AN12298

High frequency design considerations

Basic overview

Rev. 0 — 12/2018 Application Note

by: NXP Semiconductors

Contents

1 Introduction....1
2 Transmission line types....1
3 Properties of the PCB material used....4
4 Guided wavelength and propagation velocity....5
5 Characteristic impedance....7
6 Referencing....7
7 Interference between transmission lines....20
8 Radiation....28
9 Differential pairs....29
10 Other routing considerations....30
11 Simulations....34
12 Used models....34
13 Disclaimer....44
14 References....44

1 Introduction

Increasingly higher data rates bring the necessity to work with high frequency signals in an increasing number of areas. In the SoC designs, this mainly concerns the DRAM, PCIe, USB, and SATA interfaces, but applies to any other interface working with high-frequency signals. This application note helps you to understand the considerations around the transmission lines to increase the number of successful designs and to provide an insight into the background of the specific recommendations listed in the Hardware Development Guides.

Figures in this AN were obtained by simulations. For details, see Used models on page 34.

2 Transmission line types

A PCB connection on which propagates a high-frequency signal is called a transmission line. There are two types of transmission lines that are typically used in PCBs—Microstrip and Stripline. Each of them comprises a signal trace and a reference plane(s). It is very important to perceive the signal trace and the reference plane as a whole and never divide them because (based on this geometry) both of the transmission line types have their typical EM field distributions, which define their properties. By selecting the PCB material and adjusting the below highlighted dimensions, the field distribution can be shaped to some extent and, therefore, the properties of the transmission lines can be designed to suit your needs.

The EM field has two components—E (electric) and H (magnetic) that are tied together by Maxwell's equations. To keep the figures simple and informative, there is always displayed only the E-field distribution, because it is more commonly known.

2.1 Microstrip

Microstrip is a transmission line that is routed on the surface of the PCB and, therefore, surrounded by two environments: PCB material and air. It is referenced to only one reference plane under the signal trace.

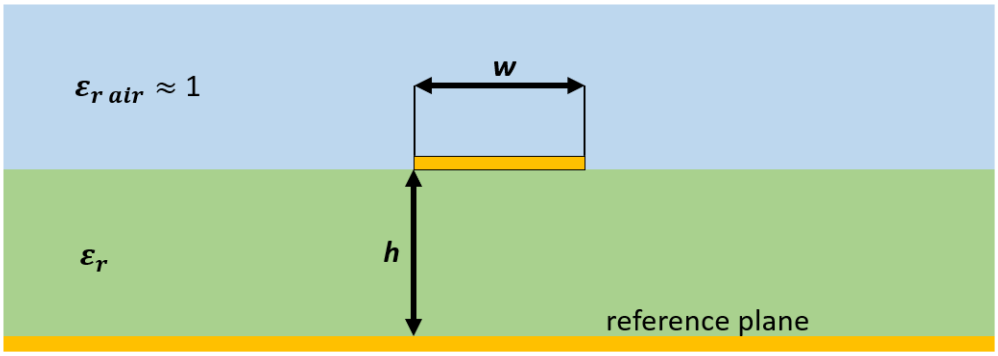

Figure 1. Microstrip geometry

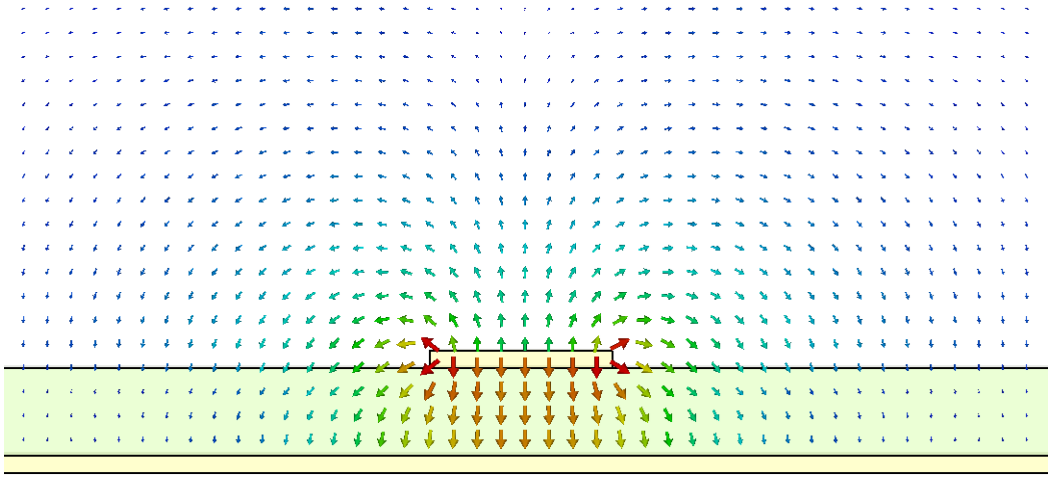

Figure 2. Typical microstrip E-field distribution (vector view)

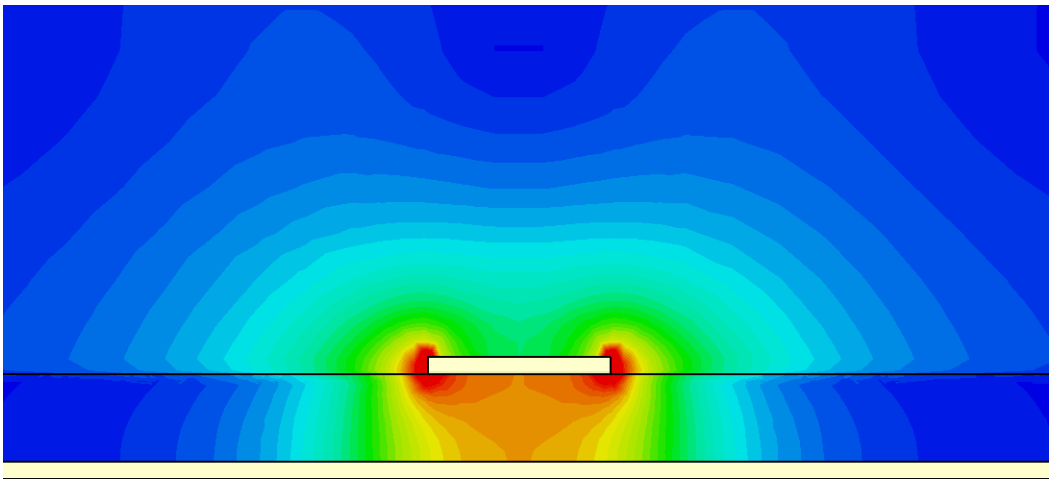

Figure 3. Typical microstrip E-field distribution (magnitude view)

The typical E-field distribution of a microstrip is shown in Figure 2. on page 2 and Figure 3. on page 2. Notice that the field runs through two environments and the vectors curve towards the reference plane. The magnitude view gives a good illustration on how far the field can spread from the signal trace and how the trace radiates into free space.

2.2 Stripline

Stripline is a transmission line that is routed in one of the inner PCB layers and, therefore, surrounded only by the PCB material. It is referenced to two reference planes—under and above the signal trace.

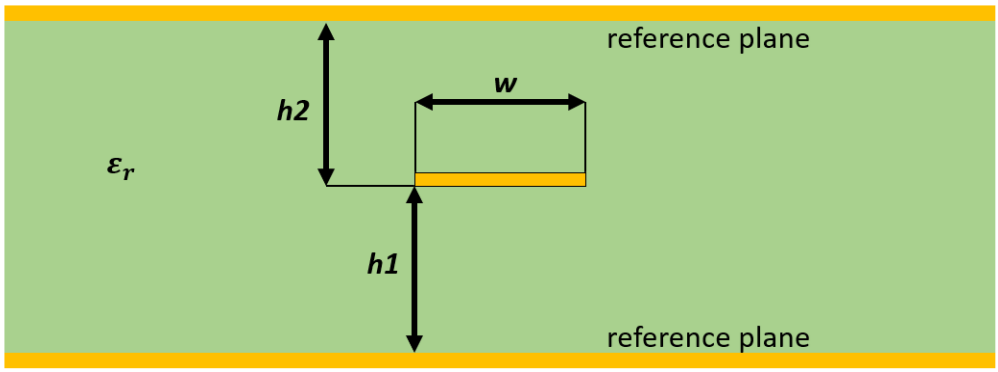

Figure 4. Stripline geometry

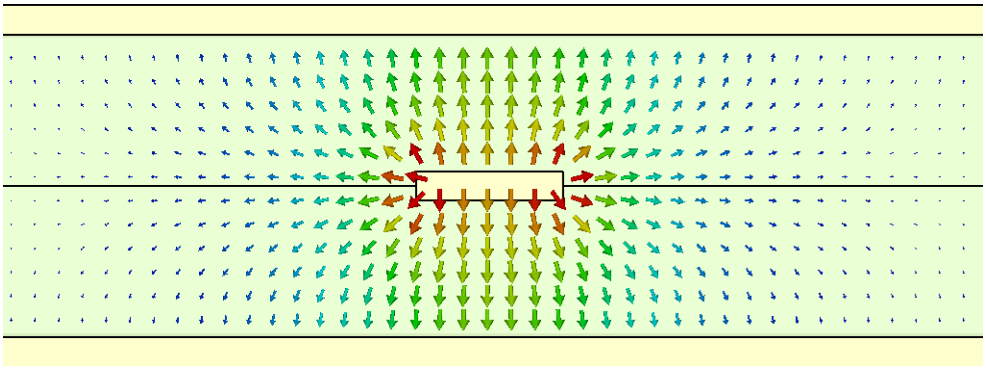

Figure 5. Typical E-field distribution of a symmetrical stripline (vector view)

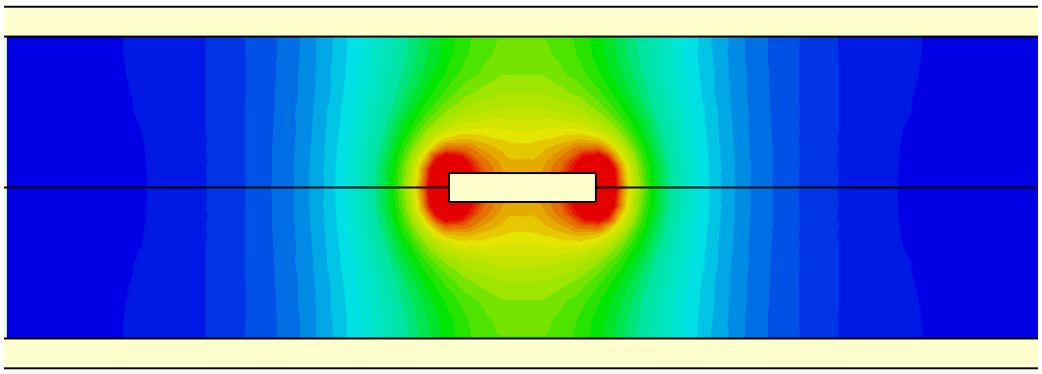

Figure 6. Typical E-field distribution of a symmetrical stripline (magnitude view)

A typical E-field distribution of a symmetrical stripline is shown in Figure 5. on page 3 and Figure 6. on page 3. The distance of the signal trace to both reference planes is the same. In effect, the signal trace is referenced to both reference planes equally.

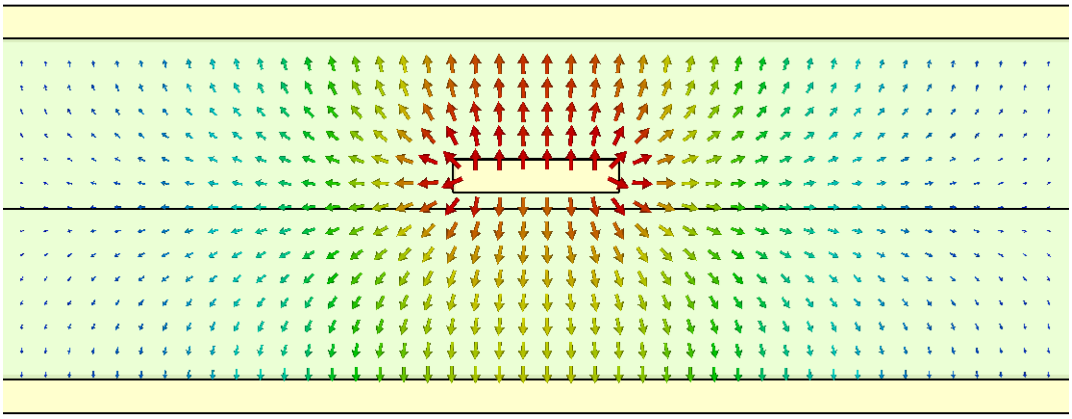

Figure 7. Typical E-field distribution of an asymmetrical stripline (vector view)

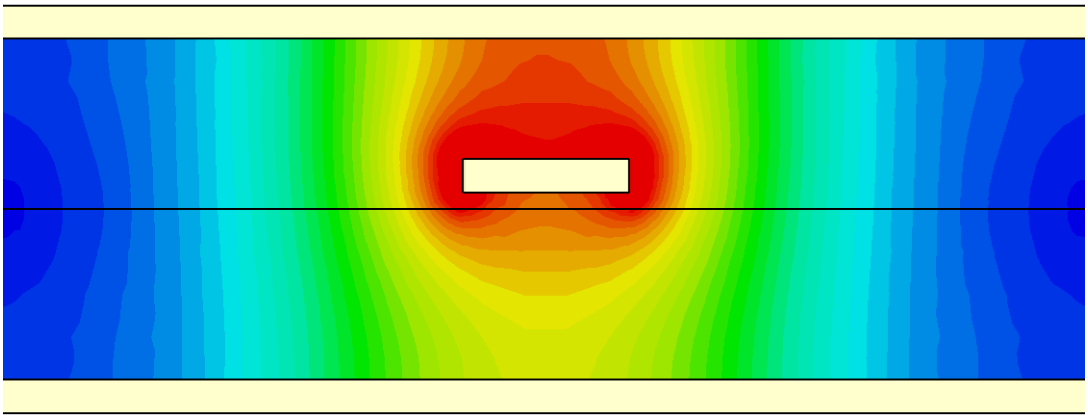

Figure 8. Typical E-field distribution of an asymmetrical stripline (magnitude view)

A typical E-field distribution of an asymmetrical stripline is shown in Figure 7. on page 4 and Figure 8. on page 4. In this case, the signal trace is located closer to the top reference plane than to the bottom reference plane. In effect, the top reference plane plays more important role in terms of referencing. However, connection to the bottom plane persists. Therefore, it still needs to be taken into consideration.

3 Properties of the PCB material used

Which PCB material is the best? This is a very common question. The answer is: There is no such thing. There are only PCB materials less and more suitable for the specific application because, in many cases, the requirements go against each other. For high-frequency design, two numbers are important:

- Dielectric constant (ε_r)—how electrically dense the material is in relation to vacuum.
- Loss tangent ($\tan \delta$)—how lossy the material is (the lower it is, the better).

Influence of the dielectric constant together with the highlighted dimensions of the transmission lines in Transmission line types on page 1 is described in the following sections.

NOTE

Many manufacturers list two dielectric constant numbers in their documentation—process and design. Always use the design number.

4 Guided wavelength and propagation velocity

Dielectric constant of the used material directly affects the guided wavelength of the signal on the PCB. As the frequency is always the same, the propagation velocity has to change. Because the dielectric constant of the PCB material is always greater than 1 (ε_r of vacuum = 1, ε_r of the air ≈ 1), the guided wavelength on the PCB will always be shorter than wavelength in vacuum (air) and propagation velocity will be lower than speed of light. The situation slightly differs, depending on which transmission line type we use.

This phenomenon is also closely related to the overall design precision demands. The higher the dielectric constant is, the shorter is the guided wavelength. Therefore, the same length mismatch between two traces will cause bigger propagation delay difference and the design precision demands go up. From this point of view, it makes sense to use a material with a low dielectric constant. However, there are more aspects to consider, which are covered in the following chapters.

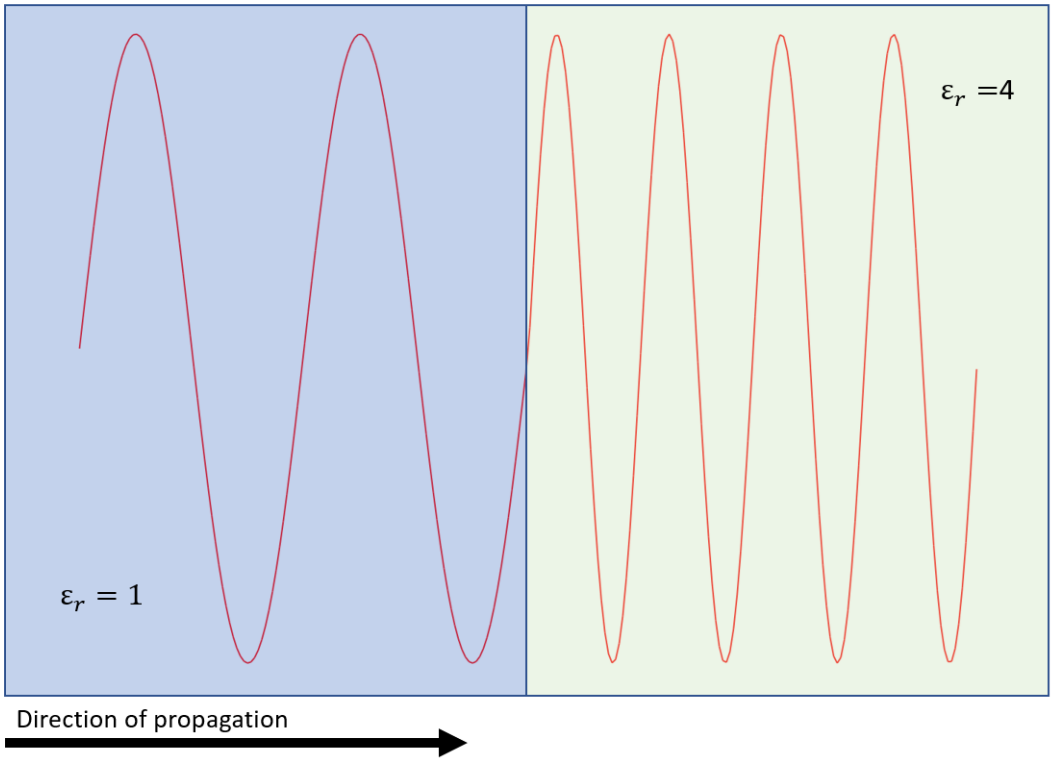

Figure 9. Shrinkage of the wavelength when transitioning to a material with a higher dielectric constant

NOTE

The X-axis (direction of propagation) is the measure of distance, not time (frequency of the signal remains the same in both environments).

