Comparison of X2Y vs 0402 Capacitors for Decoupling

Recently, there has been a lot of information floating around the electronics industry regarding the “magic” of the X2Y capacitor in terms of its decoupling capabilities. The intent of this article is to describe the characteristics of this capacitor; compare it to the conventional 0603 and 0402 capacitors and provide criteria for selecting capacitors for high frequency decoupling.

Capacitor Types

Typically two terminal ceramic capacitors such as an 0603 or 0402 capacitor have been used for high frequency decoupling. In recent years a new type of ceramic capacitor named the “X2Y” has become available. The X2Y capacitor is a four terminal device that has two separate capacitors connected to a common pair of ground terminals. This capacitor has been marketed as a very low inductance capacitor that can provide superior high frequency decoupling compared to conventional ceramic capacitors.

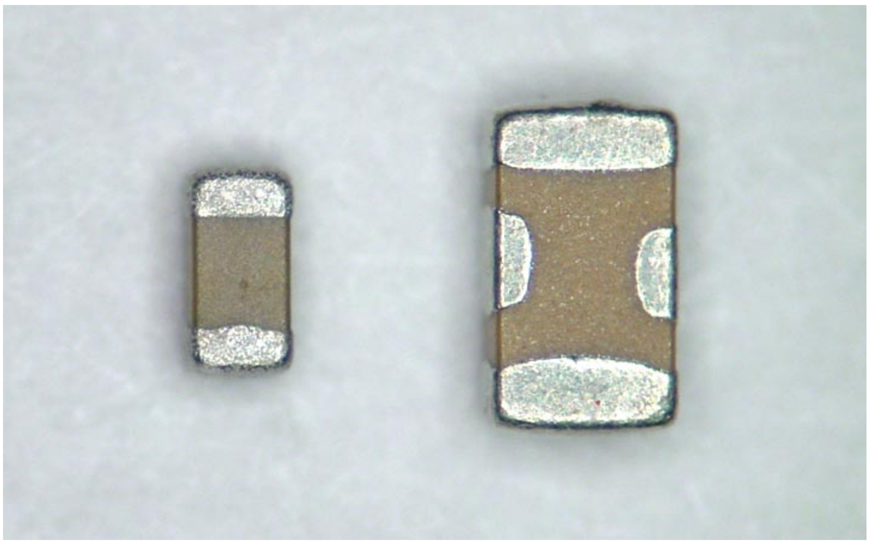

Figure 1 Photo of an 0402 and an X2Y Ceramic Capacitor

The X2Y capacitor has a case size of 0603. The two terminals in the center of each side are connected together inside the capacitor. The terminals on the two ends each connect to a separate capacitor with a common connection to the center terminals. The common center terminals are typically connected to the PCB ground. Figure 1 shows a side by side comparison of 0403 and X2Y capacitors.

High Frequency Decoupling Capacitor Requirements

The power distribution system on a PC board must provide a low impedance source over a very wide frequency range. Various size decoupling capacitors are typically mounted on the PC board for frequencies up to a few hundred MHz. At frequencies above a few hundred MHz, the parallel plate capacitance of the PC board power planes take over and provide the low impedance source.

Small capacitors are used to cover the high frequency range just below the point that requires internal PCB plane capacitance. The most important characteristic of a high frequency decoupling capacitor is the ESL (Equivalent Series Inductance). The ESL of interest is the effective inductance of the capacitor including the surface footprint and vias necessary to connect to the internal power planes of the PCB. With lower ESL, fewer decoupling capacitors (in parallel) are required to provide the low impedance at high frequencies. Figure 2 depicts the equivalent circuit of a capacitor on a PCB.

