Thermal Characteristics of PCB Laminates used in High Frequency Applications

John Coonrod
Rogers Corporation, Advanced Circuit Materials Division

As technology advances, understanding thermal management issues of high frequency PCB's increases. There are many different aspects to consider for PCB thermal management. This paper will investigate thermal management issues of high frequency PCB's as it relates to material properties, insertion loss and circuit design configurations. It will be shown that there are many tradeoffs between these aspects which can be very beneficial for the circuit designer to be aware of.

Material properties: In a very basic sense, the desire in PCB thermal management is to improve the heat flow from the heat source to the heat sink. With respect to material properties, it is obvious the thermal conductivity property of the PCB laminate is important for thermal management issues. However, when considering the simple heat flow equation, there are other aspects to consider as well. Heat flow, in a system at thermal equilibrium, can be defined as:

$$H = k \cdot A \cdot \frac{(T_H - T_C)}{L}$$

Where H is heat flow, k is thermal conductivity, A is the area between the interfaces of different temperatures, T_H is the temperature of the heat source, T_C is temperature of the heat sink and L is the distance between the heat source and heat sink. This basic equation states that an increase in thermal conductivity, will increase the heat flow which is what is desired in thermal management. Another point is decreasing the distance L, between the heat source and heat sink, will improve the heat flow as well. In the case of a simple double sided PCB the distance L is the thickness of the laminate. Here is one of the first potential tradeoffs in thermal management where the use of a laminate with a low thermal conductivity (or poor thermal conductivity) can be somewhat offset by making the laminate thinner. Of course the ideal material situation would be to have a thin laminate with high thermal conductivity. In some cases this is feasible and many cases it is not, due to other considerations.

The property of thermal conductivity for most metals is relatively high as compared to organic substrates which are most often used in the PCB industry. Copper has a thermal conductivity value of about 400 W/m/K and a typical FR-4 laminate will be about 0.2 W/m/K. When comparing the contrast of these material properties, copper behaves like a thermal conductor and the laminate acts like a thermal insulator. A list of thermal conductivity values for materials commonly used in the PCB industry is shown in table 1.

Standard FR-4	0.20
High Performance FR-4	0.24
Woven glass PTFE	0.20
Ceramic filled PTFE	0.50
Ceramic filled Hydrocarbon	0.62

Table 1. List of some common PCB laminates and their typical thermal conductivity values.

The last three materials on the list are generally used in high frequency PCB applications. There have been circuit applications which were not high frequency, but still used one of these laminates because of the improved thermal conductivity. Even though the values appear relatively insignificant in comparison to copper thermal conductivity, a change in thermal conductivity from 0.20 to 0.62 is an increase of heat flow by triple and that is significant.

Insertion loss: This can be a very complicated topic and the basics will be discussed to help illustrate the importance of understanding circuit loss in regards to thermal management issues. In an effort to keep the examples of circuit thermal conductivity simple, a microstrip transmission line circuit will be the model discussed. A microstrip circuit is a simple double copper layer circuit which has the signal plane on one side and a ground plane on the other. In this case, a microstrip transmission line with a heat sink bonded to it is shown in figure 1.

