# What You Need to Know About Bypass Capacitor S-Parameter Models

Istvan Novak, Samtec

Bypass capacitors are everywhere in power distribution networks. If you prefer thinking in the time domain, you may say that bypass capacitors store and supply charge on demand to the loads. If you like more frequency domain, you may say that bypass capacitors help us to meet the impedance target. In either way, it is very important to have good simulation models for our bypass capacitors so that we can do pre and post-layout simulations with confidence.

As an example, we use an actual capacitor made by Murata, a 47uF 1210-size X5R ceramic capacitor: GRM32ER60J476ME20. Murata has several simulation models available for their ceramic capacitors: simple and accurate C-R-L SPICE subcircuits and simple and accurate S-parameter models for shunt- and series-connected configurations as well as dynamic models. The simple SPICE model is a series C-R-L circuit with three frequency-independent elements [1], shown in *Figure 1*. For this particular capacitor, the capacitance is 37.4 uF, the resistance is 1.98 mOhm and the inductance is 535 pH. Notice that the capacitance is not the nominal 47 uF value, because this model is valid for small signals and the nominal capacitance is tested at 0.5 or 1 Vrms (for details, see for instance [2]). This model is linear, time invariant and it can be used in any kind of simulator: spreadsheets, SPICE-like circuit simulators or frequency-domain simulators. It captures the big picture correctly: the capacitive region, the inductive region and the minimum impedance at the series resonance.


```
*-----------------------------------------------
* SPICE Model generated by Murata Manufacturing Co., Ltd.
* Copyright(C) Murata Manufacturing Co., Ltd.
* Description :3225M(1210)/X5R/47uF/6.3V
* Murata P/N :GRM32ER60J476ME20
* Property : C = 47[uF]
* Data Generated on Mar 6, 2017
*-----------------------------------------------
* Applicable Conditions:
*    Frequency Range = 100Hz-6GHz
*    Temperature = 25 degC
*    DC Bias Voltage = 0V
*    Small Signal Operation
*-----------------------------------------------
.SUBCKT GRM32ER60J476ME20 port1 port2
C port1 11 3.74e-5
L 11 12 5.35e-10
R 12 port2 1.98e-3
.ENDS GRM32ER60J476ME20
```
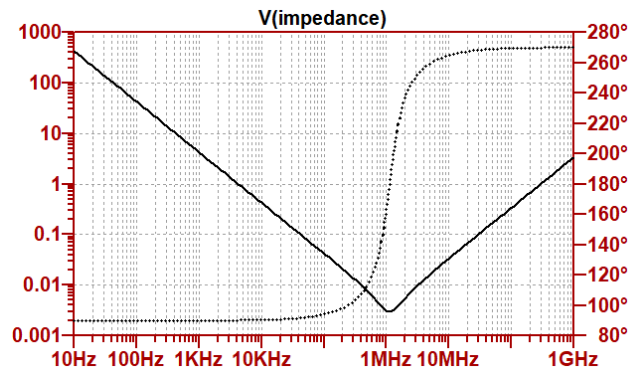
Figure 1: *C-R-L model of bypass capacitor and its impedance vs. frequency curve. Simple SPICE listing of a Murata* GRM32ER60J476ME20 *part is on the left, impedance plot is on the right. The solid line shows magnitude on the left axis and the dotted line is the phase on the right axis.*

Since the model is linear, it does not differentiate between the small-signal and large-signal behaviors. For linear models they are assumed to be identical. New high-density ceramic capacitors may have strong enough non-linear behavior that we need to take them into account in our simulations. For the very demanding simulation needs when nonlinearity or temperature/bias dependence must also be simulated, Murata provides the dynamic models [3]. In addition to dynamic models, more accurate and complex circuit-representation models are also available. We will cover those in a later article; here for the rest of the article we focus on S-parameter models.

S-parameter models are small-signal linear behavioral models of a component or circuit with any number of ports. They can easily capture frequency dependencies. For instance, it can properly capture the decay of capacitance as frequency increases, the complete variations of effective series resistance and also the lowering of inductance with increasing frequency. They can even capture secondary internal resonances. Between a Start and Stop frequency, the complex values of the S matrix elements are listed at a series of frequencies. S-parameter listings follow the Touchstone format; we show a partial listing in *Figure 5*.