4.1 Stripline

The situation of striplines is simple:

$$\lambda_g = \frac{c}{f\sqrt{\varepsilon_r}}$$

Figure 10. Calculation of the guided wavelength for striplines

As we can see, the only input parameter is the dielectric constant of the PCB material. This is because stripline is surrounded only by the PCB material.

4.2 Microstrip

For microstrips the situation is more complicated. As mentioned in Transmission line types on page 1, microstrips are surrounded by PCB material and air. As part of the EM field runs through the air and part through the PCB material, the global dielectric constant of the surrounding environment will lay somewhere between the dielectric constant of the PCB material and the dielectric constant of the air (ε_r of the air ≈ 1). This number is called the effective dielectric constant (ε_{eff}) and is dependent on the width of the signal trace (***w***), on the distance of the signal trace to the reference plane (***h***) and on the relative dielectric constant of the PCB material (ε_r). The wider the trace is the more the effective dielectric constant will approach the value of the PCB material's dielectric constant and vice versa (see the red text in the following equation). Once the effective dielectric constant is calculated, the same equation can be applied:

$$\lambda_g = \frac{c}{f\sqrt{\varepsilon_{eff}(w,h,\varepsilon_r)}}$$

Figure 11. Calculation of the guided wavelength for microstrips

This fact means that with the same PCB material, signals will always propagate faster in microstrips than in striplines. This is very important to realize when delay matching of the signals is required. If part of the signals is routed as microstrip and part as stripline, then making them the same length will not provide the same delay. That's why it is more practical to match the signals in terms of trace delay than in terms of trace length.

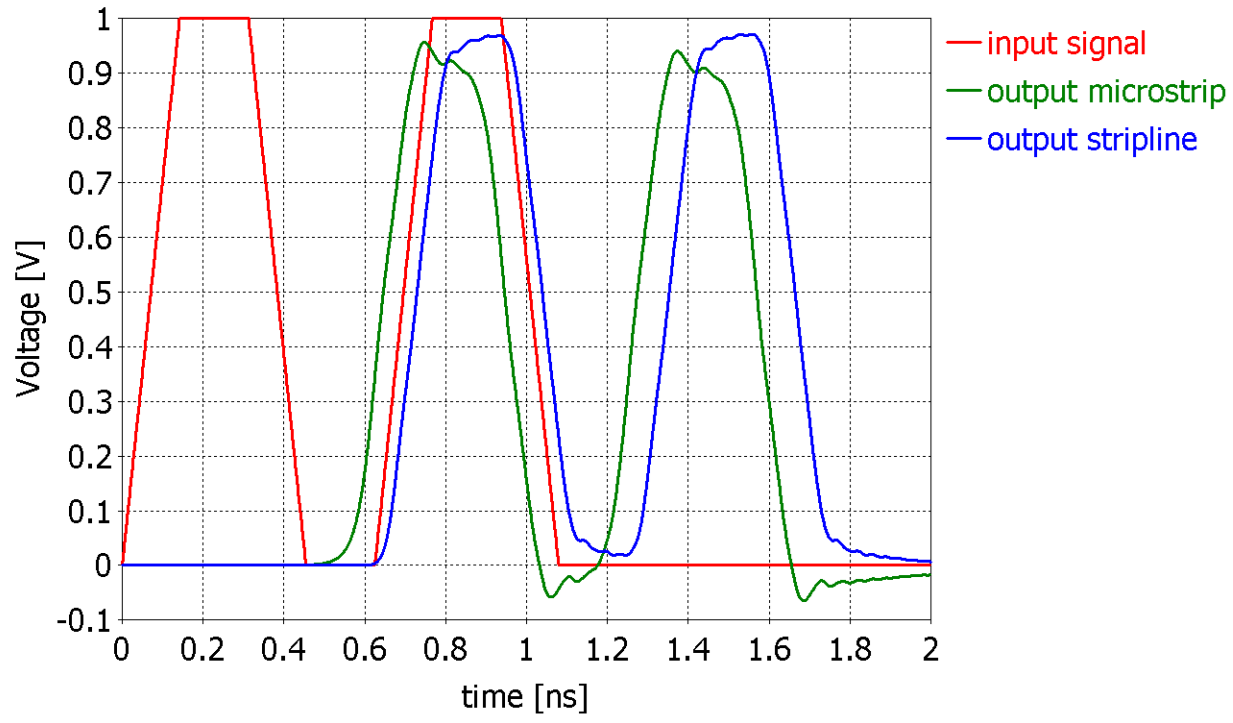

Figure 12. Propagation delay comparison for microstrips and striplines of the same length

Figure 12. on page 6 shows that the waveform propagating on the stripline arrived later than the waveform on the microstrip.

5 Characteristic impedance

Characteristic impedance is one of the most important parameters of a transmission line. It needs to be matched with the impedance of the transmitter and the receiver to avoid reflections. A practical way to approach the problem is to hide the EM field distribution problematics behind lumped elements and approximate the behavior of the transmission line by series of LC circuits.

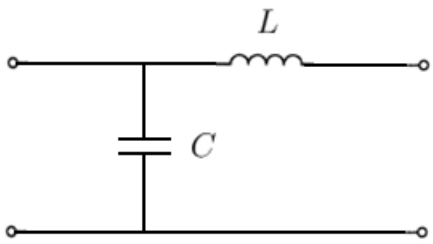

Figure 13. LC circuit segment of a transmission line model

In such setup the characteristic impedance is then calculated by the following formula [1]:

$$Z_0 = \sqrt{\frac{L}{C}}$$

Figure 14. Characteristic impedance formula

Then if the width of the trace (***w***) is increased, capacitance of the transmission line goes up and the inductance goes down. Characteristic impedance therefore goes down and vice versa if the trace width is decreased. If we decrease the height of the substrate (***h***), the capacitance goes up just like in a plate capacitor and therefore, characteristic impedance goes down and vice versa if the height is increased. Another application of plate capacitor behavior is the influence of the dielectric constant. Increase of the dielectric constant (ε_r) results in increase of the capacitance and therefore characteristic impedance goes down (and vice versa). From this point of view, it makes sense to use a material with high dielectric constant as thinner traces could be used to obtain the same characteristic impedance which would lead into PCB space savings. However, this goes against design precision demands in terms of delay matching (see Guided wavelength and propagation velocity on page 5). Therefore, compromise has to be made.

6 Referencing

The signal trace and the reference plane should never be divided in order to preserve the geometry of the transmission line and the field distribution. Ground planes (preferably) or power planes can be utilized as a reference plane. If referencing is not maintained, geometry of the transmission line drastically changes, resulting in change of the EM field distribution and its parameters. An intuitive approach is to view the problem in terms of the current return path—in case of high frequency signals the return current follows the signal trace on the reference plane. We need to make sure the current can follow the signal trace in its entire length. In addition, reference planes provide shielding from signals on adjacent layers.

NOTE

For striplines, all of the above applies to both reference planes (see Transmission line types on page 1).

6.1 Gaps or voids in the reference plane

Avoid routing the signal trace over gaps or voids in the reference plane. If there is a gap or void in the reference plane, the return current cannot follow the trace and has to flow on an alternative route around the gap. Such a discontinuity will change characteristic impedance of the trace causing distortion of the signal due to reflections and excessive radiation of the signal. Other traces in the vicinity on the same layer and on other layers then can by affected by crosstalk from this signal and the board can have troubles passing the EMC tests.

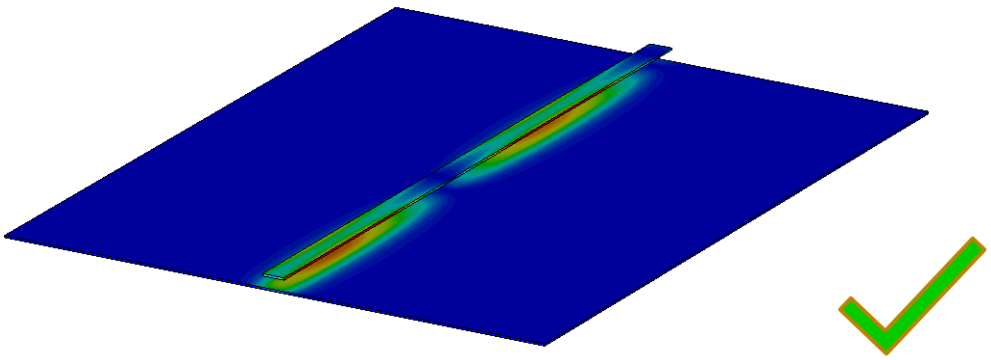

Figure 15. E-field distribution of a microstrip with proper referencing

6.1.1 Signal distortion and reflections

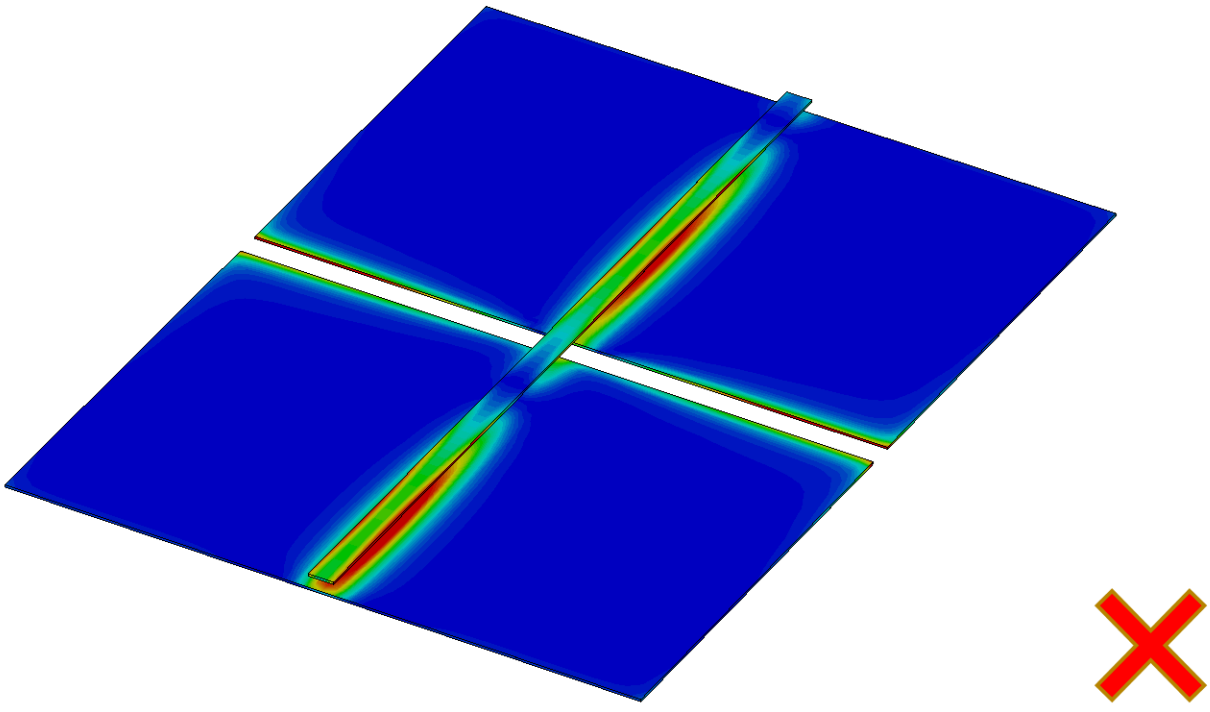

Figure 16. E-field distribution of a microstrip routed over a gap in the reference plane

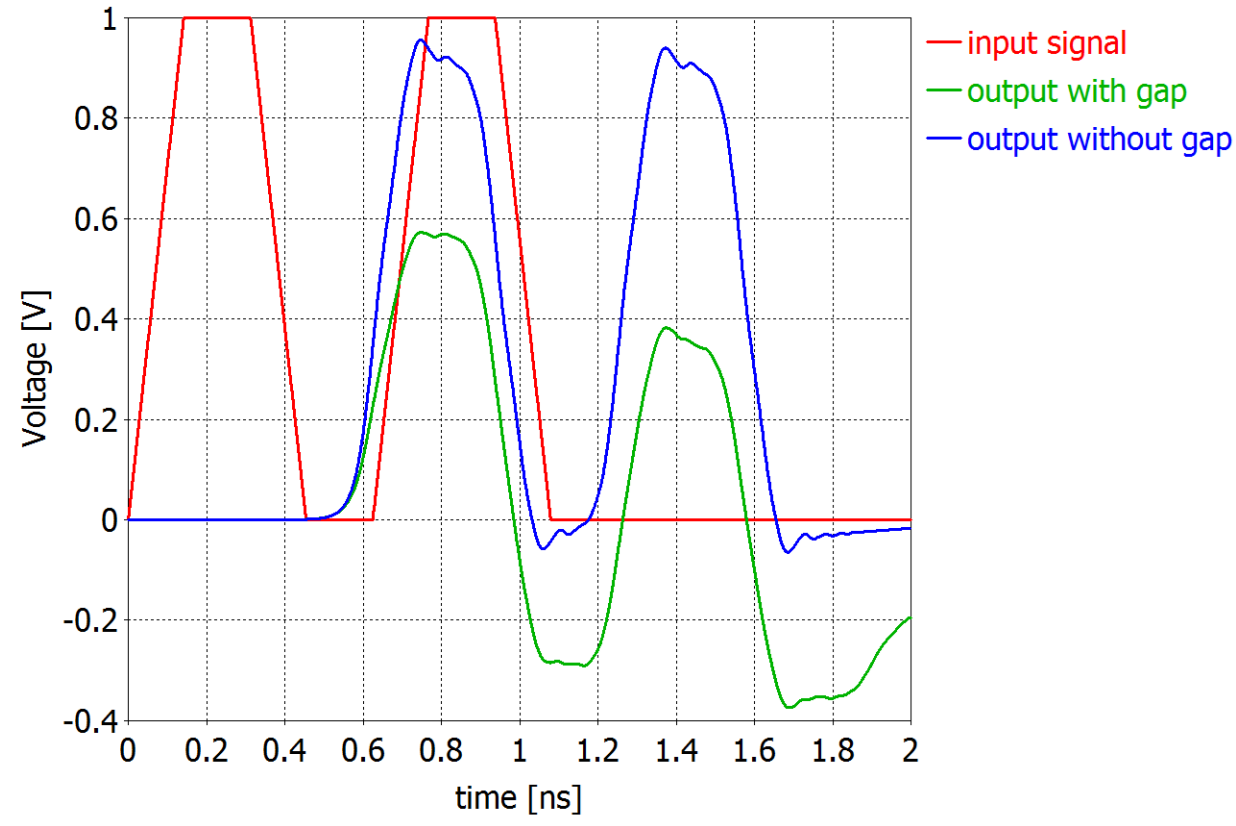

Figure 17. Comparison of the signal output—with and without a gap in the reference plane

Figure 16. on page 8 and Figure 17. on page 9 illustrate a situation when the signal trace is routed over a gap in the reference plane. Notice how E-field spreads around the gap and how the output signal degrades. The signal has lost its reference.

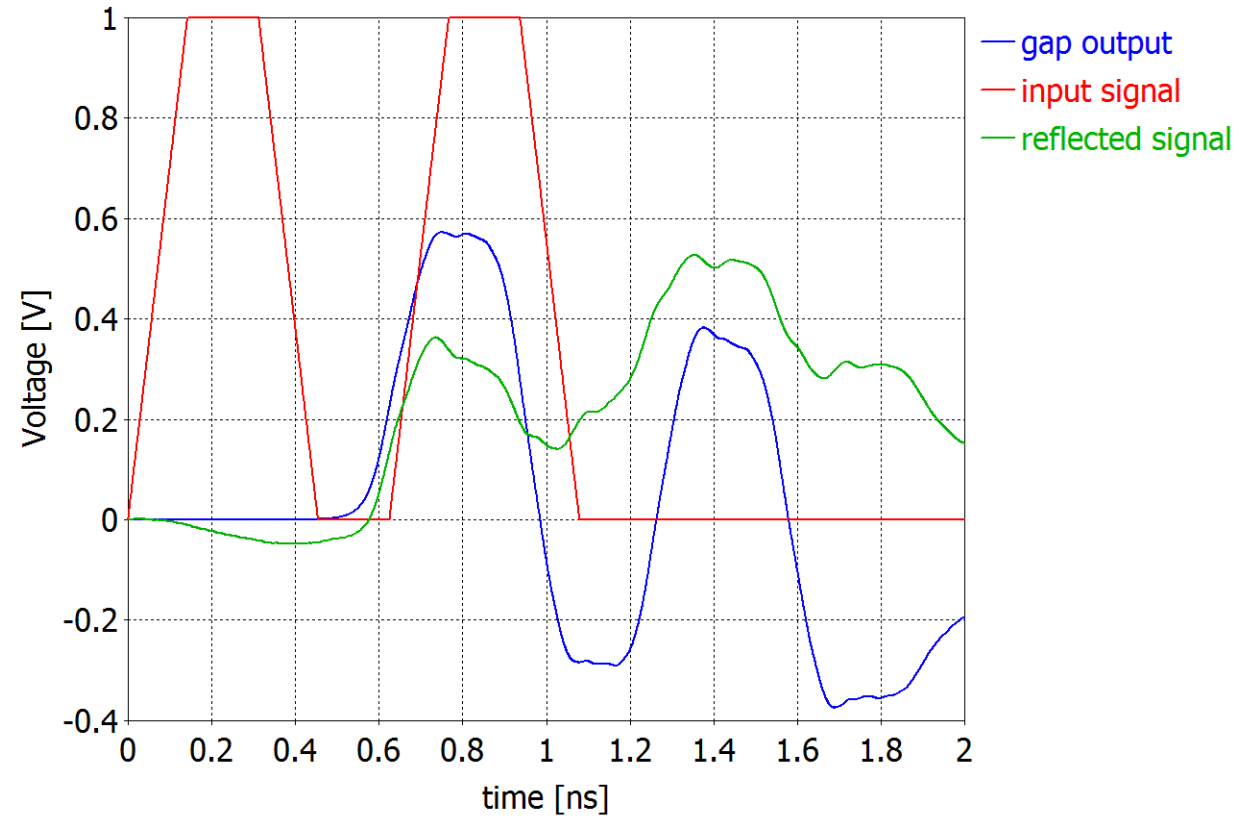

Figure 18. Reflections at the transmitter—gap

Figure 18. on page 10 illustrates how the signal reflects back to the transmitter due to the impedance inhomogeneity caused by the gap.