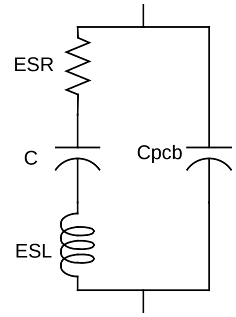

Figure 2. Equivalent Circuit of a Ceramic Chip Capacitor on a PC Board

There are three components in the capacitor shown on the left side of Figure 2. ESR (Equivalent Series Resistance) is the parasitic resistance of the metal in the plates and leads of the capacitor. ESL (Equivalent Series Inductance) is the parasitic inductance of the capacitor itself and the mounting structures such as the vias used to connect the capacitor to the planes in the PCB and the mounting pads. These parasitic components have an adverse effect on the performance of the capacitor, C.

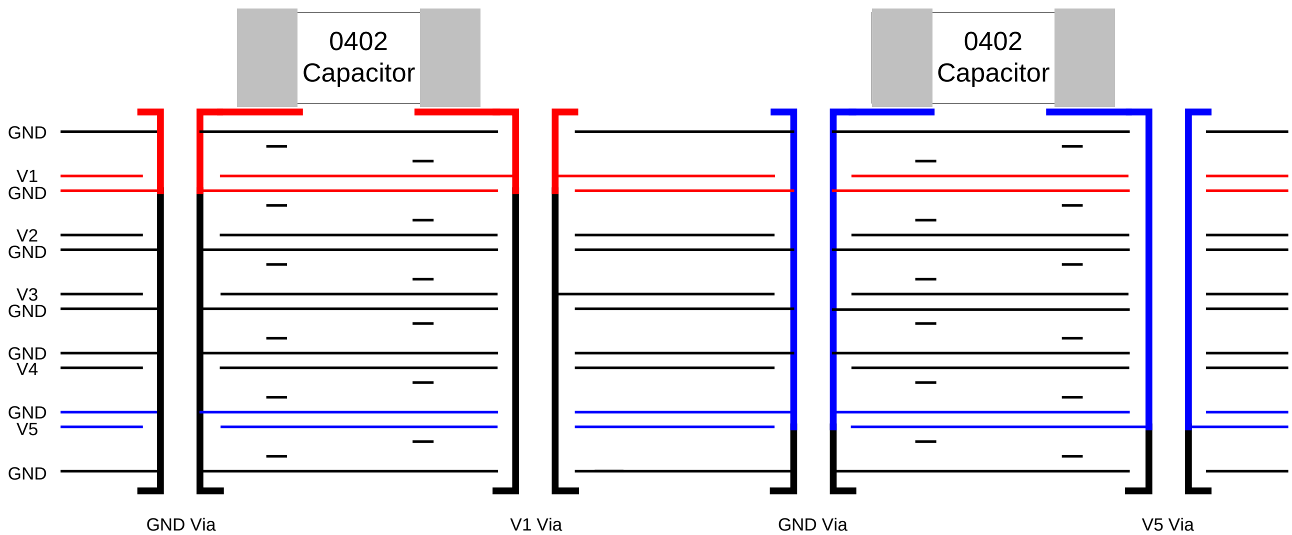

Figure 3. Cross Section Diagram of Capacitors Mounted on a 26-Layer PC Board

Both the length and the number of vias in parallel determine the ESL of a capacitor. Figure 3 shows the placement of capacitors on a PCB and the resulting effect of longer vias connecting to different power planes in the PCB. The 0402 capacitor shown on the left hand side of Figure 3 connects to the V1 power plane that is near the top surface of the PCB. This capacitor has much shorter vias and thus much lower ESL than the capacitor on the right that is connected to the V5 power plane near the bottom surface.

