1 '  num2str(X(1))];
nb{7} = ['C1    2  1 '   num2str(X(2))];
nb{8} = ['R1    out  2 ' num2str(X(3))];
nb{9} = 'Cp1    in  0  1.84e-12';
nb{10} = 'Cp2    out 0  1.84e-12';
nb{11} = 'Rin1  0   minus1   10k';
nb{12} = 'Rf1   minus1   out1    1MEG';
nb{13} = 'Rterm1  out1  vref  1';
nb{14} = 'Rin2  0   minus2   100';
nb{15} = 'Rf2   minus2   out2    600';
nb{16} = 'X1    minus1   in    out1   QuasiIdealOpamp';
nb{17} = 'X2    minus2   out1   out2   QuasiIdealOpamp ';
nb{18} = '.model nMOS nmos level=2 vto=+.8v kp=0.00006 lambda=0.8';
nb{19} = '.model pMOS pmos  level=2 vto=-.8v kp=0.00003 lambda=0.8';
nb{20} = '*Subircuit  for a Quasi Ideal Amp';
nb{21} = '.SUBCKT QuasiIdealOpamp N  P   OUT';
nb{22} = 'Ri   N   P   1000MEGAOHMS';
nb{23} = 'Ro I    OUT 1OHMS';
nb{24} = 'Eo I    0  P  N  1E6';
nb{25} = '.ENDS';
NetlistBlock = '';
for k = 1:25
```

```matlab
NetlistBlock = str2mat(NetlistBlock, nb{k});
end
%control block
cb{1} = '.control';
cb{2} = ['TRAN ' num2str(TStep) ' ' num2str(TStop)];
cb{3} = 'write  oscillator_vo.csv  vm(out2)';
cb{4} = 'quit';
cb{5} = '.endc';
cb{6} = '.end';
ControlBlock = '';
for k = 1:6
ControlBlock = str2mat(ControlBlock, cb{k});
end
% Save WinSpice circuit file
ckt_file = str2mat(Title,NetlistBlock,ControlBlock);
[rows,columns]=size(ckt_file);
fp = fopen('oscillator.cir','w+');
for i = 1:rows
for j = 1:columns
fprintf(fp, '%s', ckt_file(i,j) );
end
fprintf(fp,'%s\n','');
end
fclose(fp);
% Run WinSpice
% The matlab working directory must be where wspice3.exe and the
% *.cir files are located.
! wspice3 oscillator.cir
% Read WinSpice output file
FormatSpiceFile('oscillator_vo.csv', 'oscillator_vo.txt')
load oscillator_vo.txt;
p = oscillator_vo(:,1)';
Vout = oscillator_vo(:,2)';
function FormatSpiceFile(file1,file2)
fid1 = fopen(file1);
fid2 = fopen(file2,'w+');
line = fgetl(fid1); % Drop first line.
line = fgetl(fid1); % Read second line.
while line~=-1
line_length = length(line);
for i = 1:line_length
if line(i) == ',' % Replace comas by blankspaces.
line(i) = ' ';
end
fprintf(fid2, '%s', line(i));
end
fprintf(fid2,'%s\n','');
line = fgetl(fid1); % Read next line.
end
fclose(fid1);
fclose(fid2);
```

Conclusions

Combining MatLab and WinSpice we can have a powerful tool to obtain
Optimized values for Computer Aided Design.
Simulation provides better understanding of circuit behavior and help
to solve design issues in early design process.

Bibliography:
CAD notes , Dr. Ernesto Rayas Sánchez.
Sensores y Acondicionadores de señal , Ramón Pallas Areny
Editorial Marcombo.
Use of the TMS305C5x internal Oscillator with external Crystals
or ceramic resonators. Texas Instrument Applications report
Digital Signal Processing Products 1995. (TI.com).