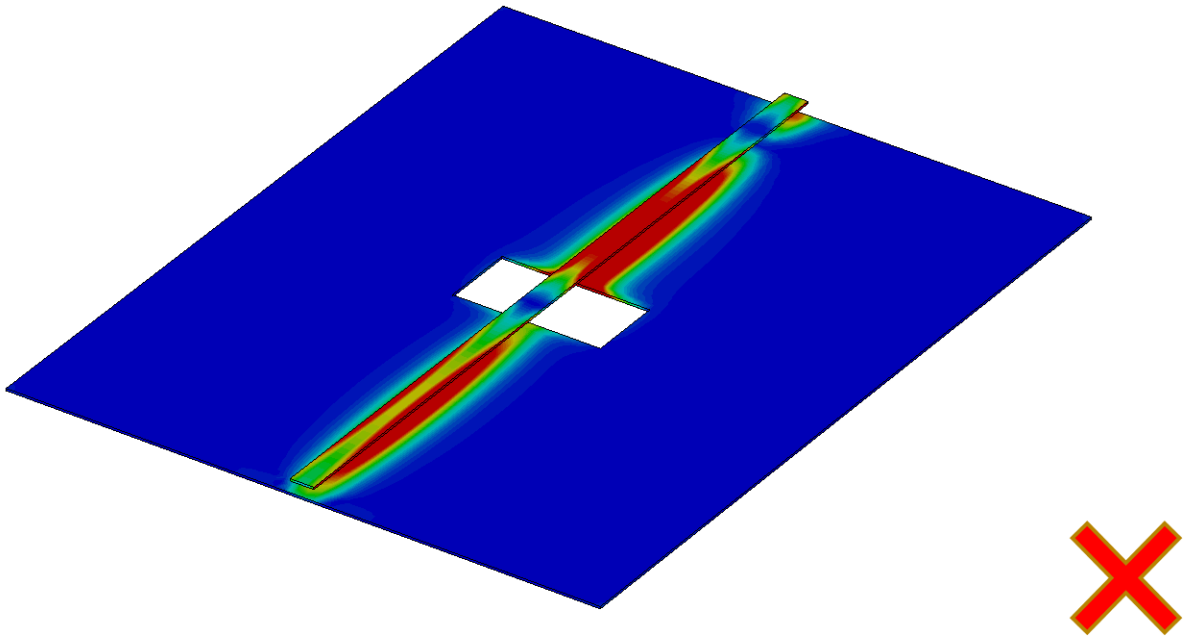

Figure 19. E-field distribution of a microstrip routed over a void in the reference plane

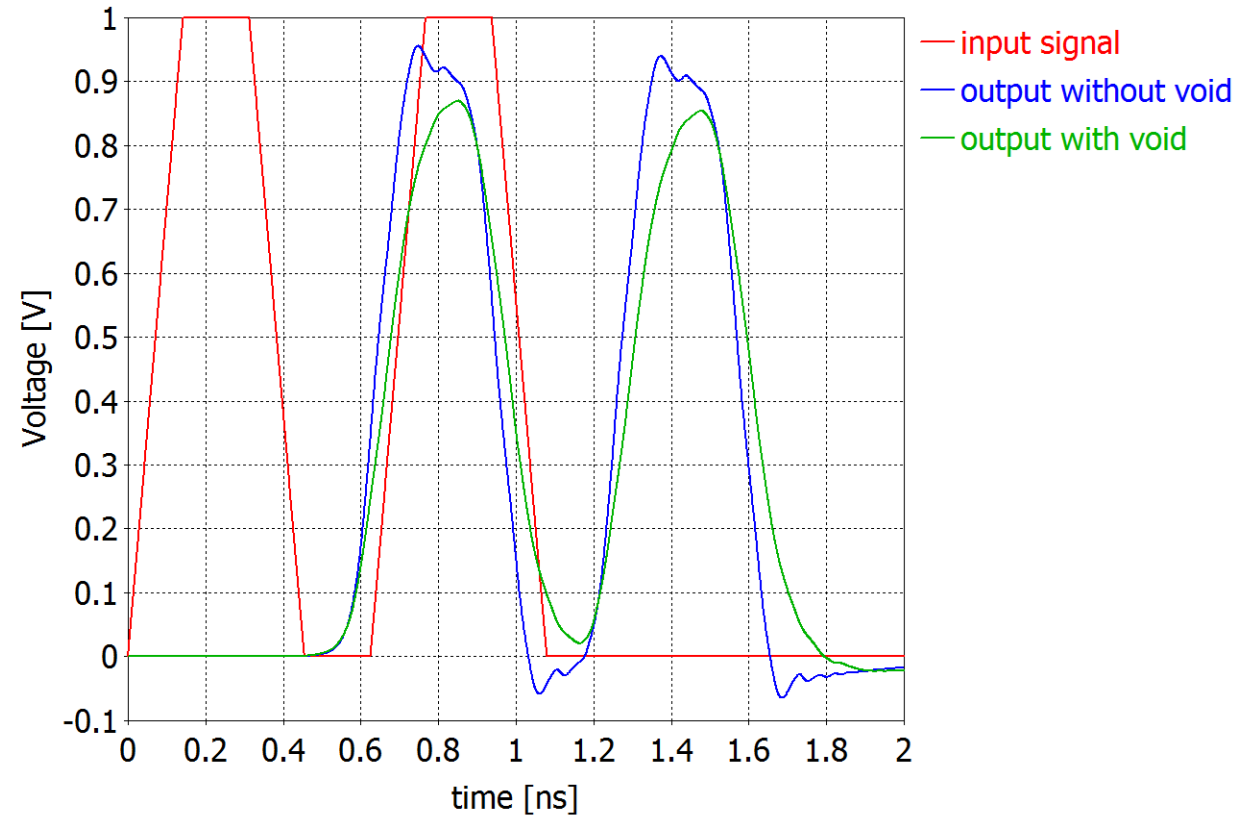

Figure 20. Comparison of the signal output—with and without a void in the reference plane

Figure 19. on page 11 and Figure 20. on page 12 illustrate a situation when the signal trace is routed over a void in the reference plane. Notice that E-field spread around the void is not that significant as in case of the gap. Signal degradation is not that severe but still considerable.

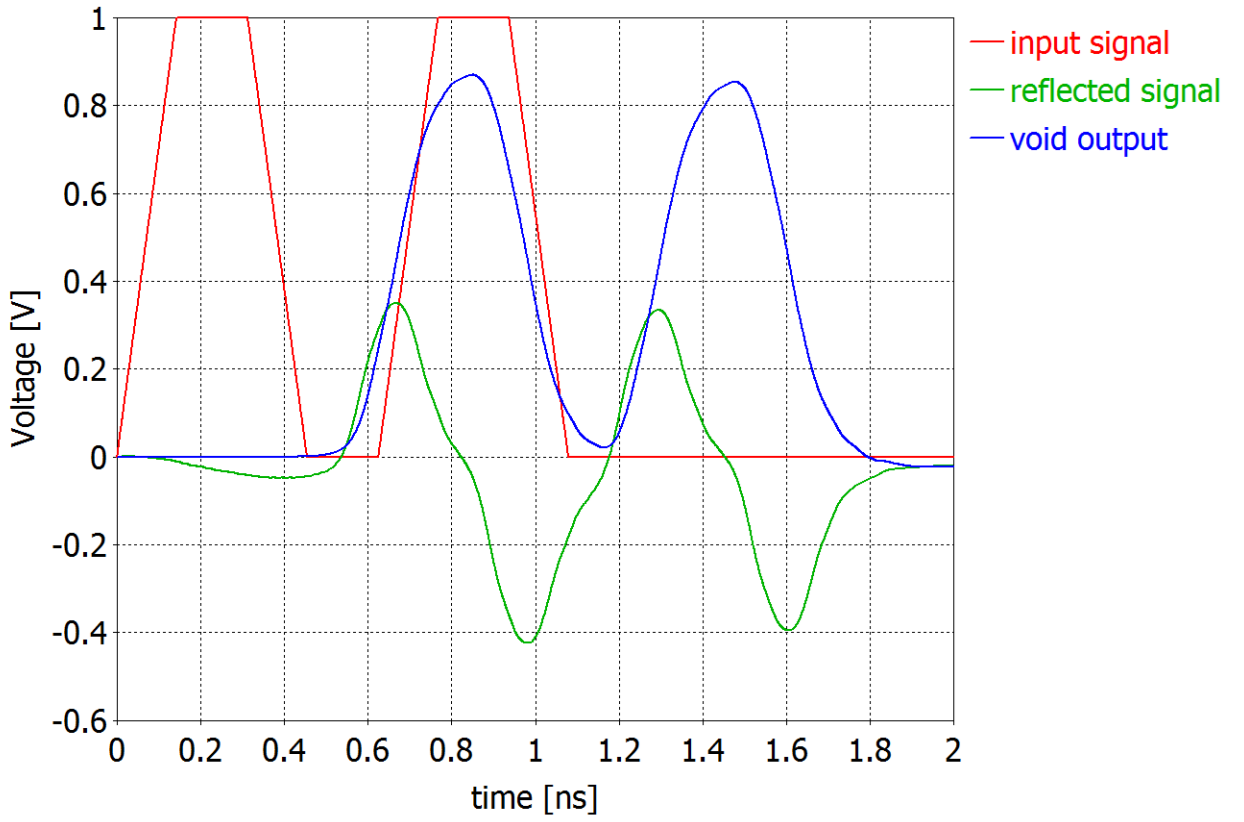

Figure 21. Reflections at the transmitter—void

Figure 21. on page 13 illustrates how the signal reflects back to the transmitter due to the impedance inhomogeneity caused by the void.

6.1.2 Crosstalk (same layer)

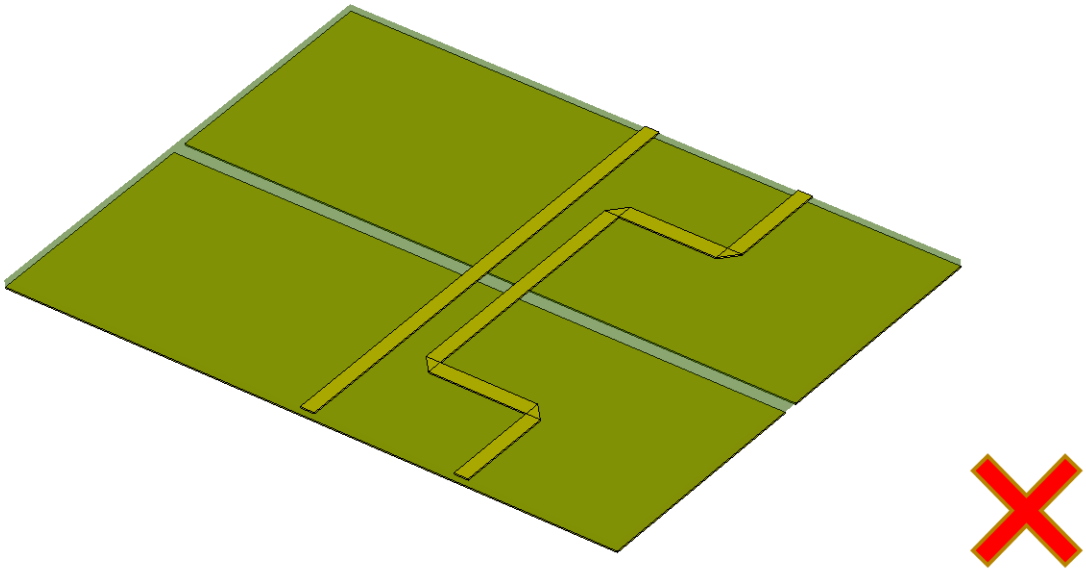

Figure 22. Two near microstrips routed over a gap in the reference plane (3 W spacing center to center)

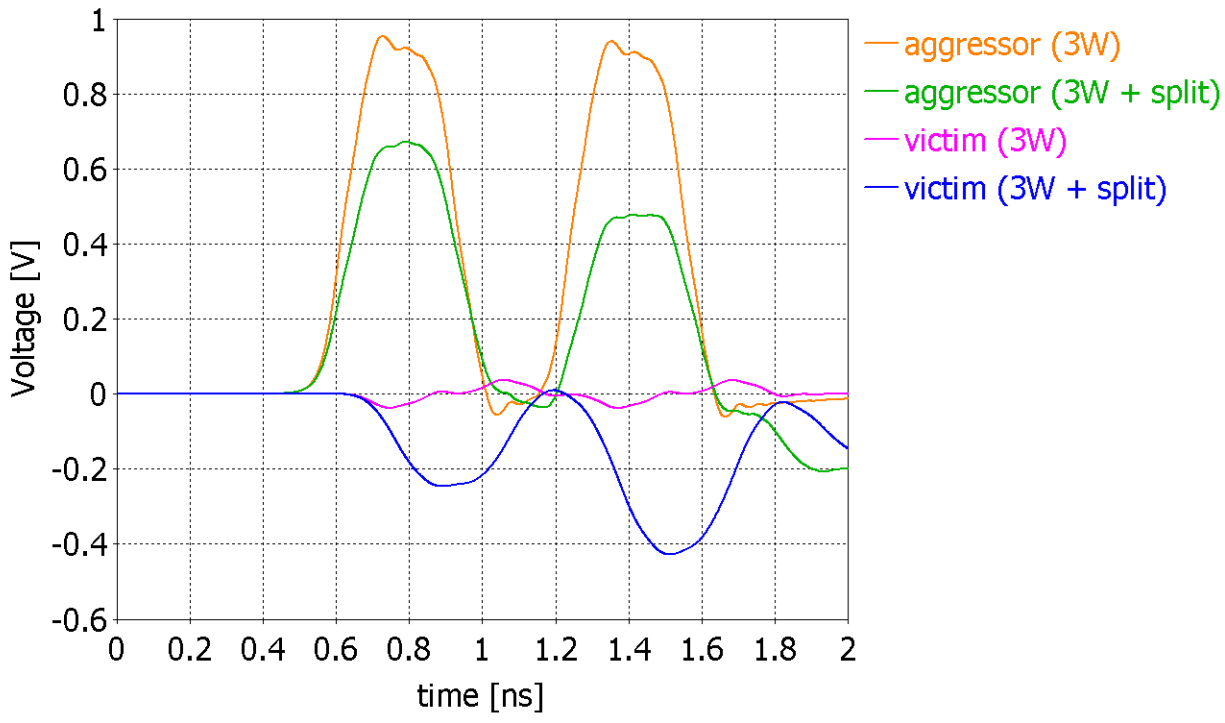

Figure 23. Comparison of crosstalk introduced to the victim (right lane) from the aggressor (left straight lane)

Figure 23. on page 14 illustrates how much of additional crosstalk can be introduced to the victim line due to parasitic radiation caused by E-field spreading around the gap in the reference plane.

NOTE

The victim signal is intentionally not driven and terminated on both sides to see only the effects of the aggressor. In ideal conditions, the voltage level at the victim's output should be 0 at all times.

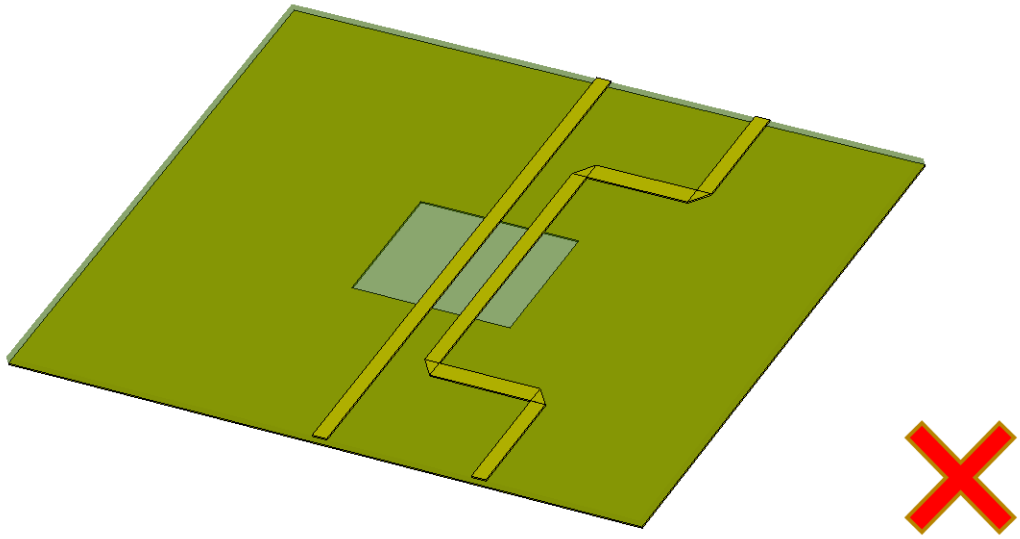

Figure 24. Two near microstrips routed over a void in the reference plane (3 W spacing center to center)

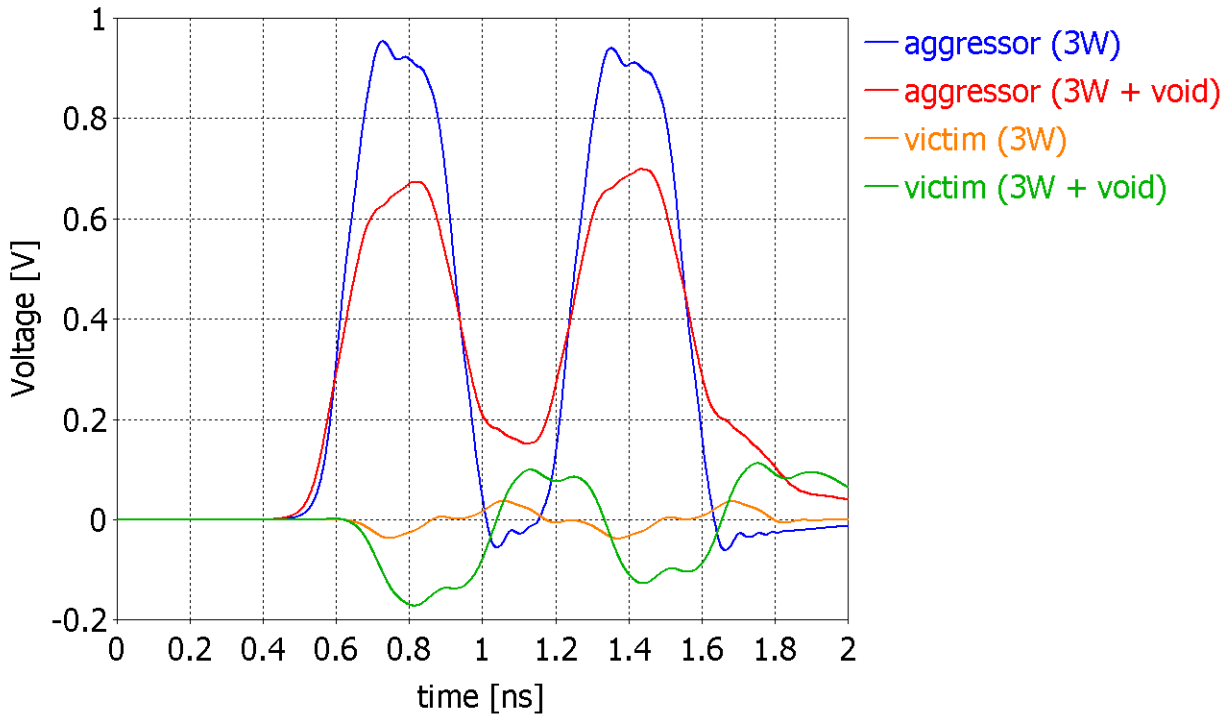

Figure 25. Comparison of crosstalk introduced to the victim (right lane) from the aggressor (left straight lane)

Figure 25. on page 15 illustrates how much of additional crosstalk can be introduced on the victim line due to parasitic radiation caused by E-field spreading around the void. Notice that the situation is better than in case of the gap however, the interference is still unacceptable.