Capacitor PCB Footprints

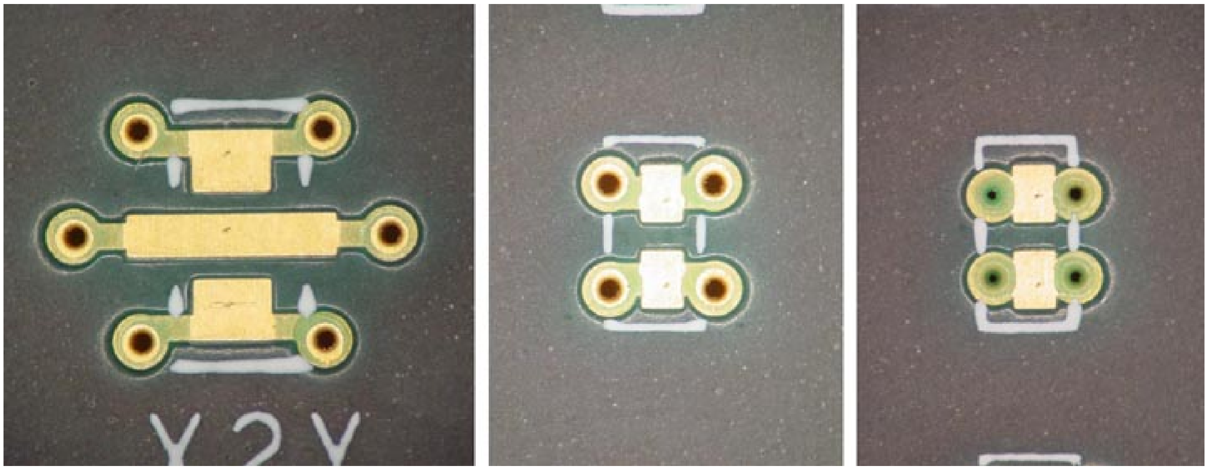

Figure.4 PCB Footprints for X2Y and 0402 Ceramic Capacitors

The design of the footprint for a capacitor must consider PCB internal signal layer routing, PCB fabrication, PCB assembly, and electrical performance. Because the X2Y capacitor has four terminals, it is a physically larger device than the two terminal 0402 capacitor and thus takes up more space on the board.

The footprints shown in Figure 4 were designed for low inductance. Multiple vias on each capacitor terminal significantly reduce the inductance compared to conventional designs with only one via per terminal.

The X2Y capacitor in on the left hand side of Figure 4. In the above photo there are two different footprints for the 0402 capacitor. The 0402 capacitor in the center of Figure 4 uses four vias on a 1.27 mm grid. The 0403 capacitor on the right in Figure 4 uses four vias on a 1.0 mm grid. Matching the capacitor via grid to the ball pitch on a large BGA package facilitates easy routing of internal signal wires. The 1.0 mm pitch produces slightly lower inductance than the 50 mill pitch footprint.

The design of the footprint requires a solder mask dam of sufficient size to keep the solder from flowing into the via during the assembly process. The X2Y and 0402 1.27mm pitch footprints have sufficient space between the edge of the capacitor terminal to the via to allow for a standard solder mask. The space on the 0402 1.00mm pitch footprint is too small to allow a standard via opening in the solder mask but solder mask can be applied with a very small opening over the via such that the mask covers the via edge and has sufficient width for proper soldering.

Test Board (Figure 5)

A small test PCB was fabricated to evaluate a variety of different capacitor types. The test PCB has the same stackup as that shown in Figure 3. It has separate capacitor footprints connected to each of the five power planes in the board in order to measure the ESL for different via lengths.

The PCB has 26 layers of 0.5oz copper planes. The epoxy glass is Isola FR406 with 2-ply 106 glass for all layers. Nominal dielectric thickness of each layer is 3-mils. The total board thickness is 98 mils. The capacitance is 2.6 nF for each voltage plane to ground.

The large vias pairs on the left and right edge of the board are used for the connection of coax cables to a Spectrum Analyzer for drive and measurement. The board size is small (2” x 4”) in order to have the internal plane capacitance be small compared to the 0.01 uF capacitor being measured.

The method of measurement is described in more detail in Chapter-34 of “Right The First Time, A Practical Handbook on High Speed PCB and System Design” handbook. For this comparison, the measurements were made using an Agilent 8594EM Spectrum Analyzer. The frequency range covered was 100 KHz to 1.0 GHz.

