Using TDK SPICE Netlist Library and Its Basic Applications

Katsufumi EHATA
Application Center, TDK Corporation

Feb. 14, 2007. AN-NL06B001Rev1_en

Abstract

Given that circuit design by simulators has recently become popular, TDK has released TDK SPICE Netlist Library for designing electronic circuits by circuit simulators. The library includes a set of equivalent circuit models of TDK electronic components written in SPICE Netlist format. Using this library, parasitic factors existing in electronic components can be considered in the circuit simulation.

This application note describes the TDK SPICE Netlist Library, using it in SPICE simulators, and its basic applications.

Library introduction

If an electronic component is simulated as an ideal one in a high frequency circuit, the simulated result does not match the actual property because real components have some parasitic factors whose effects cannot be disregarded at high frequencies. For designing high frequency circuits, TDK has released TDK SPICE Netlist Library, which includes a set of equivalent circuit models of TDK electronic components. Some of the parasitic factors in electronic components are considered in the equivalent circuit. Using this library, the actual property of electronic components can be considered in circuit simulation. All the models in the library are written in the subcircuit format of a generic SPICE Netlist. Since many circuit simulators including SPICE can interpret this format, this library can be used in many different simulators.

This library contains models of ferrite beads (145 products), three-terminal filters (70 products), common-mode filters (31 products), and pulse transformers (6 products) as of February 2007.

How to use the library

To obtain the library

The library can be downloaded from the TDK website at http://www.tdk.co.jp/etst. Since the downloaded file is compressed into a zip format, you will need to unzip the file before use. Read the readme file, which includes information about the library, before you start using the library.

Model library

Using a model from the library as an example, basic library use is

```
*---------------------------------------------
* SPICE Netlist Generated by TDK Corporation
* TDK P/N: ACM2012-900-2P (Common Mode Filter)
* Property: Zc(at 100MHz) = 90 [ohm]
* Model Generated on Dec. 04, 2006
*---------------------------------------------
* External Node Assignments:
*
*  1 ---@@@--- 4
*       ===
*  2 ---@@@--- 3
*
*---------------------------------------------
.SUBCKT ACM2012_900_2P 1 2 3 4
C11 11 12 0.84p
C12 21 22 0.84p
C13 31 32 0.84p
C14 41 42 0.84p
C21 12 42 0.02p
C22 22 32 0.02p
C41 12 22 0.95p
C42 32 42 0.95p
K11 L21 L22 0.999999999
L11 11 12 3.1n
L12 21 22 3.1n
L13 31 32 3.1n
L14 41 42 3.1n
L21 12 42 200n
L22 22 32 200n
R11 11 12 14
R12 21 22 14
R13 31 32 14
R14 41 42 14
R21 12 42 880
R22 22 32 880
R31 1 11 0.065
R32 2 21 0.065
R33 3 31 0.065
R34 4 41 0.065
R41 12 22 1G
R42 32 42 1G
R51 1 4 1G
R52 2 3 1G
.ENDS ACM2012_900_2P
*---------------------------------------------
```

Fig. 1. Model of "ACM2012-900-2P"

explained below. A model file is prepared for each product. Although the file extension is .mod, the file contents can be viewed and edited with a text editor because the file is in ASCII format. Here, a common-mode filter for differential signal lines "ACM2012-900-2P" is used as an example. A model of this product is described in the file, "ACM2012-900-2P.mod," and file contents are shown in Fig. 1. This model is written in a subcircuit format of a generic SPICE Netlist. Subcircuit is used to define the function of a specific circuit as a circuit package. A defined subcircuit can be applied whenever and wherever needed. Subcircuit is useful when a specific circuit is put in two or more places, or when part of the whole circuit is replaced by various subcircuits, to examine its properties.

Figure 1 shows the model details. Lines that start with an asterisk (*) are comment lines and not used in simulation. These lines describe the product name, product category, typical properties of the product, external node assignment of subcircuit, and other information. The subcircuit definition is from the line of ".SUBCKT" to ".ENDS". Between these lines, the equivalent circuit of ACM2012-900-2P is written in Netlist format. The actual electronic property can be considered in circuit simulation because some parasitic factors in electronic components are considered in the equivalent circuit. Information about the equivalent circuit is omitted in this application note.