NOTE

The victim signal is intentionally not driven and terminated on both sides to see only the effects of the aggressor. In ideal conditions, the voltage level at the victim's output should be 0 at all times.

6.1.3 Crosstalk—other layers

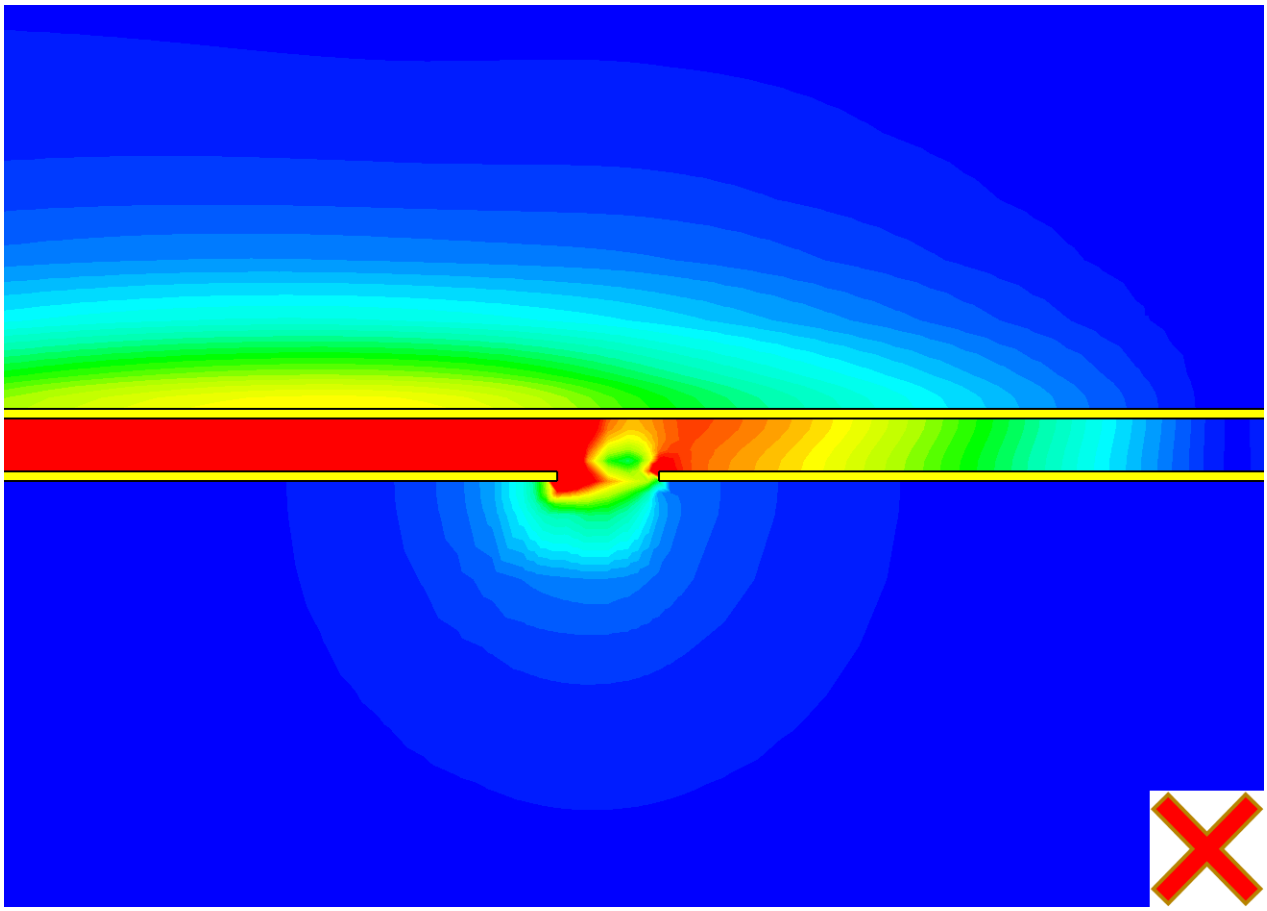

Figure 26. E-field in the vicinity of a gap—radiation of the signal trace through a gap

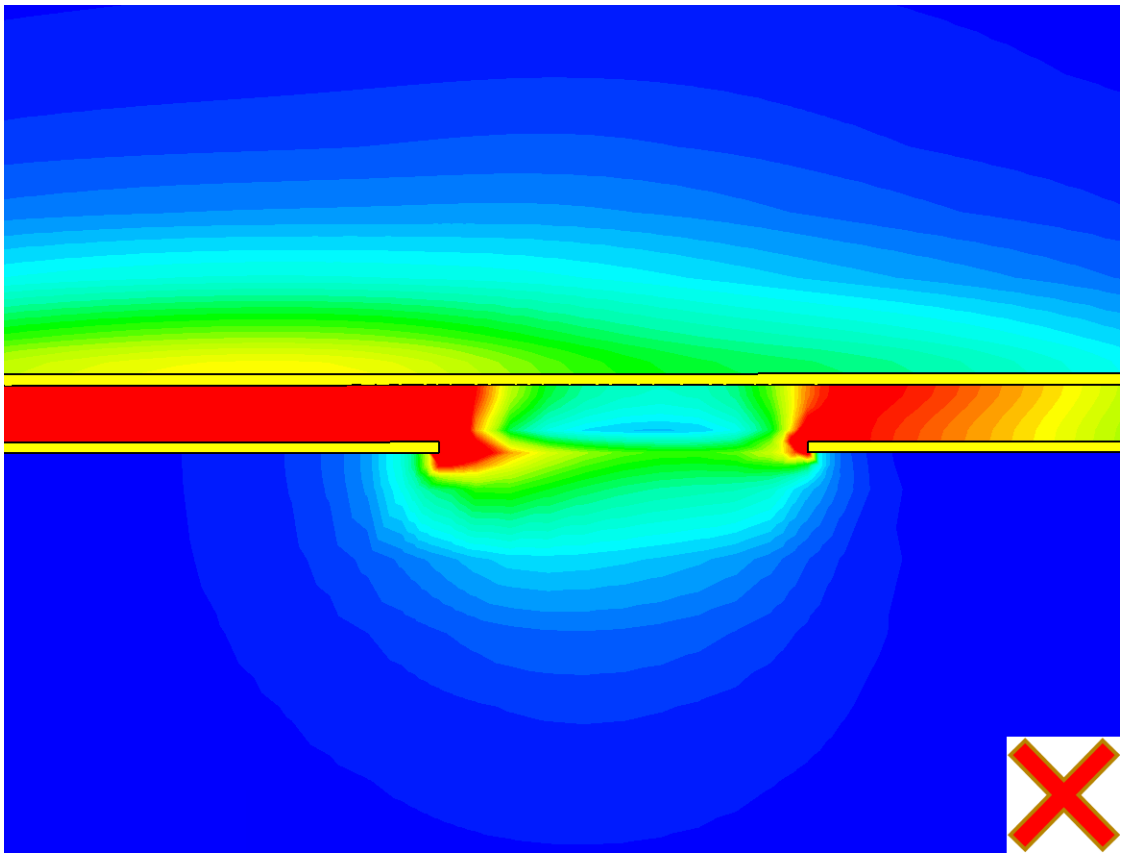

Figure 27. E-field in the vicinity of a void—radiation of the signal trace through a void

Figure 26. on page 16 and Figure 27. on page 17 illustrate how the signal trace radiates through the gap and the void in the reference plane. If there are other signals routed in the layer below, they will be affected by crosstalk. Shielding properties of the reference plane have been compromised.

6.2 Routing signal traces at the edges of reference planes

Avoid routing the signal trace at the edge of the reference plane. In such geometry the EM field distribution gets changed as it cannot be axially symmetrical in this region due to reference plane missing on one of the sides. The trace will excessively radiate causing potential EMC issues or crosstalk to the signals in other layers.

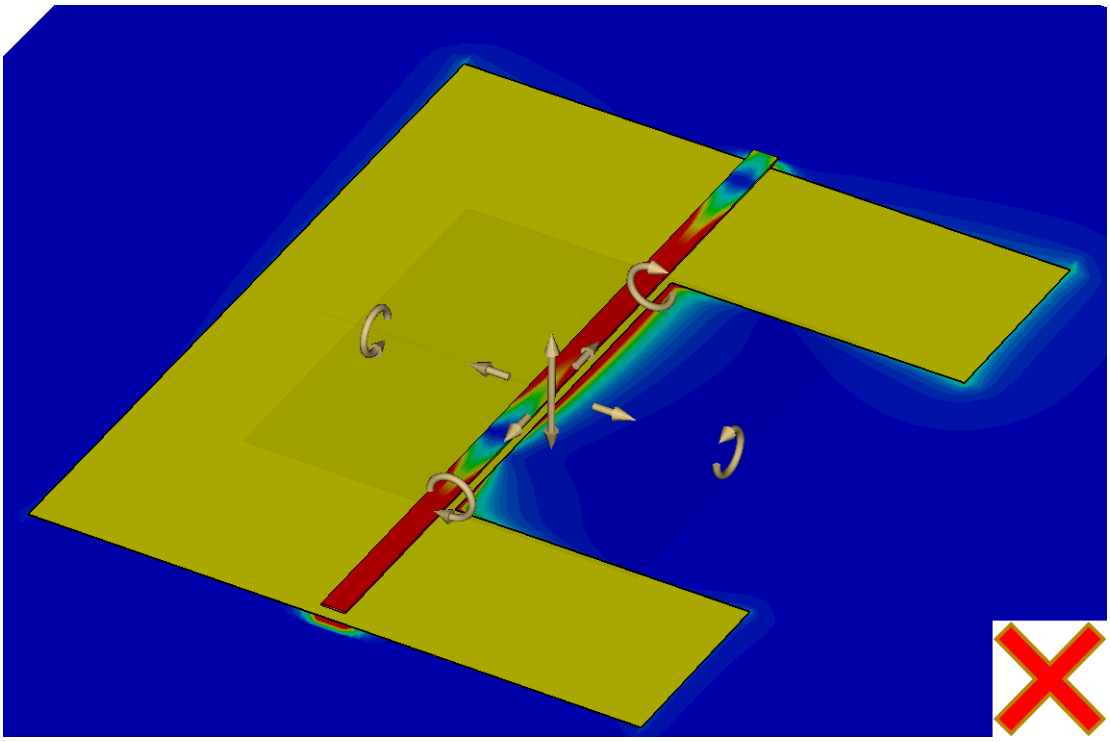

Figure 28. E-field in the vicinity of the edge—radiation of the signal trace at the edge (1)

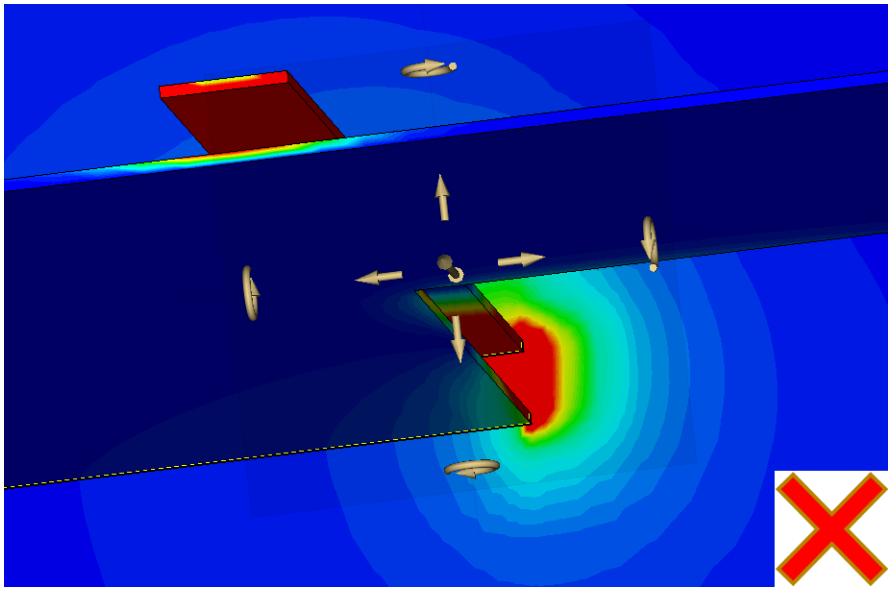

Figure 29. E-field in the vicinity of the edge—radiation of the signal trace at the edge (2)

Figure 28. on page 18 and Figure 29. on page 18 illustrate how the signal trace radiates at the edge of the reference plane. Shielding properties of the reference plane have been compromised.

6.3 Referencing in multi-layer stack-ups

When a high frequency signal is changing layers, it often passes through multiple of layers and therefore also changes the reference plane. As the current return path has to be maintained in the entire length of the signal trace, those reference planes need to be connected. In case when both reference planes are ground planes a ground stitching via is used. In case when one (or both) reference planes is a power plane, the connection has to be made by a capacitor to avoid shorts or leakage. For striplines this applies to both reference planes. Connections of the reference planes need to be located as close as possible to the signal via to preserve the geometry of the transmission line as much as possible and minimize the negative effects. Obviously, the geometry will always change to some extent with the consequences that are mentioned in Signal distortion and reflections on page 8), though the effects have much smaller magnitudes:

- Change of characteristic impedance.
- Parasitic radiation.
- Crosstalk.

That's why the number of layer changes should be reduced as much as possible.