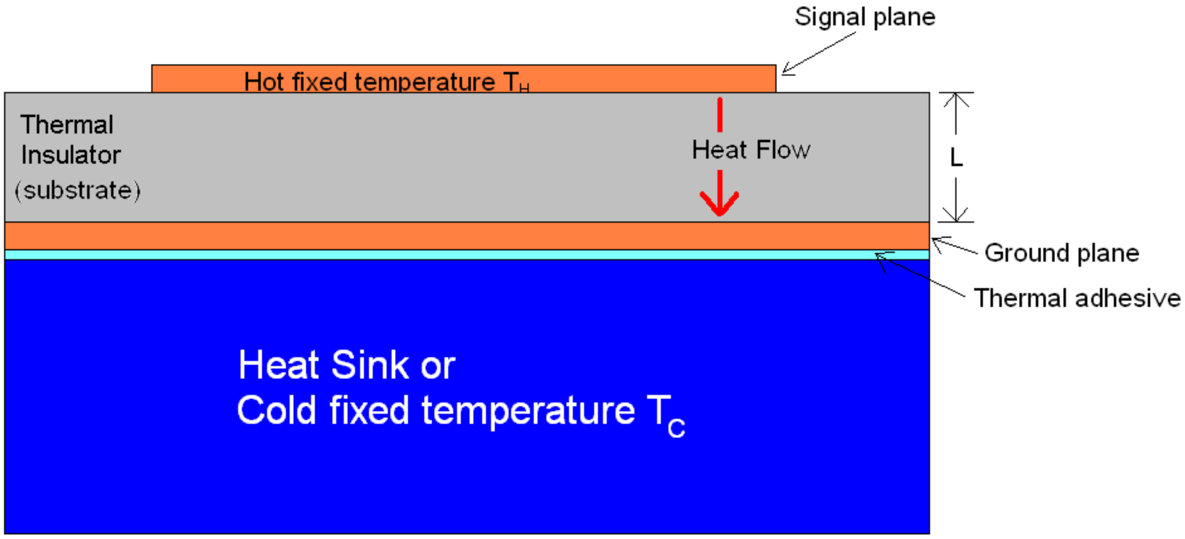

Figure 1. The circuit model used to illustrate PCB thermal management issues. This is a microstrip transmission line with a heat sink bonded to the bottom or ground plane side.

In the model shown in figure 1, it will be assumed the temperature of the ground plane of the circuit is at the same temperature of the heat sink. Also, it will be assumed the circuit is in a state of thermal equilibrium.

Some distinction should be given between how the heat is generated in a circuit. For this paper there will be two avenues of heat generation assumed; 1. Heat can be generated by a hot device which is attached to the circuit and 2. Heat is generated by RF power applied to the conductors of the circuit. When the heat is generated by the power applied to the circuit conductors, it will be referred to as trace heating.

The cause and effect of trace heating is actually more than what the term may imply and the heat is generated by the insertion loss of the circuit.

In general, when a circuit has more loss, there will be more heat generated. As it was previously stated, the desire of thermal management is to improve the heat flow from the heat source to the heat sink, however, if less heat is generated this will be an improvement in thermal management as well.

Insertion loss is the overall loss of the transmission line circuit. There are four components for microstrip insertion loss: conductor, dielectric, radiation and leakage. Leakage loss is typically an issue with materials used in the semiconductor industry. It is not affiliated with high frequency PCB applications and accordingly leakage losses will not be considered in this paper. Radiation losses are a function of frequency, circuit thickness, dielectric constant of the material and circuit design. For the following examples it will be assumed that radiation losses have been adequately accounted for in the circuit design and are not a contributing factor to thermal management. The last two losses, dielectric and conductor, will be the focus of discussions on insertion loss related to thermal management.

Dielectric losses are associated with the dissipation factor property of the PCB material. The values of dissipation factor or tangent delta, for PCB materials can range greatly and typical values are shown in table 2 for some common materials used in the PCB industry.

High performance FR-4	0.020
Woven glass PTFE	0.001
Ceramic filled PTFE	0.002
Ceramic filled Hydrocarbon	0.003

Table 2. Typical dissipation factor values for common materials used in the PCB industry.

The last three materials in table 2 are commonly used in high frequency PCB applications.

Another tradeoff to consider is dissipation factor and thermal conductivity. A material with very poor thermal conductivity, such as the woven glass PTFE, may have less thermal management issues due to the low dissipation factor enabling less heat to be generated. Ideally, an application with critical thermal management issues should use a material that is very high in thermal conductivity and very low for dissipation factor.

Conductor losses for a microstrip transmission line circuit have more concerns to consider. Conductor losses can be affected by frequency, thickness of the laminate, dielectric constant of the laminate, plated finish of the conductor and copper surface roughness.

The effect of the thickness of the laminate is due to the ratio of losses of dielectric and conductor, where a thinner laminate will have the conductor losses dominate. The dielectric constant as a property will not affect conductor loss. However, for matched impedance circuits, if a material with a lower dielectric constant is used, the conductor width will need to increase in order to maintain the same characteristic impedance. An increase in conductor width will lower the conductor losses.

Most of the electric field interaction is between the bottom side of the signal conductor and the top side of the ground plane, when viewed in a cross-section view as shown in figure 2.