External node assignment

In Fig. 1, the following expression defines the subcircuit:

```
.SUBCKT ACM2012_900_2P 1 2 3 4
```

The expression indicates that the subcircuit name is "ACM2012_900_2P" and 1, 2, 3, and 4 are assigned as the external nodes. Since the subcircuit is connected to the external circuit through these nodes, correctly recognizing each node and connecting them to the external circuit is important. In this example, external nodes 1, 2, 3, and 4 correspond to terminals as specified by the following diagram:

```
*  1 ---@@@--- 4
*       ===
*  2 ---@@@--- 3
```

The diagram shows that each pair of nodes, such as nodes 1 and 2 or nodes 4 and 3, need to be connected to I/O nodes of the differential signal to make this circuit a common-mode filter. Note that the external node assignment is different in each product. The diagram in the model or the readme file included in the library provides information.

Using the library

This section describes how to use the library in a SPICE simulator. A case in which Netlist is used to input a circuit to a simulator is explained here. For information about SPICE Netlist, consult http://bwrc.eecs.berkeley.edu/Classes/IcBook/SPICE or the manual for the simulator being used.

Two operations are necessary to use the model. The first operation is to copy the contents of the model file and paste it in the Netlist of a circuit where the model will be used. Some simulators can include an external file into a Netlist. In such simulators, a model file can be included into a Netlist without copying and pasting. The second is to add a description of calling the subcircuit in the Netlist. The second operation is explained in detail as follows:

In SPICE Netlist, a device whose name starts with X is used for a subcircuit call. For example, the following description is used to call a subcircuit named ACM2012_900_2P:

```
XCMF 12 22 23 13 ACM2012_900_2P
```

Although the device name is XCMF in the example, any other names starting with X can be used. Four numbers following XCMF show node names of the external circuit to which the subcircuit is connected. In the above example, external nodes 1, 2, 3, and 4 of the subcircuit are connected, respectively, to nodes 12, 22, 23, 13 of the external circuit. Note that the name of the external node in a subcircuit may differ from the name of the node of the external circuit to which the subcircuit is connected. The important issue is not the node name itself, but the order of description. In this example,

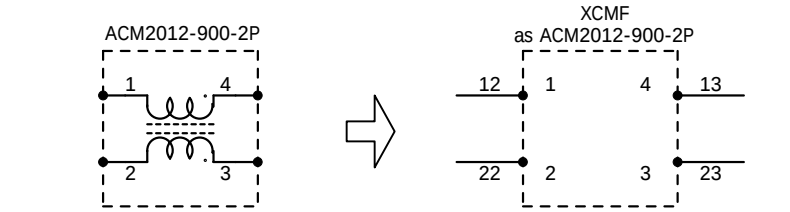


Fig. 2. Connections between the subcircuit and external circuit

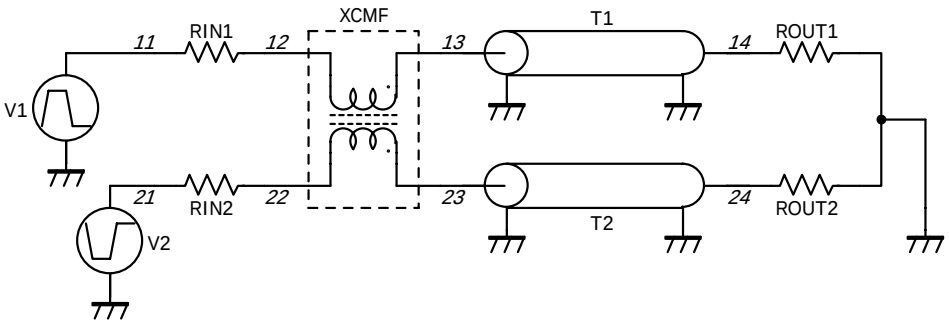


Fig. 3. Differential transmission circuit

the connecting relation between the internal construction of the subcircuit and the external circuit is shown in Fig. 2. A pair of nodes, 12 and 22, and another pair of nodes, 13 and 23, of the external circuit work, respectively, as I/O nodes of the differential signal. Recognizing the external node assignment of the subcircuit referring to the diagram and correctly connecting them to the external circuit is important.