Because S parameters are black-box behavioral models, we need to be careful about the Start and Stop frequencies and the available frequency points in between. If we need to do simulations outside of the model's frequency range, we have to check what our simulation tool does under those circumstances. For frequency points falling in between frequency points of the model, regular interpolation usually works well. But if we need to go significantly below the Start frequency or above the Stop frequency, the simulation tool has to make a guess about the behavior of our component, which may or may not match the real behavior.

Once we take care of the frequency range of the model, we need to decide how we connect the model to the rest of the circuit. As *Figure 2* illustrates it, a regular two-terminal capacitor is a one-port device. For parallel bypass applications, where one terminal is connected to the common return (ground), one-port models would be sufficient. But for DC-blocking applications, typical for signal interconnects where the driver and receiver may have different DC potential, we need two-port models. To have a consistent model format, it is common today to represent these capacitors with two-port S parameters for both applications. At this point the obvious question arises: what should we do with the two ports of our model in our simulator if this model represents a bypass capacitor?
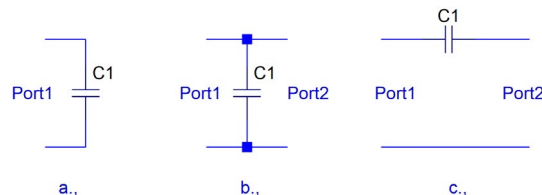
Figure 2.: One and two-port capacitor models.

The answer is based on simple common sense. For DC-blocking applications we have to select the series model and the wiring of the model is trivial. For bypass applications, we have a choice, we can use either of the model representations, we just have to do the port connections properly. When the model represents a shunt mode connection, we connect Port1 to our power network and leave Port2 open. Alternately, we can connect Port2 to our network and leave Port1 open, it will give the same result. When the model represents a series mode connection, we have to connect Port1 to our network and must short Port2, otherwise the capacitor is left floating and does not get connected to the rest of the circuit. Alternately we can connect Port2 to our network and short Port1, it will produce the same result. If we make sure that the connection of the capacitor model follows the above guidelines, the rest will be taken care of by the simulator and we get the correct result.

If we know the impedance/admittance of our capacitor at each frequency point, we can calculate the S-matrix elements for series mode and parallel mode connections. The S parameters can be found by using the following formulas [4]:

$$S_{series-thru} = \frac{1}{Z_S + 2Z_0} \begin{bmatrix} Z_S & 2Z_0 \\ 2Z_0 & Z_S \end{bmatrix}$$

$$S_{shunt-thru} = \frac{1}{Y_p + 2Y_0} \begin{bmatrix} Y_p & 2Y_0 \\ 2Y_0 & Y_p \end{bmatrix}$$

In the formulas Z_s is the impedance of the series-connected capacitor, Y_p is the admittance of the parallel-connected capacitor and Z_0 and Y_0 are the reference impedance and admittance, respectively. Note that because the matrix represents a symmetric and reciprocal network, the matrix will show symmetry along both diagonals: $S_{11}=S_{22}$ and $S_{21}=S_{12}$. Also, all four matrix elements depend on the unknown impedance or admittance. So don't get mislead when you may hear people say "Oh, this model must be fake. It shows perfect symmetry and reciprocity. It cannot be measured data." Yes, the simulation model is *not* measured data, it is *based on* measured data. The reciprocity and symmetry of the simulation model must be 'perfect' within numerical precision, otherwise we could not use it in a simulation.