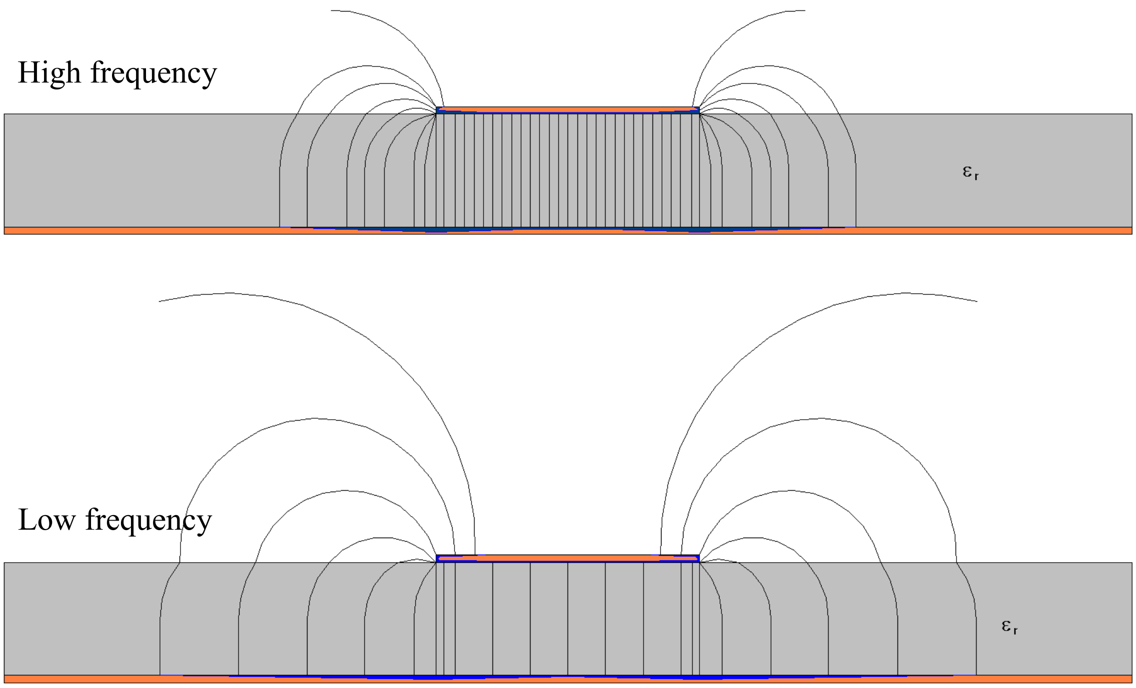

Figure 2. Approximate representation of electric field lines and current density of a microstrip circuit at high frequency (top) and low frequency (bottom).

At high frequencies, the electric fields are more condensed and there is less area of the ground plane being used for the ground return path. At both low and high frequencies, there is a concentration of current at the bottom corners of the signal conductor and this is shown as a blue fill area on the orange copper features in figure 2. The area of the conductor with current density (blue fill areas in the copper) will be less at a higher frequency. This is due to skin effects. Basically, as frequency increases, less of the bulk conductor is used and it is the “skin” of the conductor where much of the current density resides.

The copper surface at the copper-substrate interface is typically rougher than the air side of the copper for a microstrip. This is due to the microstrip using a copper clad laminate and the laminate manufacturers prefer a rougher copper for better bond of the substrate. When the current density depth in the copper (skin depth) is near the thickness of the copper which corresponds to the surface roughness, the roughness becomes more significant for increasing conductor losses. A simple explanation of the increased losses is that rougher copper will have more surface area which will increase the propagation path and a longer path will have more loss. In reality there is much more to this story and a very good paper which explains this in detail [1] can be referenced.

Another factor to consider for conductor loss is the plated finish which is applied to circuits. Some of these finishes cause more conductor losses than others and are also frequency dependent. The popular enig (electroless nickel, immersion gold) finish will generally have more conductor losses for a microstrip transmission line. The reason the losses are higher at certain frequencies, is the skin depth will mostly reside in the nickel layer which is significantly less conductive than copper. Some designers will consider a silver finish which has very similar conductivity as copper and therefore not cause a negative impact on conductor losses. A simple study [2] regarding the impact of different plated finish on microstrip circuits was performed and summary data shown in table 3.

Finish	***Average IL (dB/in) @ 22GHz***
Bare copper	0.444
Silver	0.453
HASL	0.493
ENIG	0.560

Table 3. Comparison of insertion loss using different finish on microstrip transmission line circuits.

The circuits used in the study for table 3 were manufactured using an 8 mil RO4003C™ laminate. The Microstrip transmission lines had a 17 mil conductor width.

A possible tradeoff, which is sometimes considered, is the use a copper clad laminate with a smooth surface to get lower conductor losses. In general, when the laminate has a smoother copper, the bond strength values will be lessened. Additionally, the use of a silver finish for the circuit and use of a copper clad laminate with a smooth surface would be ideal when minimizing conductor losses. To restate, when there is less loss in the circuit, there will be less heat generated.