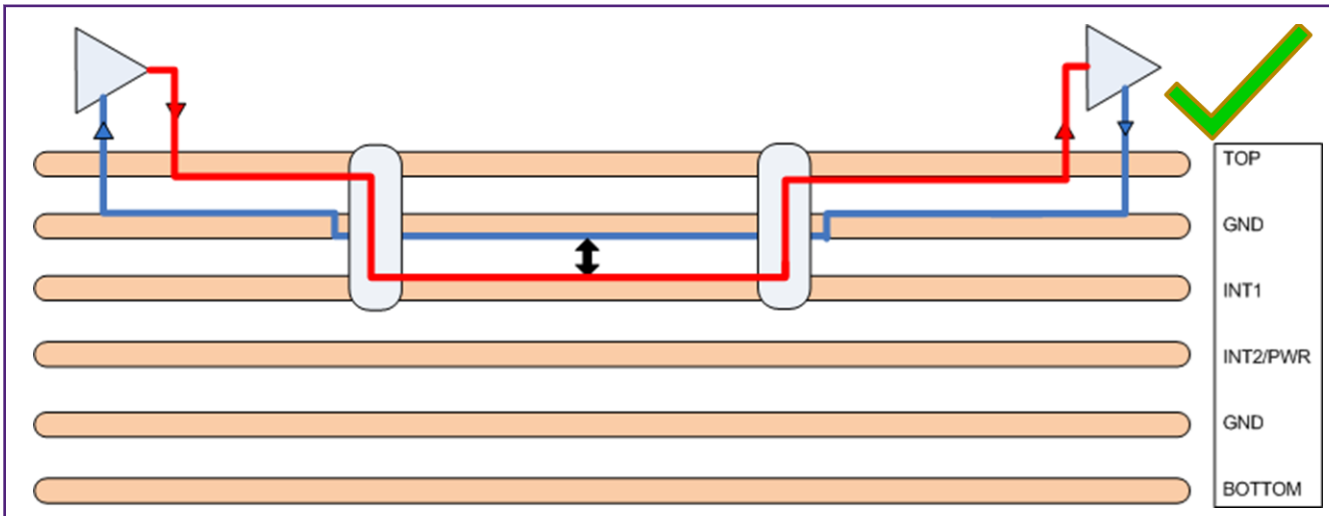

Figure 30. Layer change with the same reference layer—no stitching vias required

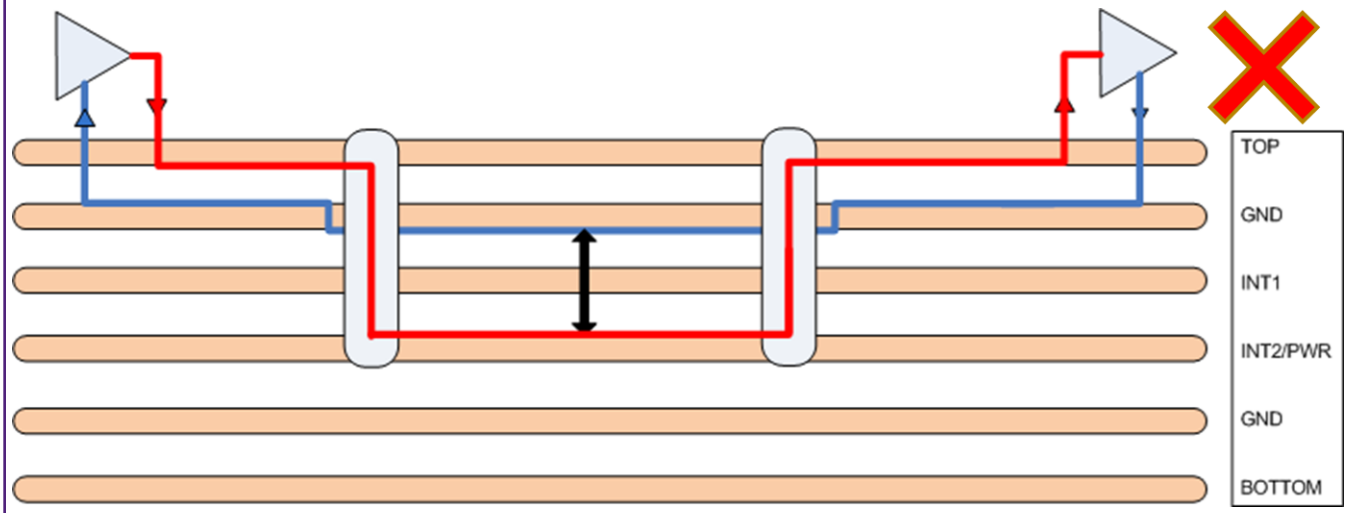

Figure 31. Layer change with switching reference planes and no stitching vias

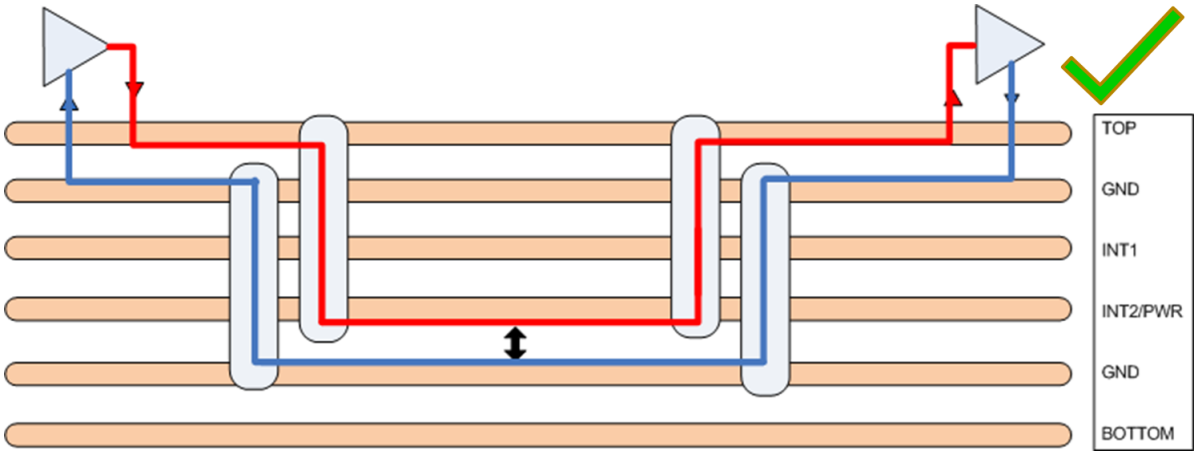

Figure 32. Layer change with switching reference plane and stiching vias in place

Figure 30. on page 19, Figure 31. on page 19, and Figure 32. on page 20 illustrate different situations when the signal is changing layers. If the same reference plane can be utilized, no stitching via is required (as seen on Figure 30. on page 19). However, if the signal cannot be referenced to the original plane because it passed through multiple of layers, stitching via is necessary. As can be seen on Figure 31. on page 19, without the stitching via, the signal will be referenced to the original reference plane after layer switch which will lead to unacceptable change in the characteristic impedance of the line (see Characteristic impedance on page 7) and in excessive interference with signals on layer INT1 (the EM field will pass through this layer.

7 Interference between transmission lines

On a PCB there are typically multiple of transmission lines on a single layer. Since their EM fields extend to much greater areas than is the width of the signal traces (see Transmission line types on page 1), the signals will interfere with each other, resulting in the following and unwanted phenomena.

7.1 Crosstalk

This issue concerns both the transmission lines—microstrips and striplines. Due to extension of the EM fields, there is coupling between the transmission lines which results in partial transition of the propagating signal into other traces i.e. crosstalk.

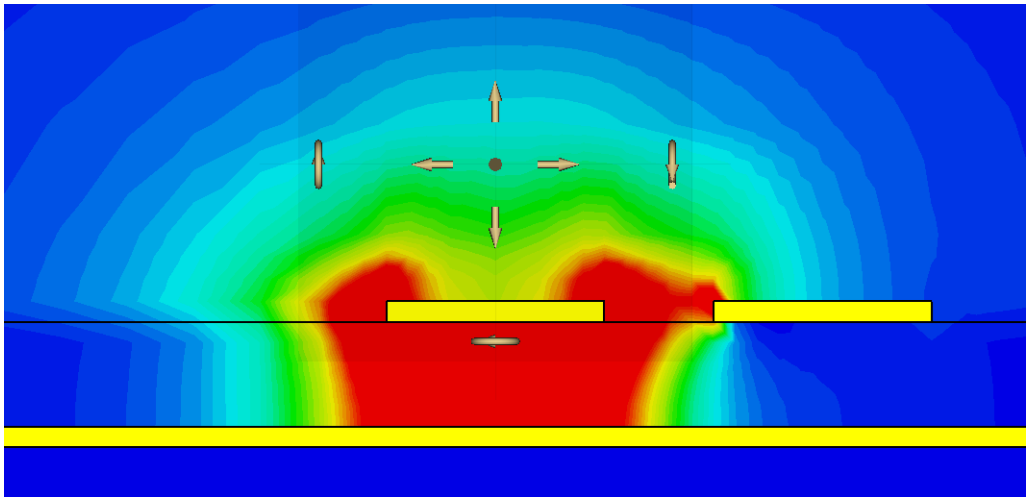

Figure 33. E-field illustration of crosstalk between two transmission lines

7.2 Increased jitter

This issue concerns only the microstrips. The EM fields of the transmission lines influence each other (coupling), causing changes in the EM field distributions. Two so called modes typically can exist between two near transmission lines:

- Even mode distribution—the signal has the same amplitude and polarity on both traces.
- Odd mode distribution—the signal has the same amplitude on both traces but the opposite polarity.

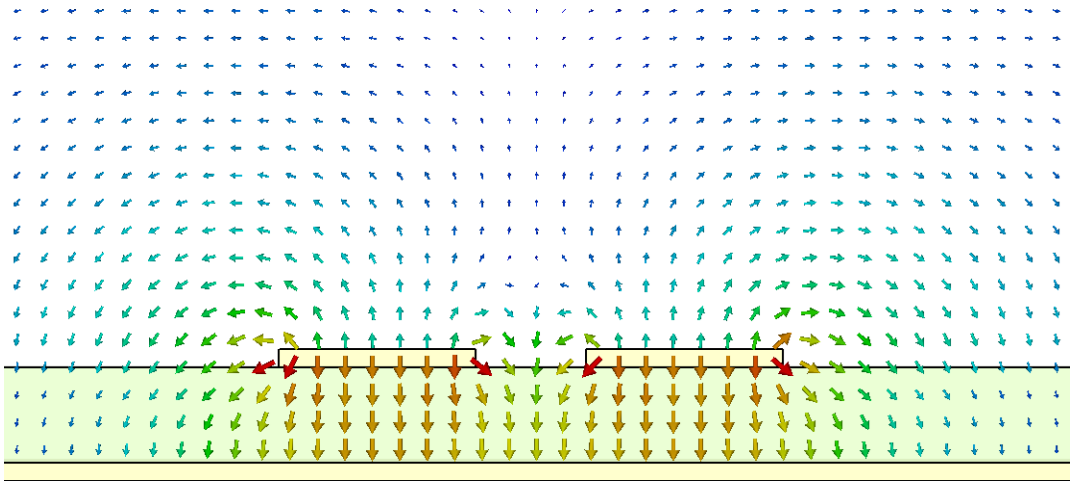

Figure 34. E-field of even mode distribution

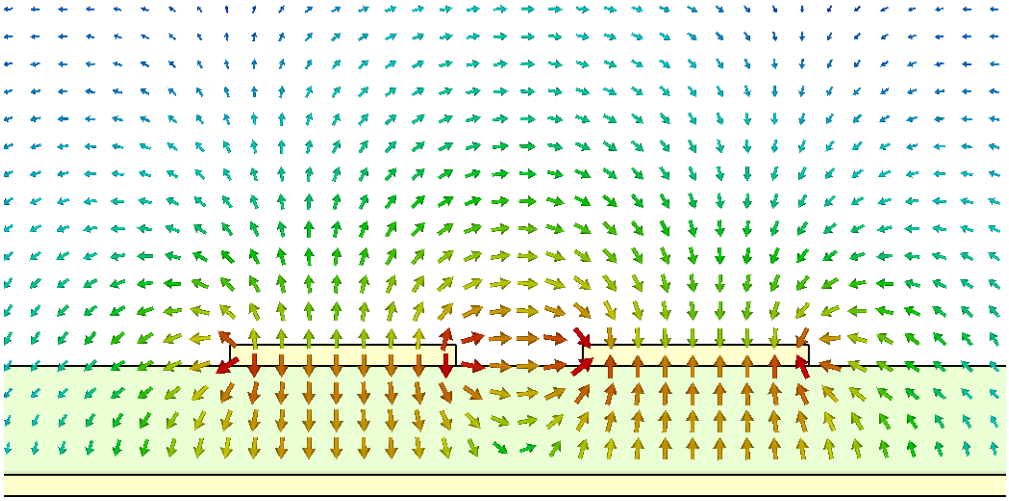

Figure 35. E-field of odd mode distribution

Since the EM field runs through two environments in case of microstrips, these distribution changes cause changes of the effective dielectric constant and in effect, guided wavelength and propagation velocity will vary (see Guided wavelength and propagation velocity on page 5). As a result, the signal propagates faster on both traces in case of the odd mode (the EM field runs more through the air) and slower in case of the even mode (the EM field runs less through the air). It is important to realize that this effect is dynamic, the modes form randomly between the traces, depending on the data patterns propagating on both transmission lines and varying voltage levels that are generally not defined as the line can belong to another interface with different specification. Therefore, the effect cannot be predicted by using reasonable means and cannot be compensated by any calibration. Some parts of the signals then propagate slower and some faster resulting in increase of the timing uncertainty at the receiver side i.e. jitter. This is a problem particularly for the clock signals..

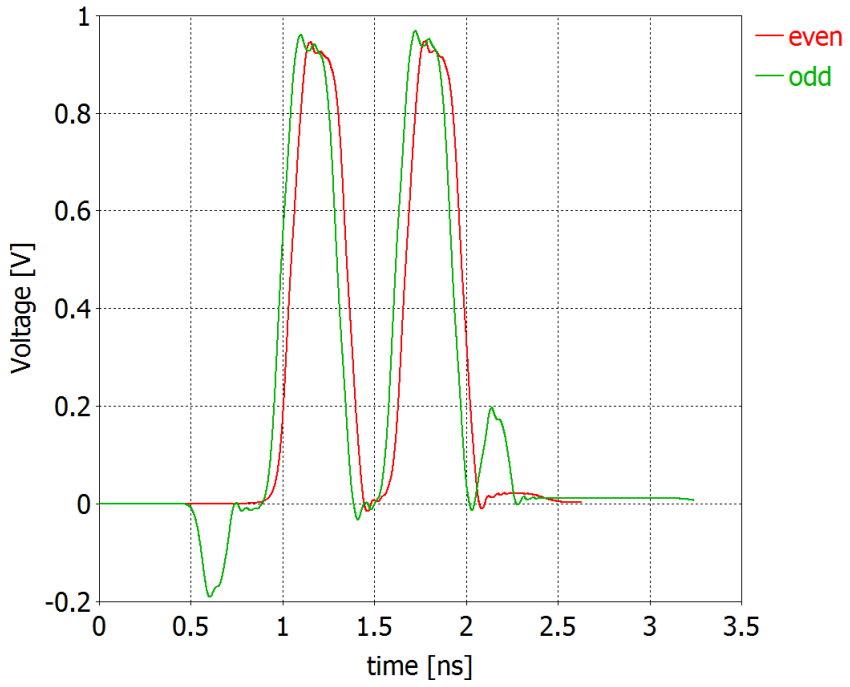

Figure 36. Timing comparison of signals affected by odd-mode or even-mode coupling

7.3 Characteristic impedance changes

This issue concerns microstrips. The EM field distribution changes discussed in Increased jitter on page 21 also lead to characteristic impedance changes and increased reflections on the transmission lines. Dynamic nature of the interference still applies and therefore nor this phenomenon can be compensated by any calibration.

7.4 Tips

Specifications of the interfaces define the timing parameters and valid voltage levels of the signals. From this data, we can derive the available budget and determine how much care needs to be taken to protect the signals from interference. Usually the most sensitive signals are clocks and strobes. Signal groups (for example a DQ byte) should be well isolated from each other. On the other hand, within a signal group we can choose a more relaxed approach because the signals usually have the same parameters and toggle at the same time.

NOTE

A good practice is not to use the entire budget and save some margin for changes caused by manufacturing variations of the devices and PCB, temperature variations, aging and EMC testing.

7.4.1 Maintain sufficient spacing

To suppress the listed phenomena to an acceptable level, sufficient spacing between the transmission lines has to be maintained. With increasing distance from the signal trace, the magnitude of the EM field decreases rapidly (see Transmission line types on page 1) and in effect also the ability of the signal to disturb others. Consider this tip also when routing meanders so the signal would not disturb itself.

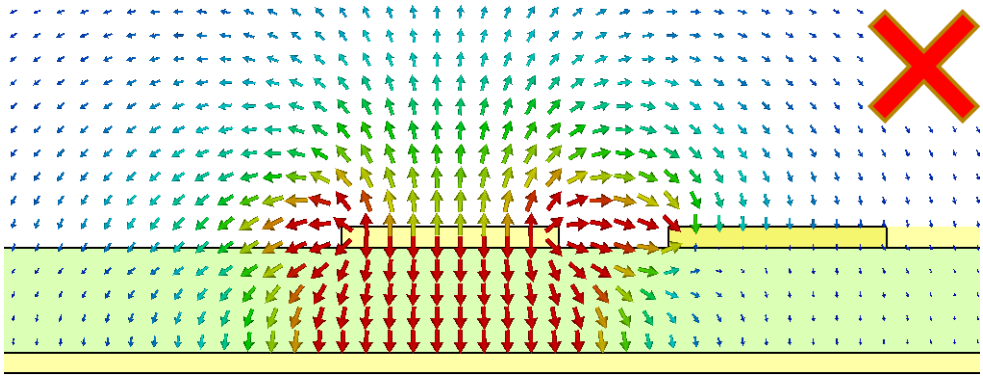

Figure 37. E-field distribution between two neighboring traces (1.5 W spacing)—vector view

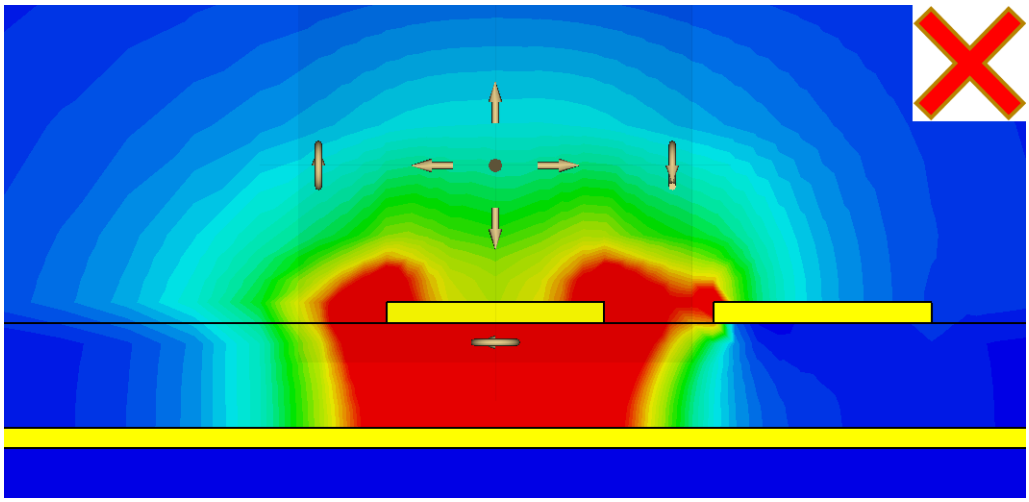

Figure 38. E-field distribution between two neighboring traces (1.5 W spacing)—magnitude view

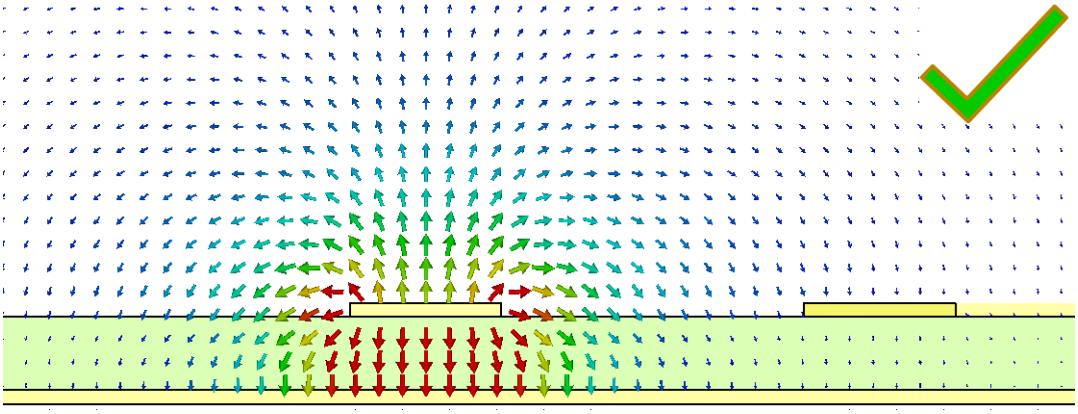

Figure 39. E-field distribution between two neighboring traces (3 W spacing)—vector view

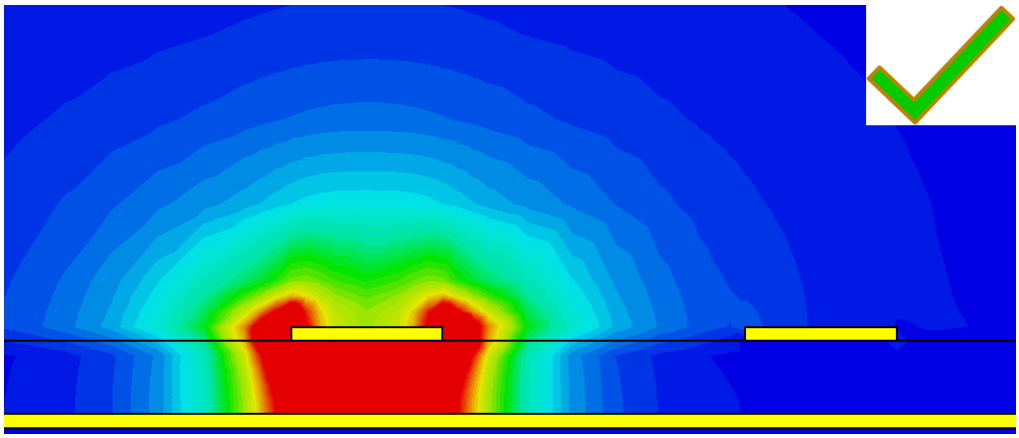

Figure 40. E-field distribution between two neighboring traces (3 W spacing)—magnitude view

Figure 37. on page 23 to Figure 40. on page 24 illustrate how the victim signal picks up interference from the aggressor and how effectively this can be avoided by increasing the spacing between the signals.