We use the 'Accurate' S parameter models of the same Murata GRM32ER60J476ME20 capacitor to illustrate the various S-parameter models. *Figure 3* plots the S-parameter magnitudes for the shunt connection model. The S parameters for the series connection model are plotted in *Figure 4*. *Figure 5* shows the first few lines of the Touchstone model file for shunt mode connection. We can analyze the data of the two model files and can convince ourselves that what looks perfect graphical agreement on the screen is in fact the case down to the numerical resolution of the data: $S_{11}=S_{22}$ and $S_{21}=S_{12}$. If we compare the plots in the two figures, we notice that the characteristic V shape associated with capacitor impedance profiles shows up in S_{11} and S_{22} in the Series mode model and in S_{21} and S_{12} in the Shunt mode model. The only difference is a vertical offset of four (or 12 dB), which comes from the difference how we calculate the unknown impedance from S_{11} vs. S_{21}.
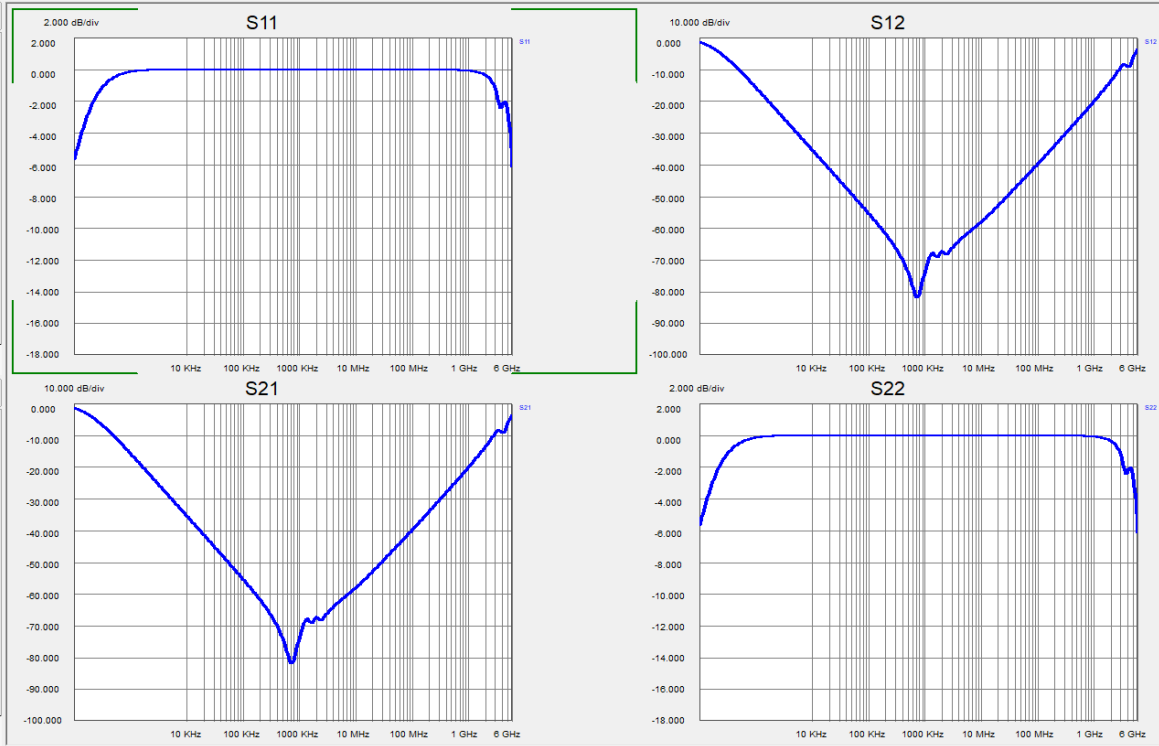
Figure 3: *Accurate S-parameter model of the GRM32ER60J476 capacitor in Shunt mode.*
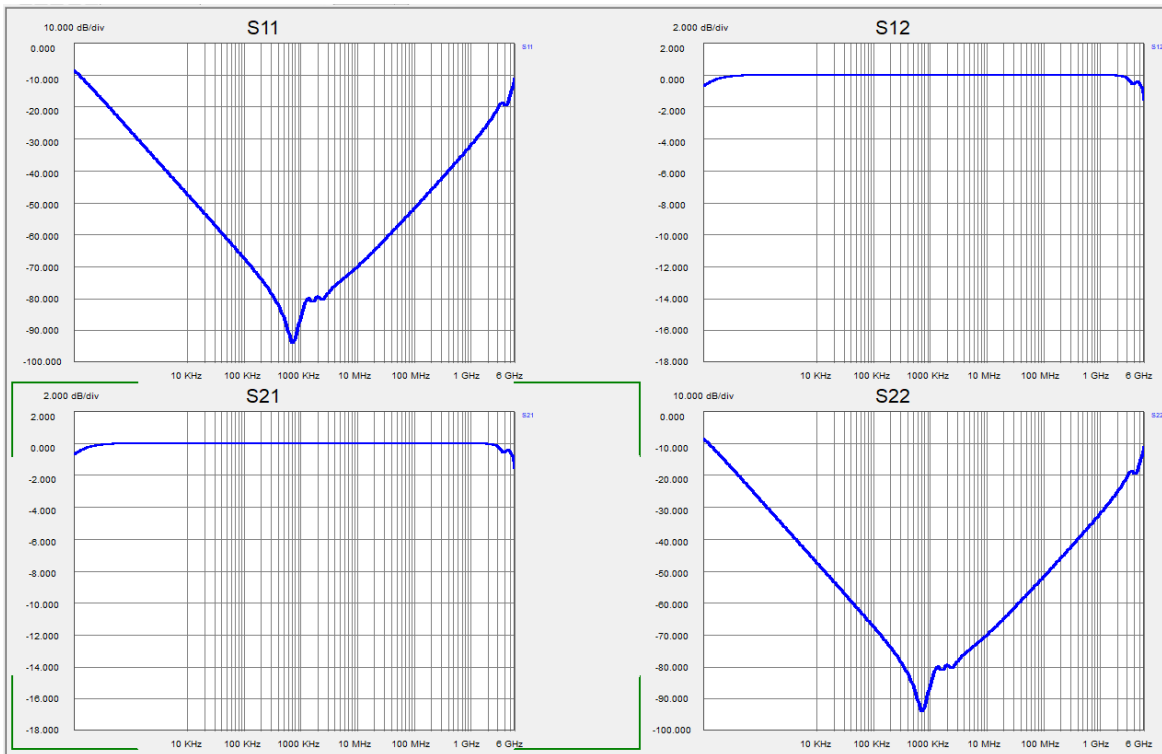
Figure 4: *Accurate S-parameter model of the GRM32ER60J476 capacitor in Series mode.*

```
!Murata Part Number: GRM32ER60J476ME20
!These Parameters are Measured in Shunt Mode Connection
!   o-----o
!   1
!Port1   =   Port2
!   1
!   o-----o
!Operation Temp=25[degC], DC Bias Voltage=0[V]
!Freq. Start=100[Hz] Stop=6[GHz], 401[Steps]
!Data Generated on Mar 6, 2017
Hz S RI R 50
!Freq.(Hz) S11(Real) S11(Imag) S21(Real) S21(Imag) S12(Real) S12(Imag) S22(Real) S22(Imag)
100.0 -0.2765365786714686 -0.4398234698073025 0.7234634213285315 -0.4398234698073025
0.7234634213285315 -0.4398234698073025 -0.2765365786714686 -0.4398234698073025
104.57921513303334 -0.2943509078800123 -0.44828004994699605 0.7056490921199876 -0.44828004994699605
0.7056490921199876 -0.44828004994699605 -0.2943509078800123 -0.44828004994699605
109.3681223784127 -0.3128230518854099 -0.45615888390741566 0.6871769481145901 -0.45615888390741566
0.6871769481145901 -0.45615888390741566 -0.3128230518854099 -0.45615888390741566
114.3763239890794 -0.33191392783131907 -0.4633974963912099 0.668086072168681 -0.4633974963912099
0.668086072168681 -0.4633974963912099 -0.33191392783131907 -0.4633974963912099
119.61386192579457 -0.35157680101111377 -0.46993628303669593 0.6484231989888863 -0.46993628303669593
0.6484231989888863 -0.46993628303669593 -0.35157680101111377 -0.46993628303669593
```