<u>Circuit design configurations</u>: there are multiple issues to consider in PCB design regarding thermal management. In keeping with the format of this paper, the following information will be referencing the microstrip circuit discussed thus far.

A study regarding thermal conductivity of PCB laminates [3] was done using the same microstrip circuit design with the only variable being the type of substrate used. The substrate type varied in thermal conductivity and other properties. However, the study was focused on the thermal conductivity properties only. A chart which summarizes some of the results is shown in figure 3.

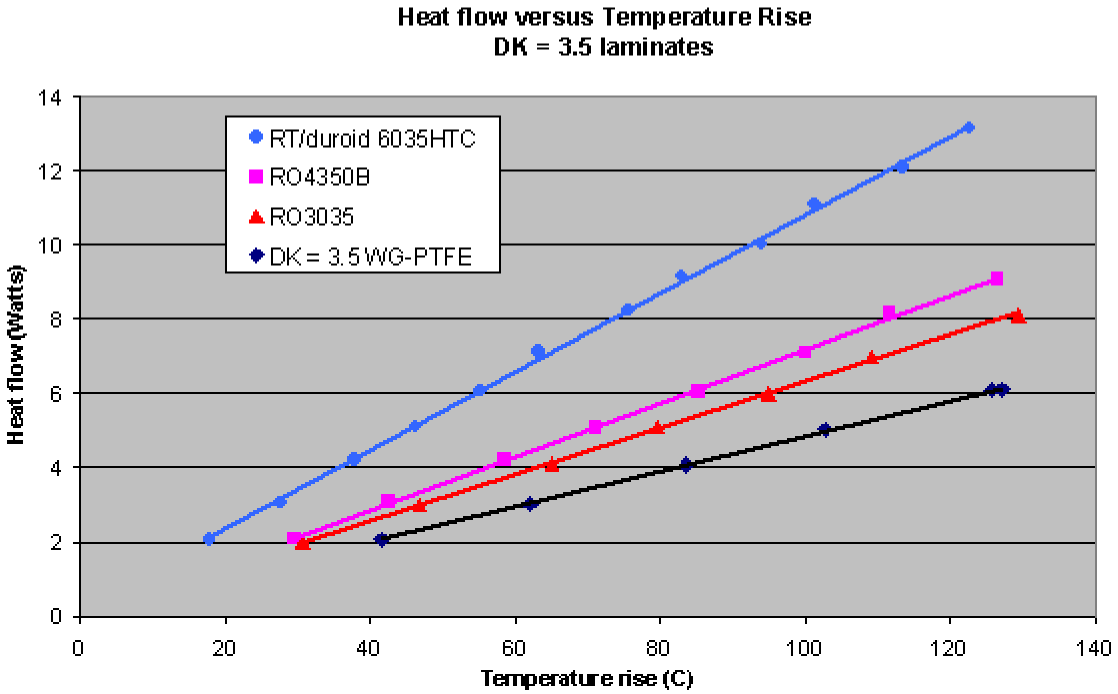

Figure 3. Heat flow vs. Temperature rise of several circuits using the same design, but different high frequency laminates.

Table 4 shows the thermal conductivity properties of the materials used in the thermal conductivity study.

Material	Thermal Conductivity (W/m/K)
3.5 WG-PTFE	0.25
RO3035™	0.50
RO4350B™	0.62
RT/duroid® 6035HTC	1.44

Table 4. Listing of thermal conductivity values for the materials used in the thermal conductivity study.

RT/duroid 6035HTC was specifically formulated for applications where thermal management was critical. Besides having the highest thermal conductivity, it also has a very low dissipation factor (0.0013) and is available with relatively smooth copper. This material has all aspects which are optimum for good thermal management designs.

Over the years many PCB designers have employed via farms to assist in effectively moving the heat out of a hot device on the signal plane to the heat sink. As part of the previously mentioned study, the same circuits with the different materials were also used with the design having via farms. The results of this portion of the study are shown in figure 4.