- Basic library applications

Basic library applications are explained below. Let us consider a differential transmission circuit of USB 2.0 as shown in Fig. 3. The USB transmission line should be a pair of coupled lines, but this example uses two single-end lines excited in negative phase to model a differential circuit for ease of understanding. V1 and V2 are voltage pulse sources, RIN1 and RIN2 are output impedance of the sources, T1 and T2 are loss-less transmission lines, ROUT1 and ROUT2 are load resistances, and XCMF is a common-mode filter. The italic number in the figure indicates the node name.

V1 and V2 are excited in the anti-phase and compose a differential signal source with RIN1 and RIN2. The actual signal source has

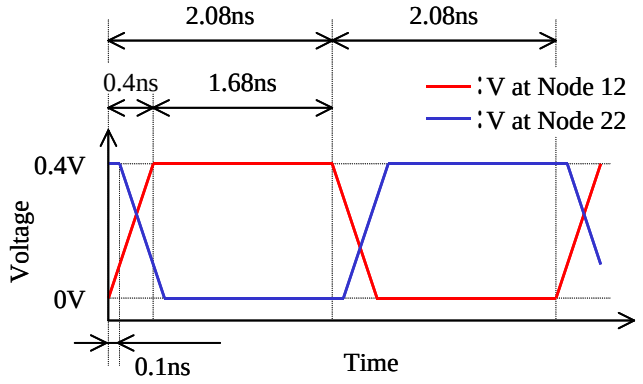


Fig. 4. The output signal pattern from signal source

```
***<< simulation of USB2.0 circuit >>************
*---< main circuit >---*
V1 11 0 PULSE 0.0 0.8 0.0N 0.4N 0.4N 1.68N 4.16N
V2 21 0 PULSE 0.8 0.0 0.1N 0.4N 0.4N 1.68N 4.16N
RIN1 11 12 45
RIN2 21 22 45
XCMF 12 22 23 13 ACM2012_900_2P
T1 13 0 14 0 Z0=45 TD=2N
T2 23 0 24 0 Z0=45 TD=2N
ROUT1 14 0 45
ROUT2 24 0 45
.TRAN 0.05N 10N
.PRINT TRAN V(14) V(24)
*
*---< subckt >---*
*-----------------------------------------------
* SPICE Netlist Generated by TDK Corporation
* TDK P/N: ACM2012-900-2P (Common Mode Filter)
* Property: Zc(at 100MHz) = 90 [ohm]
* Model Generated on Dec. 04, 2006
*-----------------------------------------------
* External Node Assignments:
*
*  1 ---@@@--- 4
*       ===
*  2 ---@@@--- 3
*
*-----------------------------------------------
.SUBCKT ACM2012_900_2P 1 2 3 4
C11 11 12 0.84p
C12 21 22 0.84p
C13 31 32 0.84p
C14 41 42 0.84p
C21 12 42 0.02p
C22 22 32 0.02p
C41 12 22 0.95p
C42 32 42 0.95p
K11 L21 L22 0.999999999
L11 11 12 3.1n
L12 21 22 3.1n
L13 31 32 3.1n
L14 41 42 3.1n
L21 12 42 200n
L22 22 32 200n
R11 11 12 14
R12 21 22 14
R13 31 32 14
R14 41 42 14
R21 12 42 880
R22 22 32 880
R31 1 11 0.065
R32 2 21 0.065
R33 3 31 0.065
R34 4 41 0.065
R41 12 22 1G
R42 32 42 1G
R51 1 4 1G
R52 2 3 1G
.ENDS ACM2012_900_2P
*-----------------------------------------------
*
.END
```