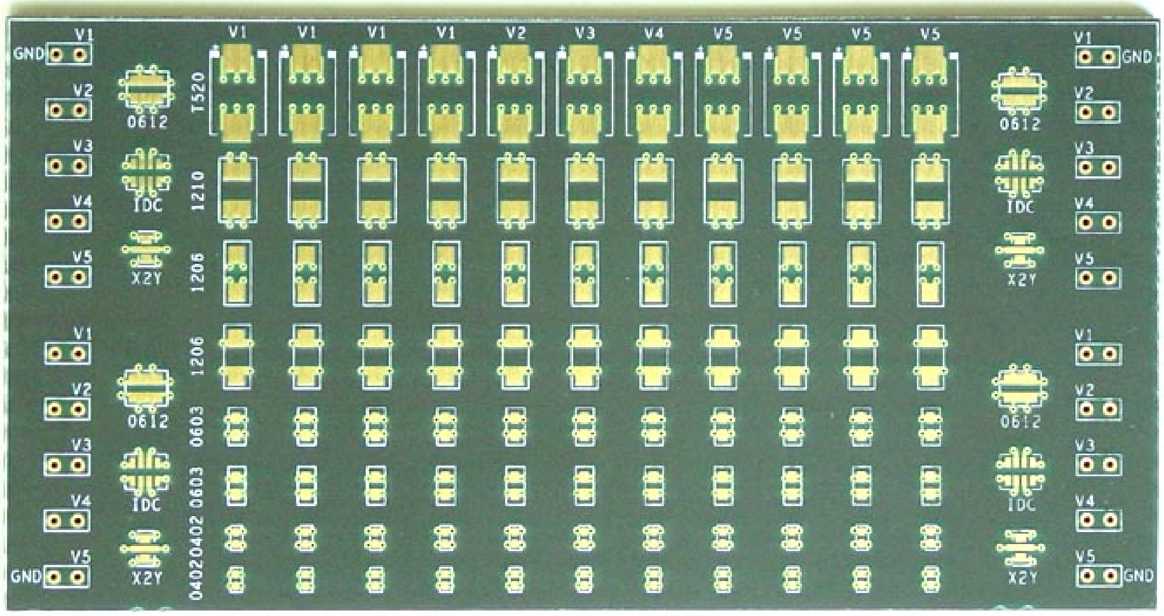

Figure 5 Capacitor Test PC Board (Size = 2" x 4")

Capacitors Evaluated

The two capacitor types used for this study are described in Figure 6.

Parameter	X2Y	0402
Vendor	Johanson	AVX
Vendor P/N	500X14W103MV4	0402YC103MAT2A
Value	0.01 uF	0.01 uF
Tolerance	+/- 20%	+/- 20%
Dielectric	X7R	X7R

Figure.6 Capacitors Evaluated

The following factors were evaluated as part of this study:

- Effect of capacitor footprint size on PCB design
- Which capacitor provides the best ESL
- Cost of capacitors
- Availability of capacitors

Measurement Results

Figure 7 is a plot of the impedance vs frequency for the X2Y capacitor. The plot contains four traces. The red trace labeled “V1” is for a capacitor connected to the top power plane pair. The green trace “V5” is to the bottom plane pair. The other two traces (Model-1 and Model-2) are plots from an equation that models the equivalent circuit shown in Figure 2, Model-1 was adjusted to match the V1 trace and Model-2 was adjusted to match the V5 trace.