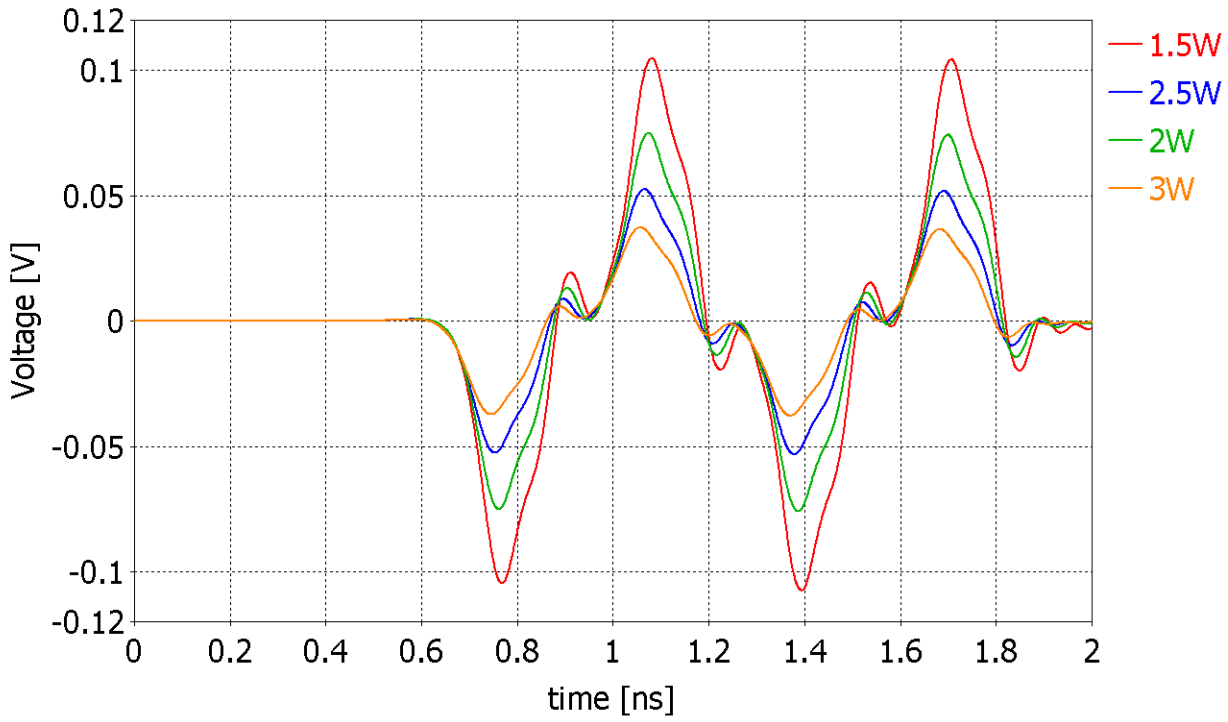

Figure 41. Crosstalk picked up from the aggressor on the victim trace—different spacings

Figure 41. on page 24 shows the waveforms that have been introduced to the victim trace due to crosstalk from the aggressor. Note how the levels decrease with increased spacing.

NOTE

The victim signal is intentionally not driven and terminated on both sides to see only the effects of the aggressor. In ideal conditions, the voltage level at the victim's output should be 0 at all times.

Other transmission lines are not the only danger. Basically, any nearby source of fast edges has to be taken into account and spacing has to be maintained. This can be for example a power plane subjected to high transient loads.

7.4.2 Minimize the length of the signal trace

The shorter the trace is, the less it will interfere with other signals and pick up interference. In addition, propagation losses will be lower, which will increase the available budget.

7.4.3 Minimize the distance where the signals are routed close to each other

The longer distance the traces run next to each other the more they influence each other. Minimizing this distance considerably reduces the interference.

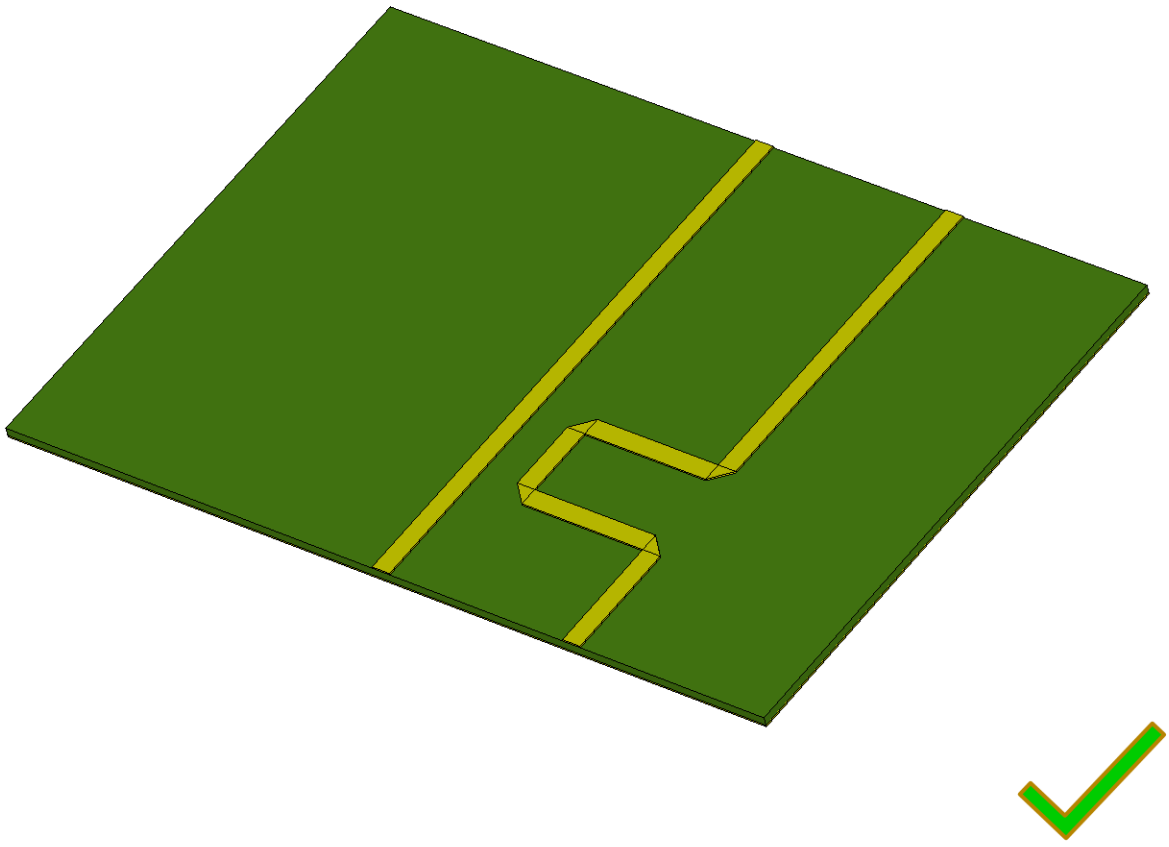

Figure 42. Distance minimized (common length 0.25 L)

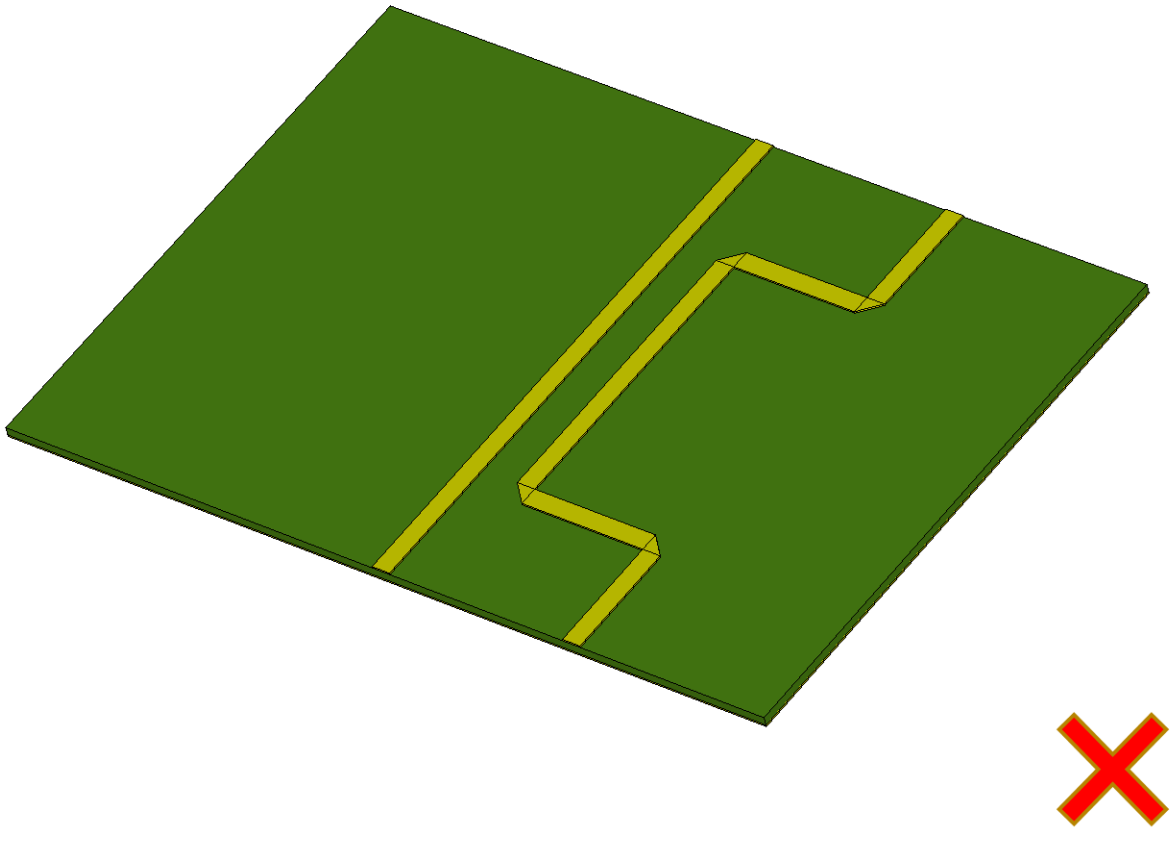

Figure 43. Distance not minimized (common length L)

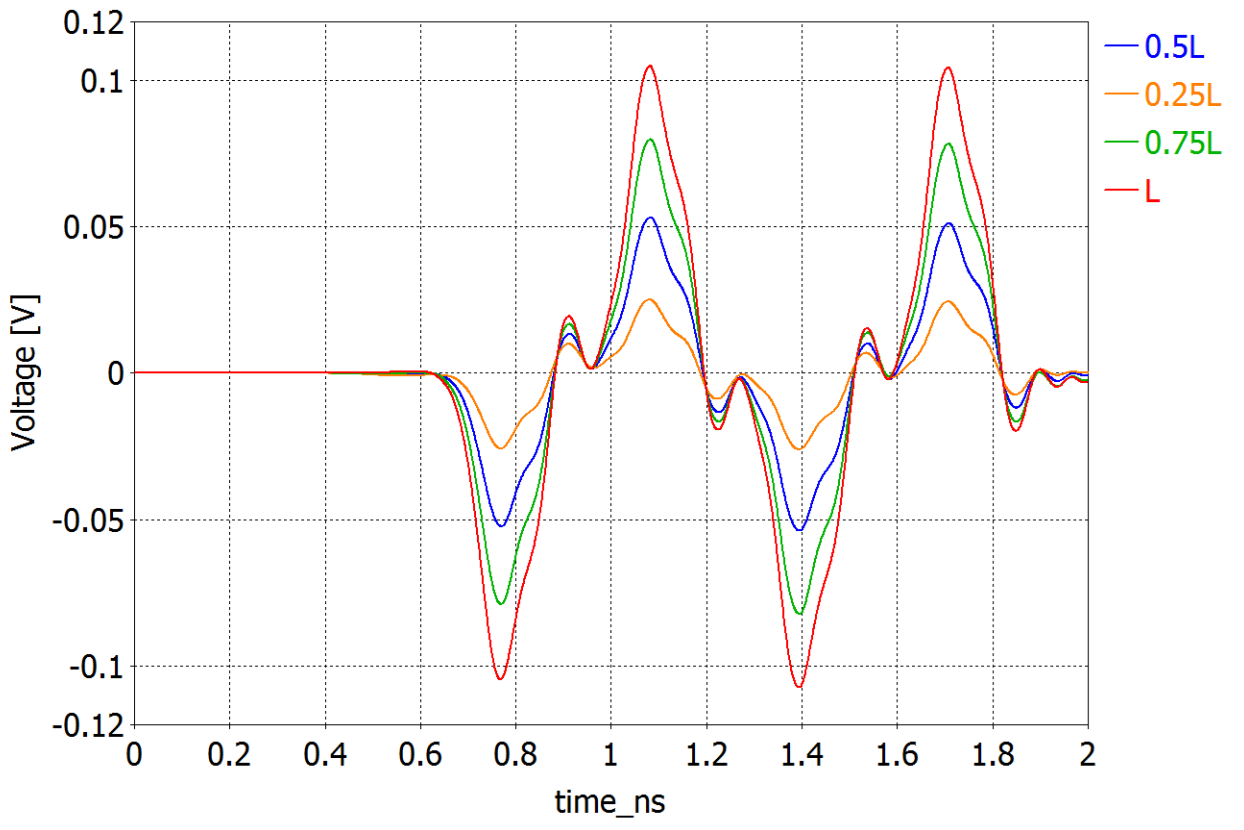

Figure 44. Crosstalk picked up on the victim trace from the aggressor—different distances routed close to each other

Figure 44. on page 27 shows the waveforms that have been introduced to the victim trace due to crosstalk from the aggressor. Note how the levels decrease with decreasing distance where the traces are routed next to each other.

NOTE

The victim signal is intentionally not driven and terminated on both sides to see only the effects of the aggressor. In ideal conditions, the voltage level at the victim's output should be 0 at all times.

7.4.3.1 Combination of increased spacing and distance minimization

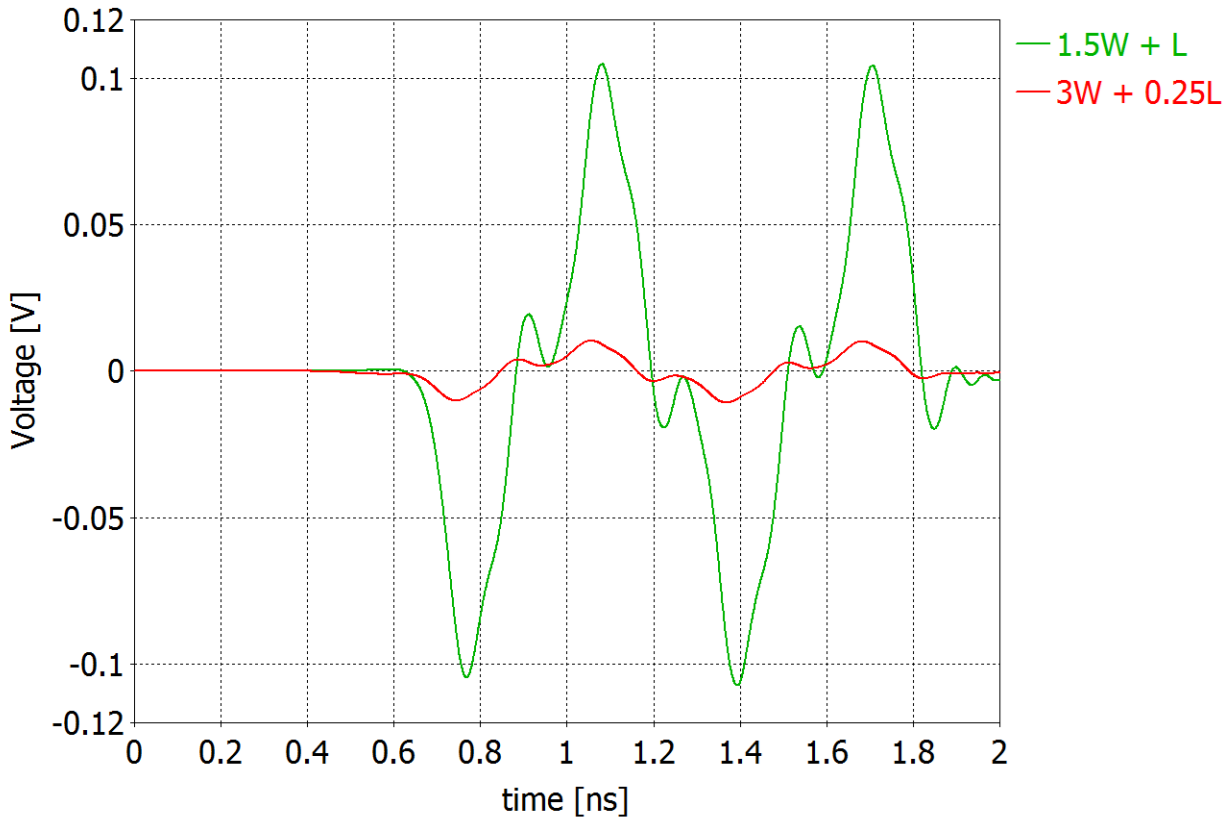

Figure 45. Combination of approaches—increased spacing and minimizing the distance

Figure 45. on page 28 compares the worst case (1.5W spacing + L common length) vs the best case (3W spacing + 0.25L common length). Combination of both techniques brings significant improvement in crosstalk reduction.