**Figure 5:** Partial listing of the Touchstone file with the Shunt-mode S-parameter data.

A circuit simulator does not know what the model represents and therefore to work properly, the simulator needs all four S parameters in the matrix. We, on the other hand, know that the model represents a single capacitor, moreover we also know whether the model represents a series mode connected or parallel mode connected capacitor. Based on the definitions of the S matrix elements, we can calculate the unknown impedance from any of the four matrix elements. *Figure 6* illustrates the conditions for calculating S-matrix elements: $S_{11}$ on the left for series mode connection and $S_{21}$ on the right for parallel mode connection. Similar equivalent schematics can be drawn for the other S matrix element definitions as well.
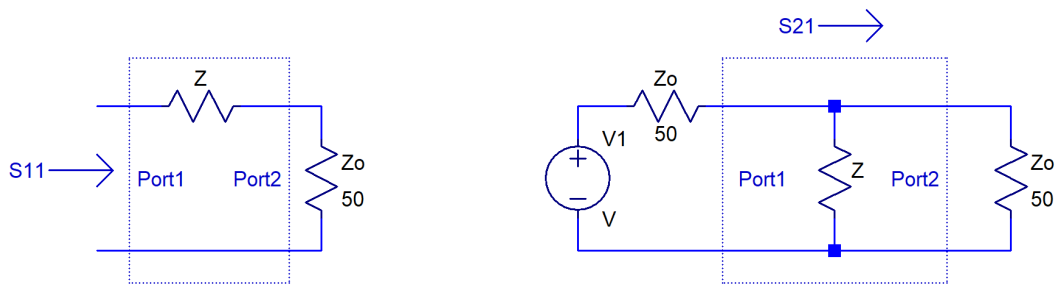