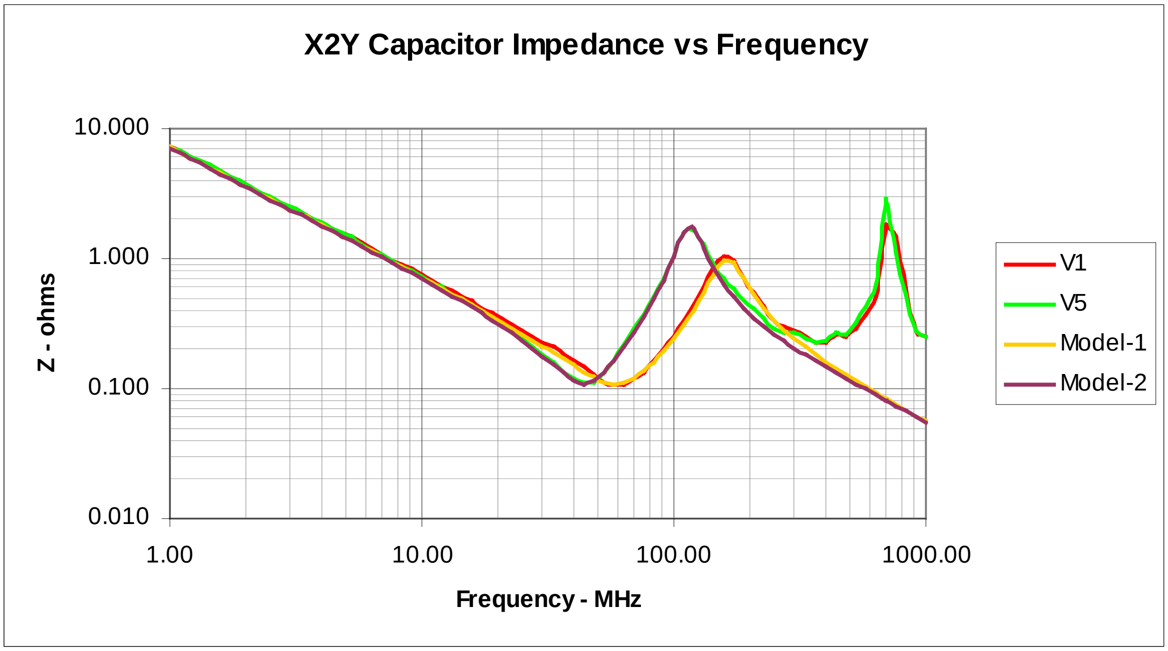

Figure 7 X2Y Capacitor Impedance vs Frequency

There is a difference between the plot of the measured results and the models in the high frequency range above 300 MHz. This is because the PC board power planes go into a standing wave resonance due to the physical size of the plane. The equation used for the model uses a zero inductance capacitor for the plane capacitance.