7.4.4 Adding more layers

More layers allow more space around the transmission lines as they can be routed on more layers. One limitation here could be that signal groups should be routed together.

8 Radiation

This mainly concerns microstrips. The dielectric constant of the PCB material directly affects the ability of the signal to radiate into free space. The lower the dielectric constant is, the more uniform is the EM field distribution around the trace, supporting the radiation. On the other hand, materials with higher dielectric constant better encapsulate the EM field within the PCB material, suppressing the radiation. Therefore, in general we can say that materials with low dielectric constant are more suitable for applications where radiation is desired - PCB antennas. Another reason is that the guided wavelength in such materials is longer – the antenna will have higher bandwidth and the demand on the precision of the manufacture is not that great. On the other hand, materials with high dielectric constant are more suitable where radiation is not desired (interface interconnects). From this point of view, it makes sense to use a material with a high dielectric constant. However, this goes against design precision demands in terms of delay matching (see Guided wavelength and propagation velocity on page 5). Therefore, a compromise must be made.

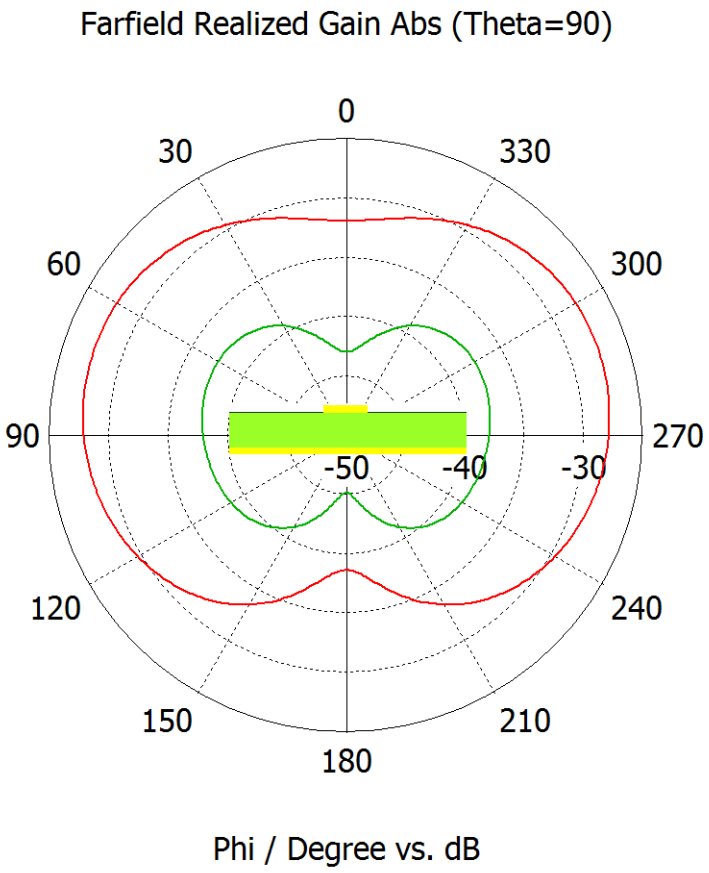

Figure 46. Comparison of the radiation patterns of two-microstrip microstrips with different dielectric constants

Figure 46. on page 29 shows the radiation patterns of two microstrips constructed with materials with two dielectric constants (ε_r = 2 and ε_r =10). In case of ε_r = 10 we can see that the overall magnitude is approximately 10 times smaller than in case of ε_r = 2 (note the magnitude is in dB).

9 Differential pairs

Differential pairs are special sub-types of the mentioned transmission lines – microstrip and stripline. All the previously discussed topics apply with one addition - the signal propagates on two signal traces with the opposite amplitude. In this case, coupling between the two traces is desired. Because the signals are synchronized, have defined voltage levels and have always the opposite polarity, only odd mode exists between the traces. This system is therefore not affected by the dynamic changes (see Increased jitter on page 21) and its parameters are stable in time. The spacing between them (**s**) is one of the dimensions that influence the main parameter of a differential pair – differential impedance.

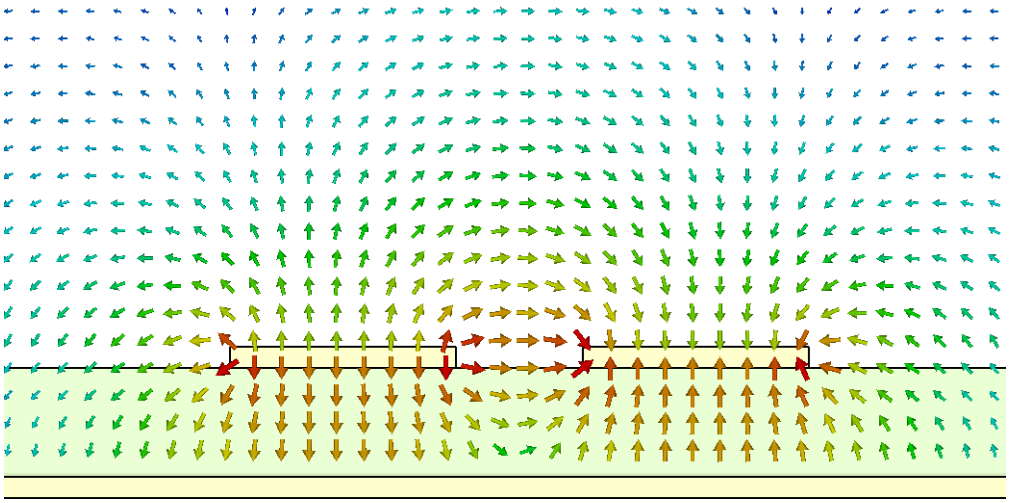

Figure 47. Typical E-field distribution of a differential pair

In order to maintain the discussed properties of this transmission line and to maximize the advantages which the differential pairs bring, the traces always have to be routed as a pair – the same spacing should be maintained in the entire length, the traces should change layers at the same place and they should have the same length (if the two previous requirements are met, making them the same length will ensure the same propagation delay on both traces).

Differential pairs are used for the most sensitive signals, like clocks and strobes. Since voltage level can be sampled as the differential between the two traces, differential pairs are much more resilient to interference – the signal picks up similar interference on both traces and therefore the differential value is not influenced. In addition, the voltage differential between the two logical states is doubled in comparison to a single ended line (+V/-V vs. +V/0V). Therefore, signal to noise ratio is better. A disadvantage is that such a geometry consumes much more space on the PCB than traditional single-ended transmission lines. Differential pairs are therefore used for data transmissions only in cases where there are only few data lanes – PCIe, USB, SATA etc.

10 Other routing considerations

10.1 Signal groups

A signal group wraps signals that belong to each other. This can be for example a DQ byte in the DRAM interface, consiting of 8 DQ signals, DM signal and the assosiated DQS (strobe). Signals within a signal group should always be routed together on the same layer in order to ensure minimum timing differences (see Guided wavelength and propagation velocity on page 5 and Interference between transmission lines on page 20).

10.2 Turns

If the trace is required to be curved due to layout or matching requirements (meanders), it should be done as smoothly as possible to avoid reflections and parasitic radiation. The effect on the signal is not that significant, however there are usually multiple turns for each signal on the PCB and the effects cumulate.

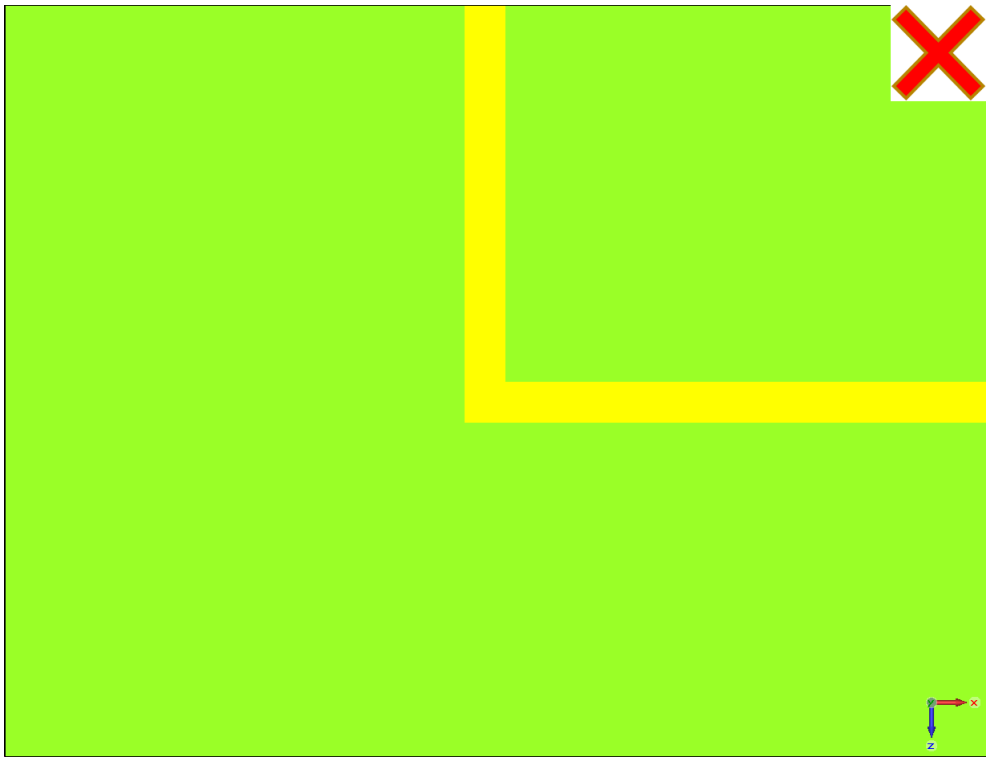

Figure 48. Sharp edge turn

Multiple ways are possible:

- 45° turn:

Figure 49. 45° turn

- Turn with a chamfered edge:

Figure 50. Turn with a chamfered edge

- Rounded turn:

Figure 51. Rounded turn

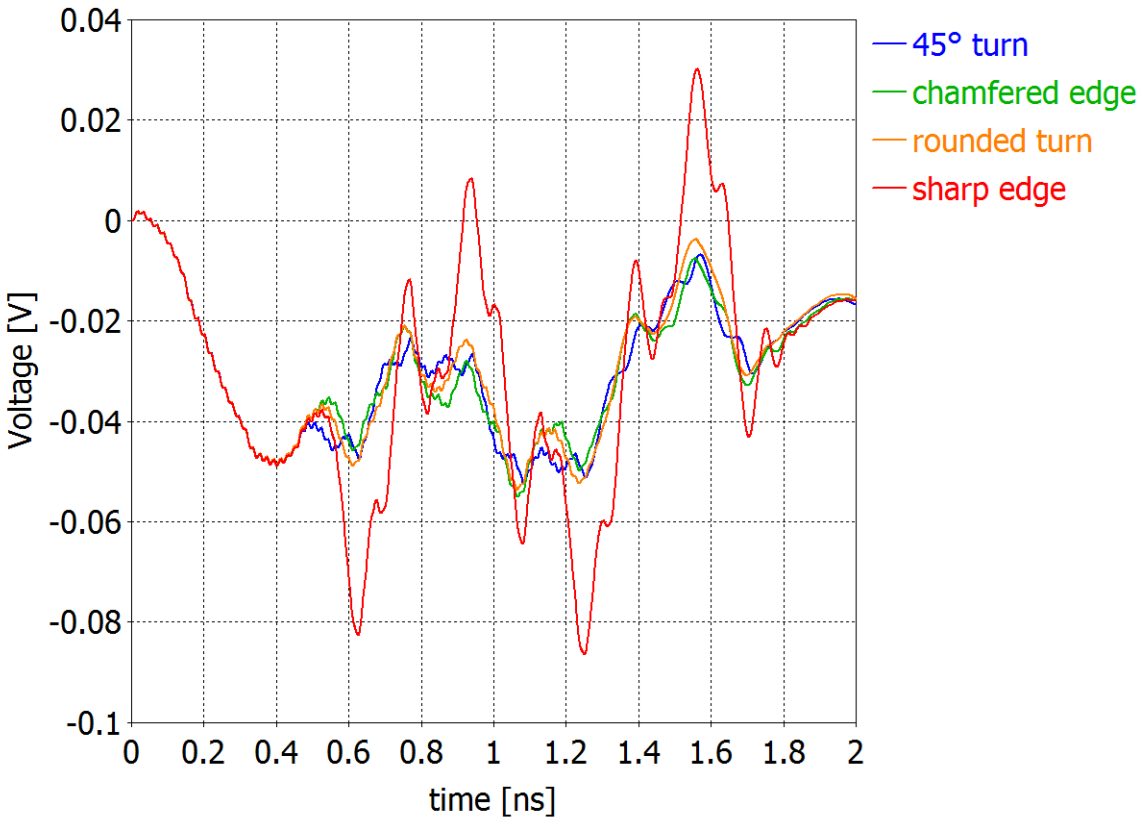

Figure 52. Comparison of turn techniques

The signal reflections are compared in Figure 52. on page 34. All the methods bring a significant improvement when compared to the sharp edge turn.

11 Simulations

From the previous chapters it is obvious that there are many aspects to consider when designing a PCB with lots of high frequency signals and in many cases, it is almost impossible for human eye to take into account all the phenomena. It is therefore recommended to perform simulations to validate the finished design to see if there are any overlooked issues.

The recommended tools for this purpose are Hyperlynx and Power-SI. For more complex structures where the smallest details need to be considered a full 3D solvers can be used, like CST Microwave Studio, which was used to create the data in this application note.

12 Used models

Models of transmission lines (microstrip and stripline) were used to create the figures. The simulations were performed using CST Microwave Studio (using the transient solver).

12.1 Microstrip and stripline models

These models were used to create the figures in Transmission line types on page 1, Guided wavelength and propagation velocity on page 5, and Radiation on page 28.

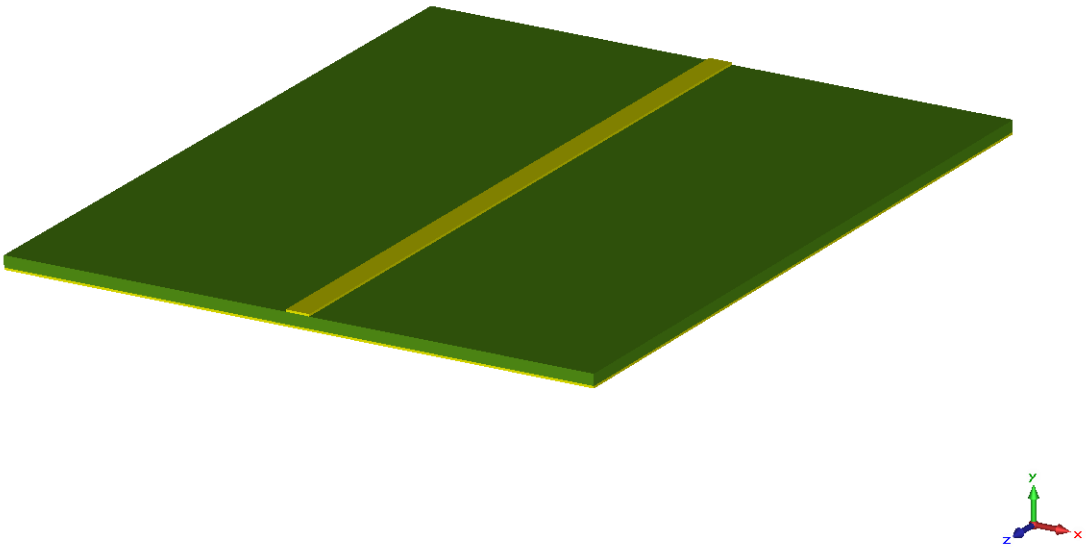

Figure 53. Microstrip model

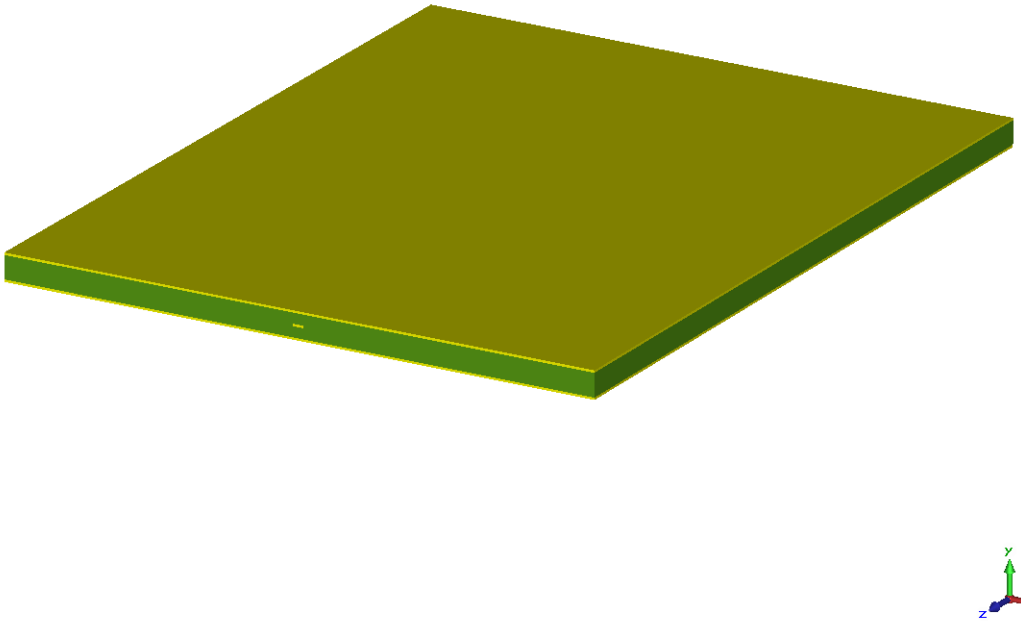

Figure 54. Stripline model

12.2 Reference plane models

These models were used to create the figures in Gaps or voids in the reference plane on page 8.