Fig. 5. The Netlist to simulate the circuit shown in Fig. 3

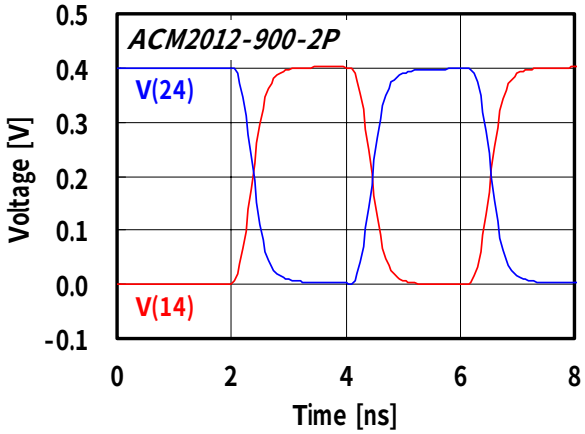
non-zero impedance. But the output impedance of a voltage source is zero in simulation. Hence, the output impedance of the source is represented by RIN1 and RIN2. If there is time lag between anti-phase pulse signals excited by V1 and V2, skew (time lag between differential signals) is generated and a common-mode (in-phase) noise is added to the differential signal. Since a common-mode noise causes radiation noise, this effect should be suppressed as much as possible by using a common-mode filter or other measures. When a filter not suitable for the system is applied, the signal pattern will be distorted and transmission errors will occur. Here, the circuit is simulated by transient analysis function of SPICE, showing a common-mode filter suppresses a common-mode noise generated by skew and an unsuitable common-mode filter distorts the signal pattern.

In high-speed USB 2.0, data is transmitted at 480 Mbps using a differential signal with voltage amplitude of 0.4 Vp-p, resulting in a time span of about 2.08 ns per bit. Supposing the rise and fall time of the pulse is 0.4 ns, the time lag of 0.1 ns is set in V2 to generate skew in order to add a common-mode noise. The output voltage pattern from the signal source (pulse source + output impedance) is shown in Fig. 4. The conditions of V1 and V2 are set to make the signal patterns of node 12 and 22 as in this figure.

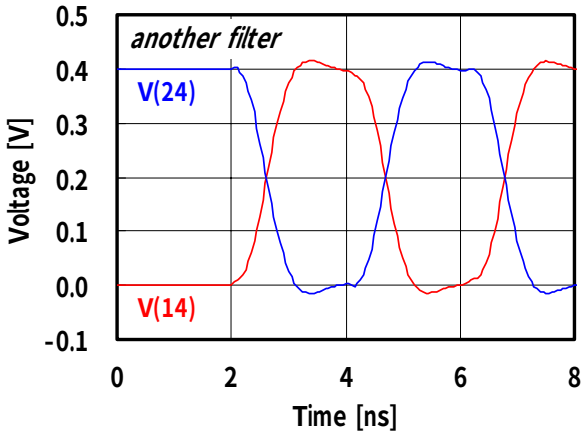
USB 2.0 specifications have a differential characteristic impedance of 90 ohm. The characteristic impedance of each single-end line is set to 45 ohm to model this. The output impedance of the signal source, characteristic impedance of the transmission line, and load resistance are also 45 ohm.

Figure 5 shows the Netlist to simulate the circuit shown in Fig. 3. The main circuit and analysis commands are written in the top-half and the model of the common-mode filter is written in the bottom-half of the Netlist. Signal sources, transmission lines, and resistors are generic SPICE elements, while the common-mode filter is a model of "ACM2012-900-2P" included in the library. Since the input nodes of the differential signal to the common-mode filter are nodes 12 and 22 and output nodes of the signal from the filter are nodes 13 and 23, as shown in Fig. 3, the subcircuit call is described as "XCMF 12 22 23 13 ACM2012_900_2P". The electric length (delay time) of the transmission line is supposed to be 2 ns. The analysis commands are specified to perform a transient analysis to 10 ns with a time step of 0.05 ns and to output voltage signal patterns at nodes 14 and 24.