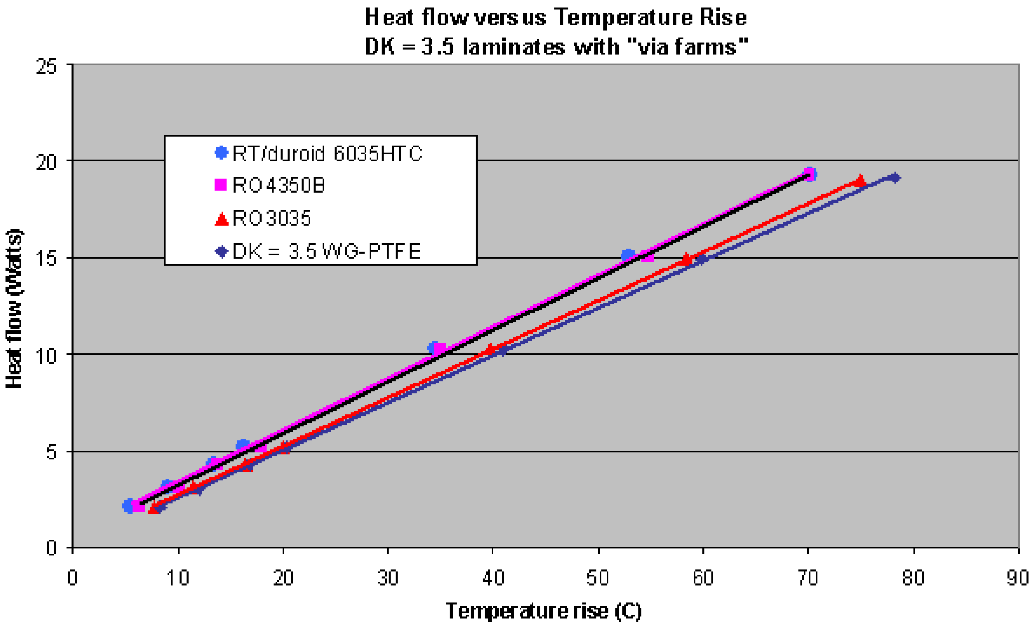

Figure 4. Heat flow vs. Temperature rise with via farms.

The same trend, as noted in figure 3, is found in figure 4, however with much less spread in the curves. Even though the scale on the x-axis of the charts in figure 3 and 4 are different, when using 80°C as the reference on the two charts, it is obvious the heat flow for the circuits with the via farms is much greater than the circuits without the via farms.

The major issue with via farms is that they cannot be employed on an active signal conductor, where the via's would short the signal to ground. Via farms are most often used with heat generating devices which have a large ground pad under them, where heat can be removed more effectively with the via farms.

For trace heating, a quick summary of what is ideal for thermal management would be using a laminate that: is thin, has high thermal conductivity, has a lower dielectric constant, has smooth copper and a low dissipation factor.

There is another design consideration which will assist in thermal management issues for trace heating, however the RF design characteristics will be different and the designer would need to account for this difference. This design is an alteration of the microstrip where it has ground planes on the signal layer and in close proximity to the signal conductor with the trace heating concern. The ground planes on the signal layer will need to have vias to the ground plane below. These vias will need to be spaced on a periodic pitch for good RF performance and to act as a thermal fence. In regards to RF performance, the pitch should be 1/10th the wavelength or less. A denser via pattern will be good for the RF as well as the thermal fence performance.

The thermal fence, when in close proximity to the signal conductor, will receive heat by the surface plane conduction and transfer it to the heat sink by using the via's in the same manner as the via farm. In order to implement this well, the material should have high thermal conductivity to move the heat from the signal conductor to the via fence in the adjacent ground planes. The circuit configuration is called a grounded coplanar waveguide and the impedance and other electrical properties will be different in this coplanar area of the circuit. A simplified drawing of a grounded coplanar is shown in figure 5.