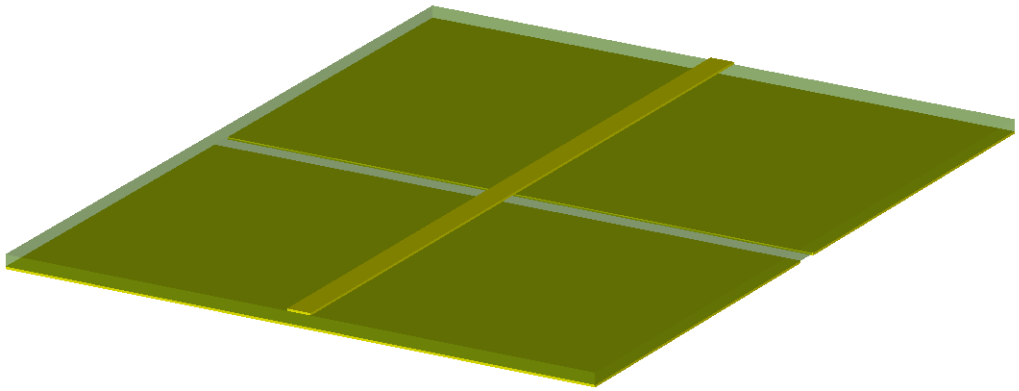

Figure 55. Reference plane gap model

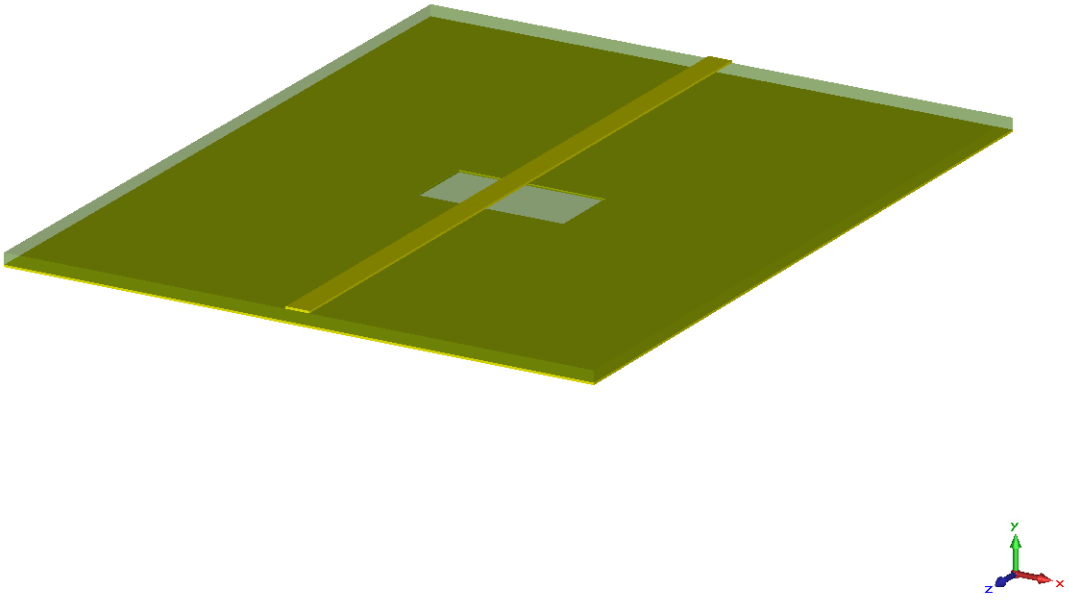

Figure 56. Reference plane void model

12.3 Cut edge model

This model was used to create the figures in Routing signal traces at the edges of reference planes on page 17.

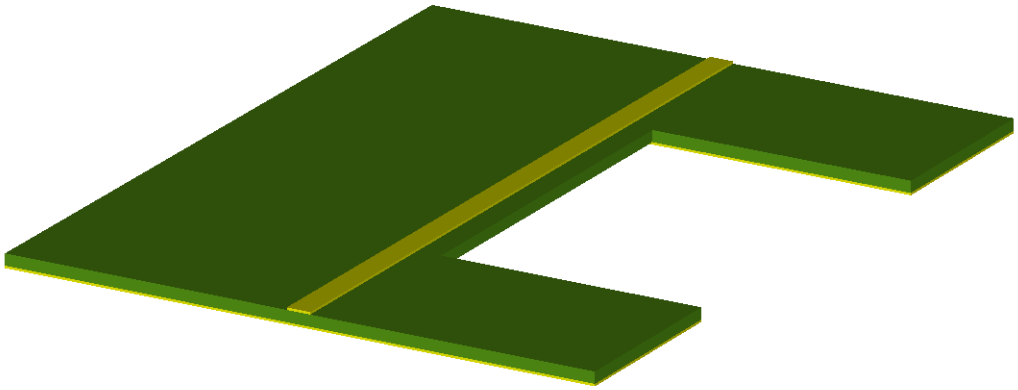

Figure 57. Cut edge model

12.4 Crosstalk models

These models were used to create the figures in Gaps or voids in the reference plane on page 8, Crosstalk on page 20, and Tips on page 22.

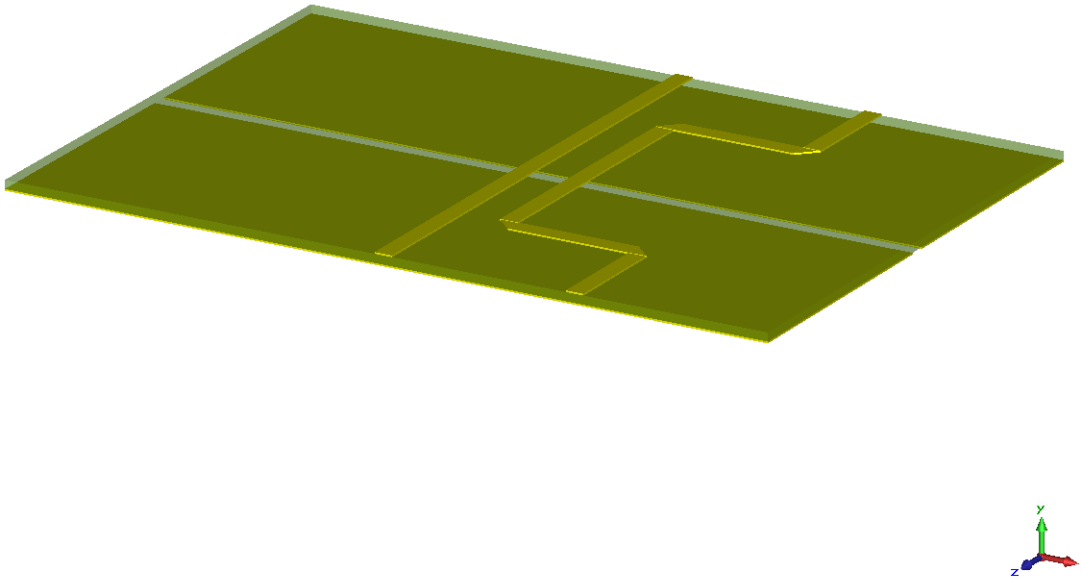

Figure 58. Crosstalk model with a gap in the reference plane

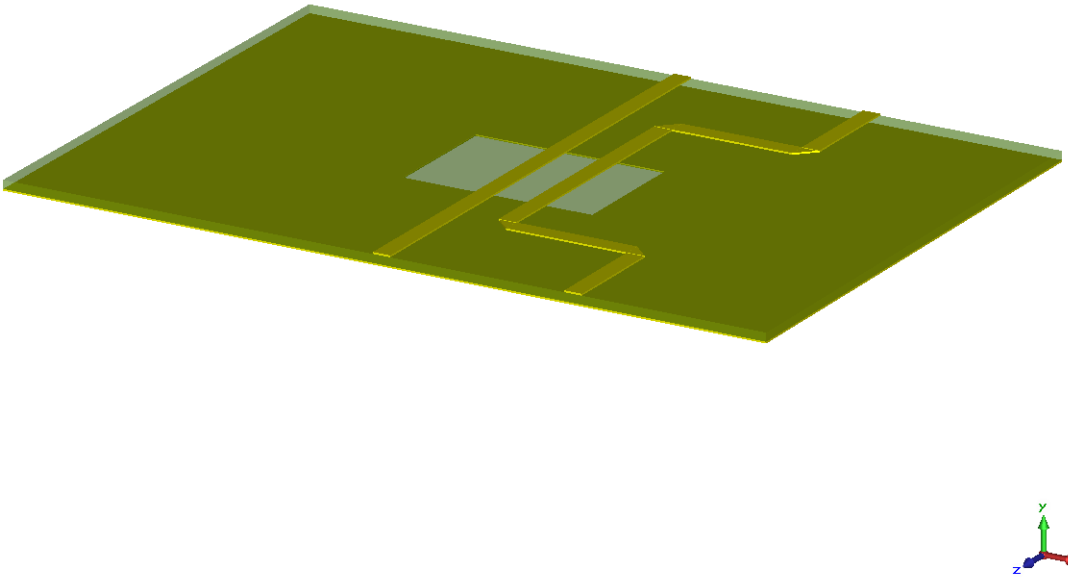

Figure 59. Crosstalk model with a void in the reference plane

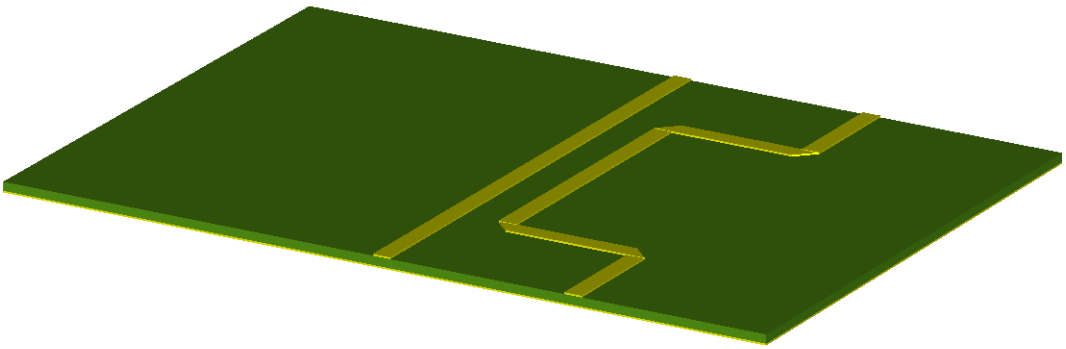

Figure 60. Crosstalk model with a solid reference plane

12.5 Jitter model

These models were used to create the figures in Increased jitter on page 21 and Differential pairs on page 29.

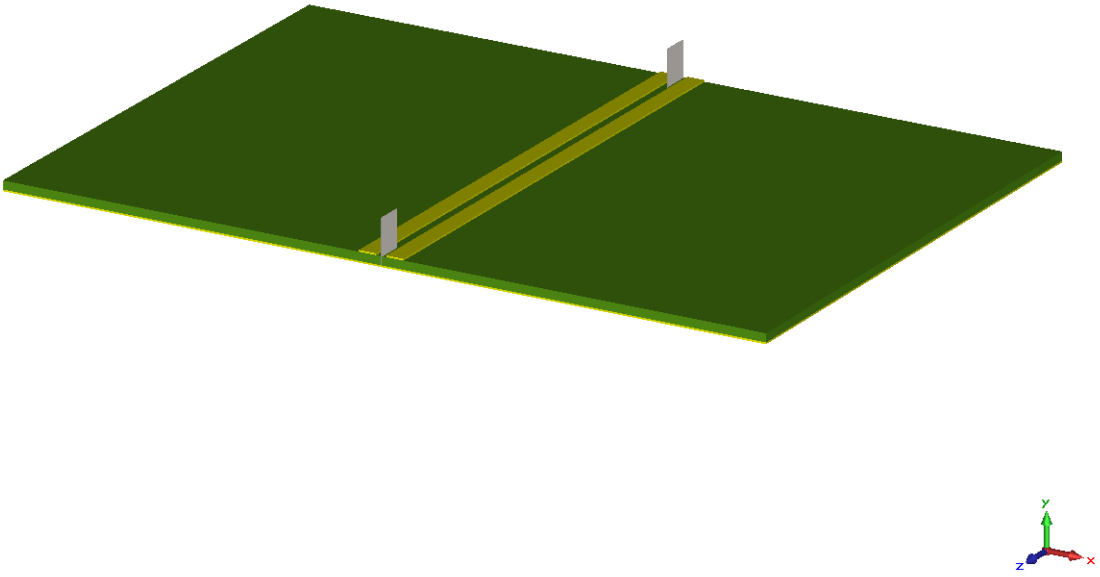

Figure 61. Jitter model

12.6 Turns model

This model was used to create the figures in Turns on page 30.

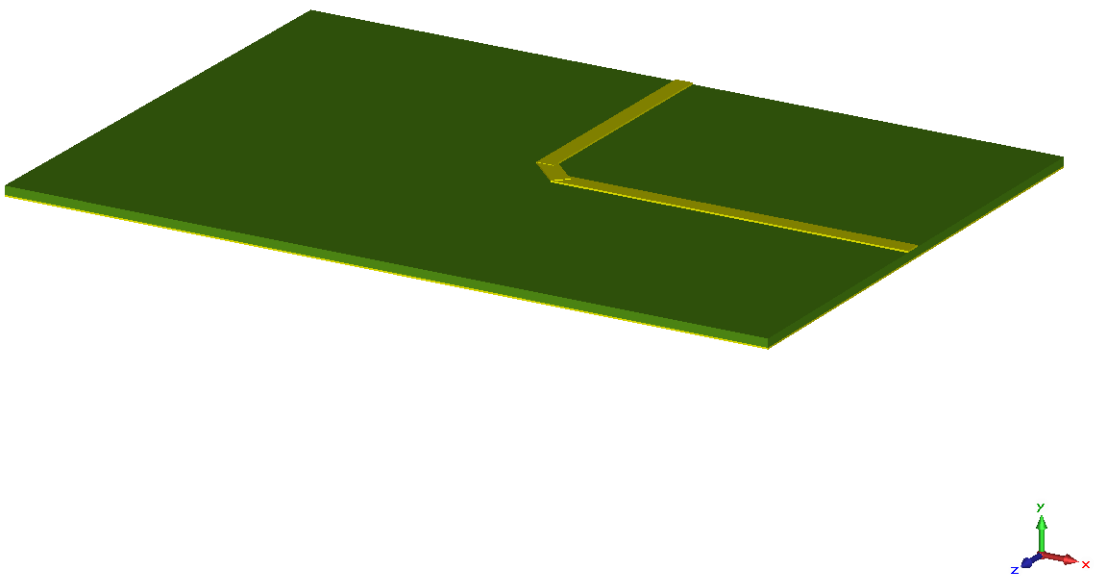

Figure 62. Turns model

13 Disclaimer

This application note is not an all-encompassing training document that can be used by beginner designers to produce reliable PCB designs using modern processors.

Design engineers should use all available design guidelines provided by the manufacturers of other high-speed components used in the system. If a conflict between this document and the guidelines from other manufacturers appears, contact NXP for support.

14 References

[1] GARG, Ramesh, I. J BAHL a Maurizio BOZZI. Microstrip lines and slotlines. 3rd ed. Boston: Artech House, [2013]. ISBN 9781608075355.

How To Reach Us

Home Page:

www.nxp.com

Web Support:

www.nxp.com/support

Information in this document is provided solely to enable system and software implementers to use NXP products. There are no express or implied copyright licenses granted hereunder to design or fabricate any integrated circuits based on the information in this document. NXP reserves the right to make changes without further notice to any products herein.

NXP makes no warranty, representation, or guarantee regarding the suitability of its products for any particular purpose, nor does NXP assume any liability arising out of the application or use of any product or circuit, and specifically disclaims any and all liability, including without limitation consequential or incidental damages. “Typical” parameters that may be provided in NXP data sheets and/or specifications can and do vary in different applications, and actual performance may vary over time. All operating parameters, including “typicals,” must be validated for each customer application by customer's technical experts. NXP does not convey any license under its patent rights nor the rights of others. NXP sells products pursuant to standard terms and conditions of sale, which can be found at the following address: www.nxp.com/SalesTermsandConditions.

While NXP has implemented advanced security features, all products may be subject to unidentified vulnerabilities. Customers are responsible for the design and operation of their applications and products to reduce the effect of these vulnerabilities on customer’s applications and products, and NXP accepts no liability for any vulnerability that is discovered. Customers should implement appropriate design and operating safeguards to minimize the risks associated with their applications and products.NXP, the NXP logo, NXP SECURE CONNECTIONS FOR A SMARTER WORLD, COOLFLUX, EMBRACE, GREENCHIP, HITAG, I2C BUS, ICODE, JCOP, LIFE VIBES, MIFARE, MIFARE CLASSIC, MIFARE DESFire, MIFARE PLUS, MIFARE FLEX, MANTIS, MIFARE ULTRALIGHT, MIFARE4MOBILE, MIGLO, NTAG, ROADLINK, SMARTLX, SMARTMX, STARPLUG, TOPFET, TRENCHMOS, UCODE, Freescale, the Freescale logo, AltiVec, C - 5, CodeTEST, CodeWarrior, ColdFire, ColdFire+, C - Ware, the Energy Efficient Solutions logo, Kinetis, Layerscape, MagniV, mobileGT, PEG, PowerQUICC, Processor Expert, QorIQ, QorIQ Qonverge, Ready Play, SafeAssure, the SafeAssure logo, StarCore, Symphony, VortiQa, Vybrid, Airfast, BeeKit, BeeStack, CoreNet, Flexis, MXC, Platform in a Package, QUICC Engine, SMARTMOS, Tower, TurboLink, and UMEMS are trademarks of NXP B.V. All other product or service names are the property of their respective owners. AMBA, Arm, Arm7, Arm7TDMI, Arm9, Arm11, Artisan, big.LITTLE, Cordio, CoreLink, CoreSight, Cortex, DesignStart, DynamIQ, Jazelle, Keil, Mali, Mbed, Mbed Enabled, NEON, POP, RealView, SecurCore, Socrates, Thumb, TrustZone, ULINK, ULINK2, ULINK-ME, ULINK-PLUS, ULINKpro, µVision, Versatile are trademarks or registered trademarks of Arm Limited (or its subsidiaries) in the US and/or elsewhere. The related technology may be protected by any or all of patents, copyrights, designs and trade secrets. All rights reserved. Oracle and Java are registered trademarks of Oracle and/or its affiliates. The Power Architecture and Power.org word marks and the Power and Power.org logos and related marks are trademarks and service marks licensed by Power.org.

© NXP B.V. 2018. **All rights reserved.**

For more information, please visit: http://www.nxp.com
For sales office addresses, please send an email to: salesaddresses@nxp.com

Date of release: 12/2018
Document identifier: AN12298

arm