*Figure 6: Conditions for calculating the S parameters for series mode and parallel mode connections.*

By using the symmetry and reciprocity, we have four different ways to calculate the Z impedance of the capacitor: a) from $S_{11}$ (or $S_{22}$) of the series connection model, b) from $S_{21}$ (or $S_{12}$) of the series connected model, c) from $S_{11}$ (or $S_{22}$) of the parallel connection model and d) from $S_{21}$ (or $S_{12}$) of the parallel connection model.

a) $Z = Z_0 \frac{1-S_{11}}{1+S_{11}} - Z_0$

b) $Z = 2Z_0 \frac{1-S_{21}}{S_{21}}$

c) $Z = \frac{1}{Re\{Y_{in}\}-\frac{1}{Z_0}+jIm\{Y_{in}\}}$    $Y_{in} = \frac{1}{Z_0} \frac{1-S_{11}}{1+S_{11}}$

d) $Z = \frac{Z_0}{2} \frac{S_{21}}{1-S_{21}}$

To illustrate that in fact all of the above ways produce the same result, *Figure 7* plots the impedance magnitude calculated from the above four possible options. As expected, all four lines run exactly on top of each other. The red line shows the measured impedance profile of the same capacitor. Note that the only minor difference comes from a slight shift of the series resonance frequency, which is due to differences in the test fixtures used for measuring the component. The accurate S parameter model accurately captures the frequency dependence of inductance and the secondary resonances.
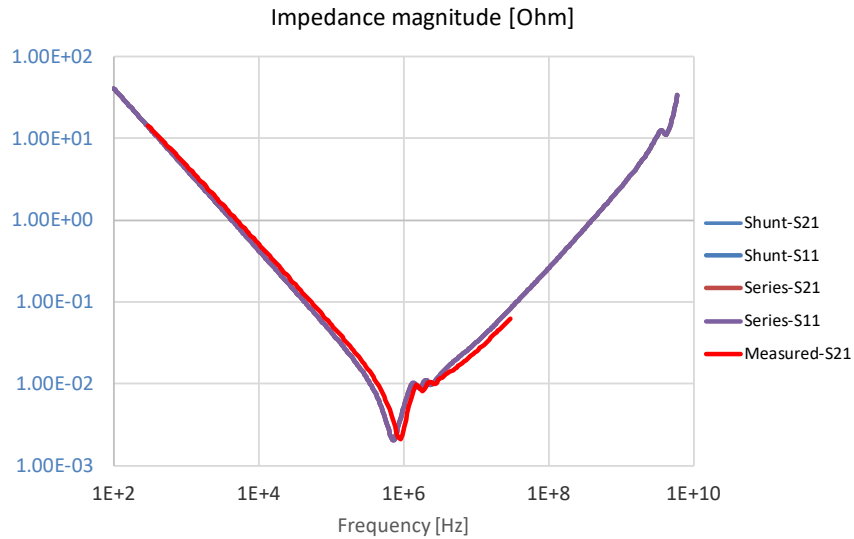


*Figure 7: Capacitor impedance magnitude extracted from the series mode and shunt mode S-parameter models over the 100 Hz 6 GHz frequency range. Red trace: measured impedance from Two-port Shunt-through configuration.*

Finally, we need to remember that while any of the above four possibilities can be used to calculate the impedance of the capacitor from simulation models, when it comes to measurements of low-impedance capacitors, practical limitations and typical measurement errors limit us and we can use only $S_{21}$ (or $S_{12}$) from the parallel connection data.

**References:**

[1]   Murata Simsurfing tool for ceramic capacitors, https://ds.murata.co.jp/simsurfing/mlcc.html?lcid=en-us

[2]  "How much capacitance do we really get?" http://www.electrical-integrity.com/Quietpower_files/QuietPower-40.pdf

[3]  "Dynamic models for Passive Components," http://www.electrical-integrity.com/Quietpower_files/QuietPower-36.pdf

[4]  https://product.tdk.com/en/technicalsupport/seat/pdf/etutorial_007.pdf