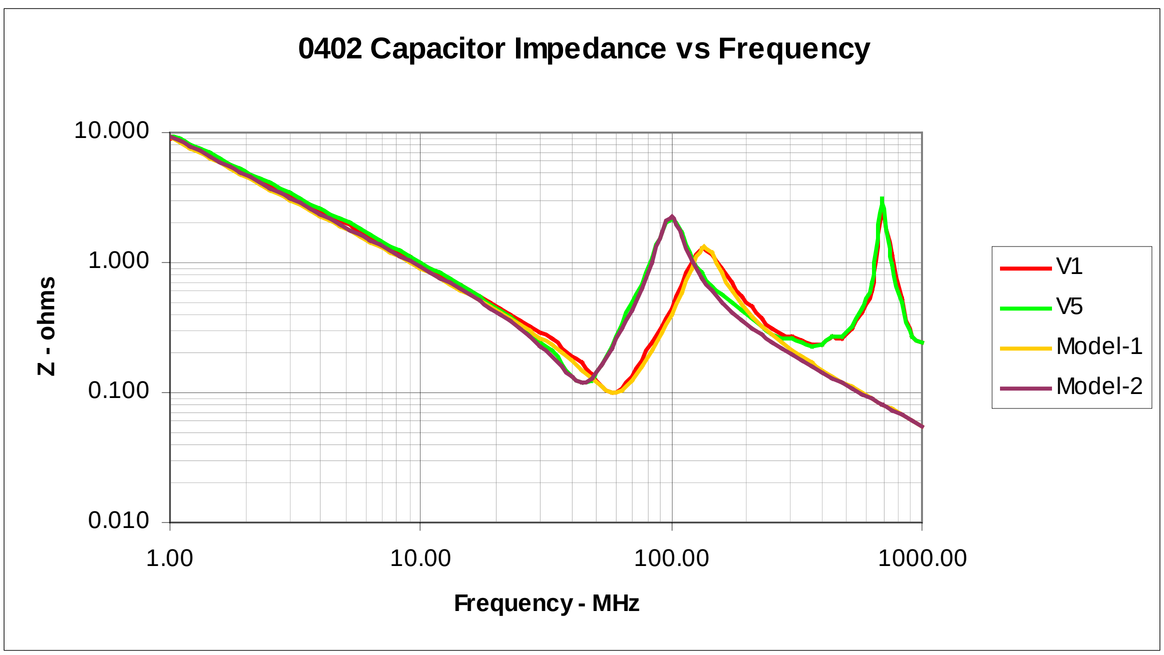

Figure 8 0402 Capacitor Impedance vs Frequency (1.0mm Pitch Vias)

Figure 8 is a plot of the impedance vs frequency for the 0402 capacitor using the footprint with a 1.0 mm via pitch. The plot contains four traces that have the same description as in Figure 7.

There is a difference between the X2Y (Figure 7) and the 0402 (Figure 8) capacitor impedance plots. In the X2Y plot, the V1 trace shows a series resonance at 60 MHz and a parallel resonance at 158 MHz. The V1 plot for the 0402 capacitor shifts to a slightly lower frequency with the series resonance at 57 MHz and the parallel resonance at 135 MHz. This is due to the lower ESL for the X2Y capacitor.

There is significant difference between the V1 and V5 plot for both capacitors. The V1 plot is a lower impedance at parallel resonance and shifted to a higher frequency than the V5 plot. This is because there is considerably higher ESL due to the via length for the capacitor connected to the V5 (bottom) power plane. In Figure 8 the V1 series resonance is at 57 MHz and the V5 resonance is at 43 MHz. The parallel resonance for the V1 plot is 135 MHz and for the V5 plot it drops to 100 MHz. The bottom line is that there is a much greater difference in the ESL due to via length than there is for the capacitor type.

The table in Figure 9 contains the circuit model parameters that best match the Spectrum Analyzer impedance plots.

	Power Plane Top (V1)			Power Plane Bottom (V5)		
Capacitor	Cap	ESR	ESL	Cap	ESR	ESL
And Footprint	nF	mΩ	nH	nF	mΩ	nH
X2Y	16.4	110	0.35	16.5	110	0.65
0402 1.00 mm	11.6	102	0.48	11.1	120	0.88
0402 1.27 mm	12.0	102	0.62	12.0	110	1.05

Figure 9 Parameter Measurement Results for X2Y and 0402 Capacitors

Summary

The X2Y capacitor definitely has lower ESL than the 0402 capacitor. On the top power plane the X2Y ESL is 0.35 nH while the 0402 (1.0mm footprint) ESL is 0.48 nH. The 0402 ESL is 37% larger. However the X2Y capacitor has 6 vias vs 4 for the 0402 capacitor and it takes up considerably more area on the PCB. The difference in ESL is much less on the bottom power plane due to the added via inductance.

High frequency decoupling on a real PCB requires many small capacitors in parallel between the voltage plane and ground in order to get the effective impedance down to the range of a few milliohms. Three 0402 capacitors would require a total of 12 vias. This is the same as two of the X2Y capacitors. The three 0402 capacitors in parallel would produce a little lower impedance than the two X2Y capacitors at a significantly lower cost.

Factors that influence my preference to use 0403 capacitors over X2Y are:.

1. The square via pattern for the 0402 capacitor is much better for internal signal layer routing.
2. The smaller footprint of the 0402 is easier to place on the PCB. In fact it can be placed on the opposite side of the board directly under an IC using a BGA package.
3. The cost of the X2Y is about 5x higher than the 0402.
4. The 0402 is manufactured in high volume by all suppliers of ceramic capacitors while the X2Y has limited sources.