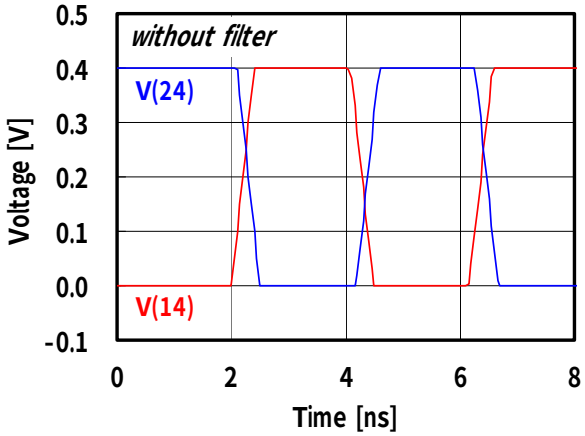
As shown in Fig. 5, the Netlist specifies that the voltage amplitude of pulse generated by V1 and V2 is 0.8 Vp-p, making the output signal pattern from the voltage source (pulse source + output impedance) determined only by load resistance, as shown in Fig. 4. This results from considering that the output voltage of the pulse source is divided by output impedance and load resistance.



(a) with common-mode filter "AMC2012-900-2P"



(b) with other common-mode filter



(c) without common-mode filter

Fig. 6. The simulated results of transmitted signal pattern

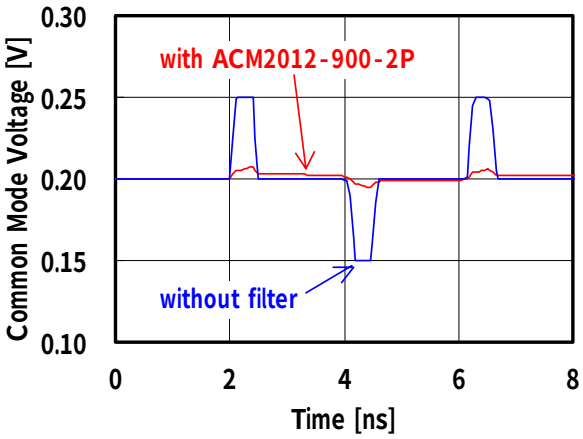


Fig. 7. Common-mode noise patterns

Fig. 6 shows the simulated results of the Netlist in Fig. 5 by a SPICE simulator. For comparison, three results are shown: (a) with common-mode filter AMC2012-900-2P (common-mode impedance: 90 ohm @ 100 MHz; suitable for USB); (b) with other common-mode filter (common-mode impedance: 600 ohm @ 100 MHz; not suitable for USB); (c) without common-mode filter.

In all results, an output signal pattern responding to the input pulse is obtained after a time lag corresponding to the delay time (2 ns) of the transmission line. Without a common-mode filter (Fig. 6c), skew generated at the signal source appears directly at the receiving point. In contrast, common-mode filters (Figs. 6a, b) suppress the skew very well. Using ACM2012-900-2P optimized for USB, a good transmitted signal pattern is obtained with little distortion of the signal pattern. Using another common-mode filter distorts the transmitted signal pattern and may cause transmission error. Selecting a suitable product for the system is important.

Common-mode noise patterns are shown in Fig. 7 where the common-mode noise is calculated by a formula of (V(14)+V(24))/2. Figure 7 shows the result without common-mode filter as well as with ACM2012-900-2P. The amplitude of the common-mode noise is decreased from 100 mVp-p to 14 mVp-p using ACM2012-900-2P, which is expected to suppress radiation noise.

• Conclusion

This application note has described the TDK SPICE Netlist Library, using it in SPICE simulators, and its basic applications, explaining the basic issues of the library. The library can be used more conveniently, depending on the simulator used. Loading a model file using the file including function or making a circuit on a GUI using a circuit editor are two ways to benefit from such added convenience. Using a model of an actual IC as a driver or receiver can enable circuit simulation matching the actual properties. A model of a transmission line considering loss and coupling is also effective. Your simulator manual provides details.

Please visit the website of our technical support tools at http://www.tdk.co.jp/etst.


Cautionary notes

- The data in this application note and TDK SPICE Netlist Library do not guarantee product features.
- TDK is not responsible for losses of any kind arising from using the contents of this application note or TDK SPICE Netlist Library.
- The contents of this application note are as of February 2007.
- The contents of this application note and TDK SPICE Netlist Library are subject to change without notice.