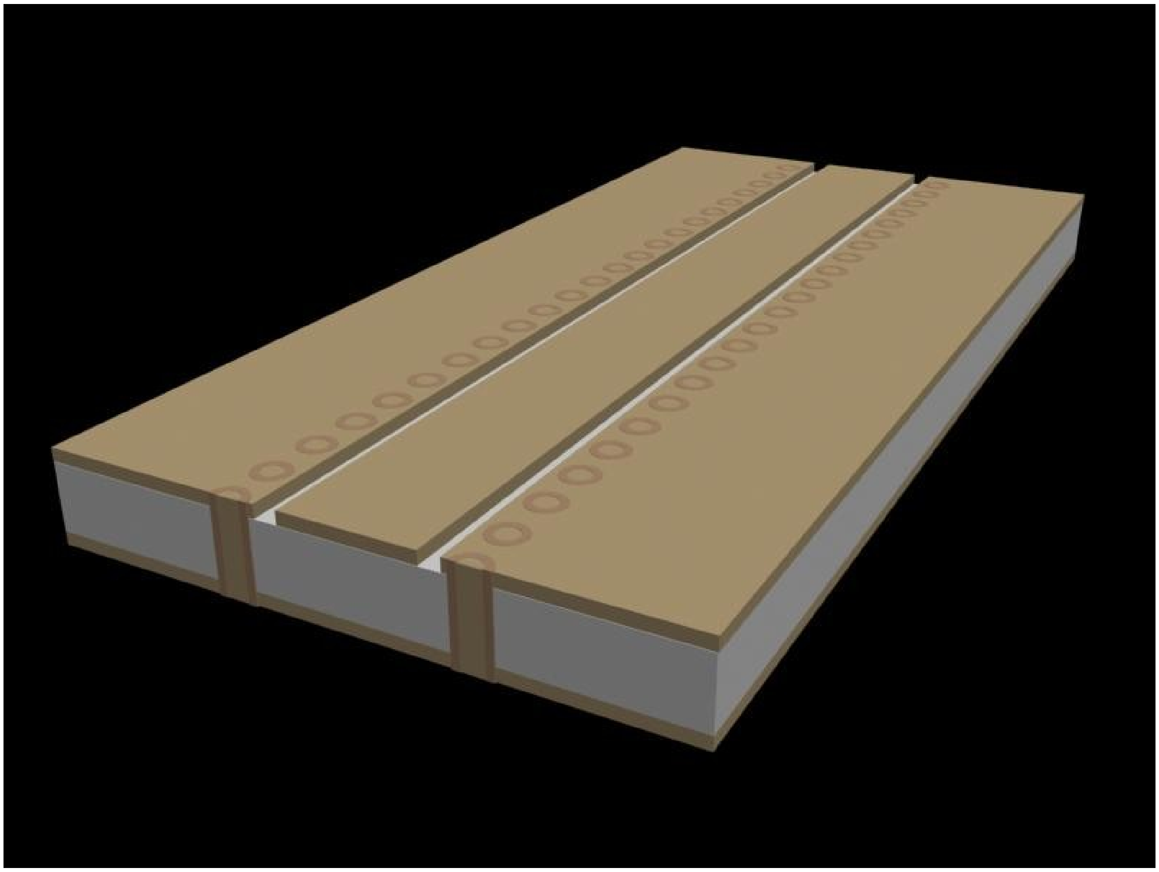

Figure 5. Simple drawing of a grounded coplanar transmission line.

The coplanar example shown in figure 5 can be a portion of a microstrip transmission line, where a coplanar portion is employed to better the thermal management for trace heating in critical areas. The grounded coplanar area will behave different for electrical performance where the impedance and losses will change unless the geometry is considered. The grounded coplanar will have more conductor losses due to the nature of the circuit geometry, however there is more flexibility for achieving an impedance target. The grounded coplanar impedance can vary by a ratio of the conductor width and the spacing to the coplanar ground edges, making it possible to increase the conductor width and still maintain the controlled impedance value. The increase in conductor width will lessen the conductor losses. To have efficient heat flow, the ground spacing on the signal plane should be very close to the signal conductor. Due to this and the desire to have a wide conductor for less losses, this is a typical tradeoff to be considered.

In summary, it has been shown that there are several PCB material properties that are critical to thermal management. There are also several tradeoffs between different performance issues and thermal management issues. The optimum material consideration for thermal management would be to use a laminate with a high thermal conductivity, low dissipation factor, smooth copper surface and a lower dielectric constant. There are several design considerations as well and if the issue is trace heating, the optimum material considerations are applicable along with the option for a grounded coplanar structure. If the thermal management issue is a heat generating device on the circuit, the optimum material suggestions are still valid, however the use of via farms would be highly recommended.

[1] J.W. Reynolds, P.A. LaFrance, J.C. Rautio & A.F. Horn III, “Effect of conductor profile on the insertion loss, propagation constant, and dispersion in thin high frequency transmission lines”, DesignCon 2010.

[2] A.F. Horn III, “Increased circuit loss due to Ni/Au”, Internal Rogers Corporation technical study, Jan. 2006.

[3] John Coonrod, Allen F. Horn III, “High Frequency Circuit Materials With Increased Thermal Conductivity”, High Frequency Electronics, Nov. 2010.