

# **Section 7 Extraneous Items**

**from**

**A Collection of Thoughts, Tips and  
Techniques for Microwave Circuit Design**

**by**

**Dr. Robert L. Eisenhart  
November 2020**



eisenhart & associates  
[r.l.eisenhart@ieee.org](mailto:r.l.eisenhart@ieee.org)  
[www.DR-HFSS.com](http://www.DR-HFSS.com)

This set of 45 pages is a section focusing on a wide range of topics, taken from an extended presentation on microwave design.

If you have a question, feel free to write me at [R.L.Eisenhart@ieee.org](mailto:R.L.Eisenhart@ieee.org).

# **Extraneous Items Outline**

- Using Optimizers
- Co-Simulation Examples
- De-embedding
- Impedance Estimation
- Large Array Waveguide  
Simulation Technique
- Ku-band Orthomode Transducer

We will start out looking at Optimizer operation

# **Optimizer Operation**

- **How do you use an optimizer?**
  - You build a reference model with nominal values, defining the parameter values with names
  - Define the parameter step size and operating ranges
  - Set the optimization cost function
- **What happens next?**
  - The optimizer manipulates the parameters, runs a simulation for each set, compares performance against the criteria, and keeps track of the parameter values
  - When the performance can no longer be improved, it stops with the selected values for the best, or “optimized” parameter set

Robert Eisenhart

3

Sometimes setting up the model can be tricky, particularly when parameter values are coupled.

The value of optimization is that you can be doing something else while the computer is running.

Key practices

## **Optimizer Practices**

- **Take advantage of any model symmetries**
- **Parameter selection – minimize # and make sure the model reacts as desired**
- **Have some knowledge of your circuit performance, and reasonable parameter values**
- **Start optimization with as good as an estimate of the expected solution as possible**
- **Avoid periodic conditions**

Before you have the optimizer start, view the model with each of the parameter changes, both higher and lower values to make sure the model will work.

Some features

## **Optimizer Features**

- **Provides iteration capability with many model parameters**
- **Allows visual inspection of modified models**
- **Allows a STOP criteria based upon defined performance**
- **Takes the operator out of the loop**

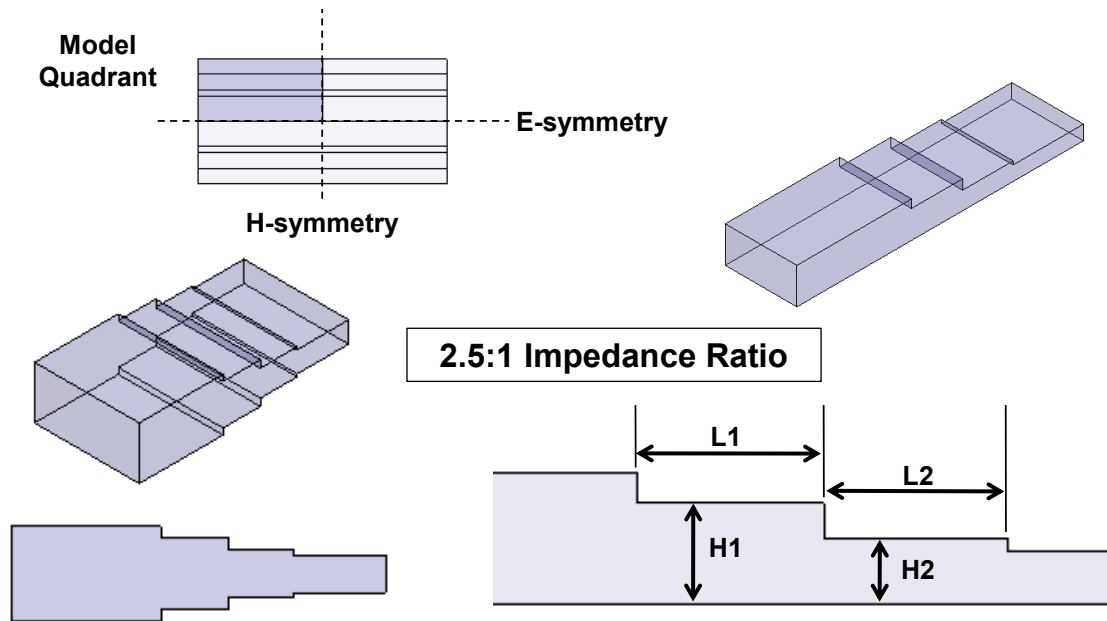
### **However -**

**Each configuration of the model parameters requires a full 3-D meshing and analysis which can be very time consuming for multiple runs.**

Don't get carried away with too many parameters and very small step sizes.

Consider a simple impedance transformer in WG

# 3-Step WG Impedance Transformer Design



**Using two symmetry planes reduces model size, particularly important for optimization with 4 parameters**

Robert Eisenhart

6

Here we have a 2-section impedance transformer

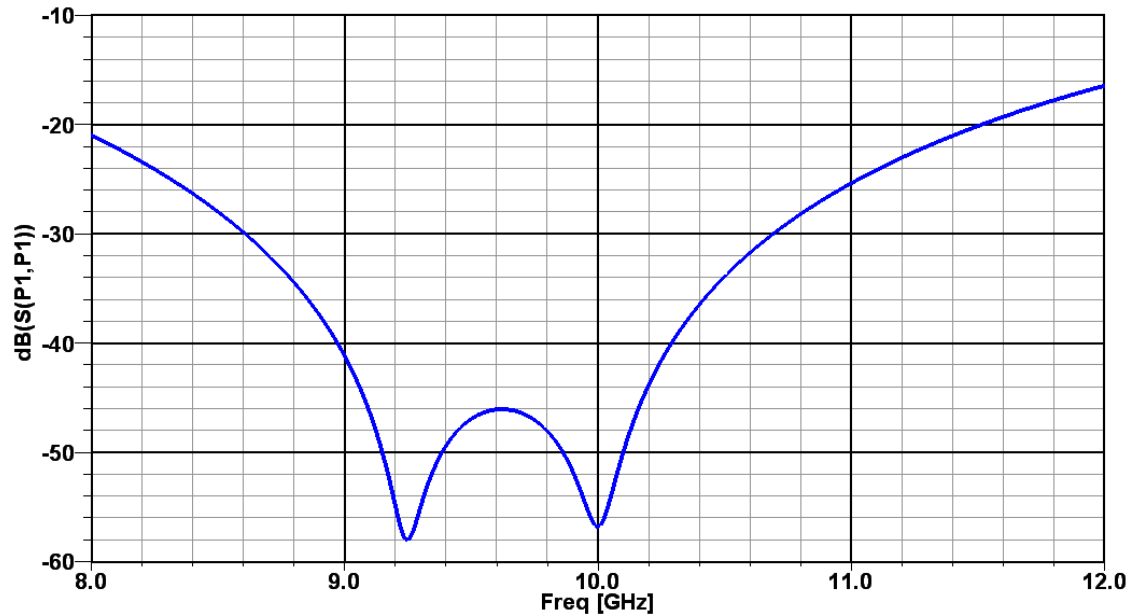
Use symmetrical step if possible for better performance. Reduces fringing capacitance.

Use smallest part of the model which accurately represents the behavior. Here we use just one quadrant.

Four parameters are two heights and two lengths. The fringing capacitances effect the line lengths.

So we'll optimize at center band

## Optimizer Result at Single Frequency?



**Optimizer got caught in a local minima – some experience needed in setting goals**

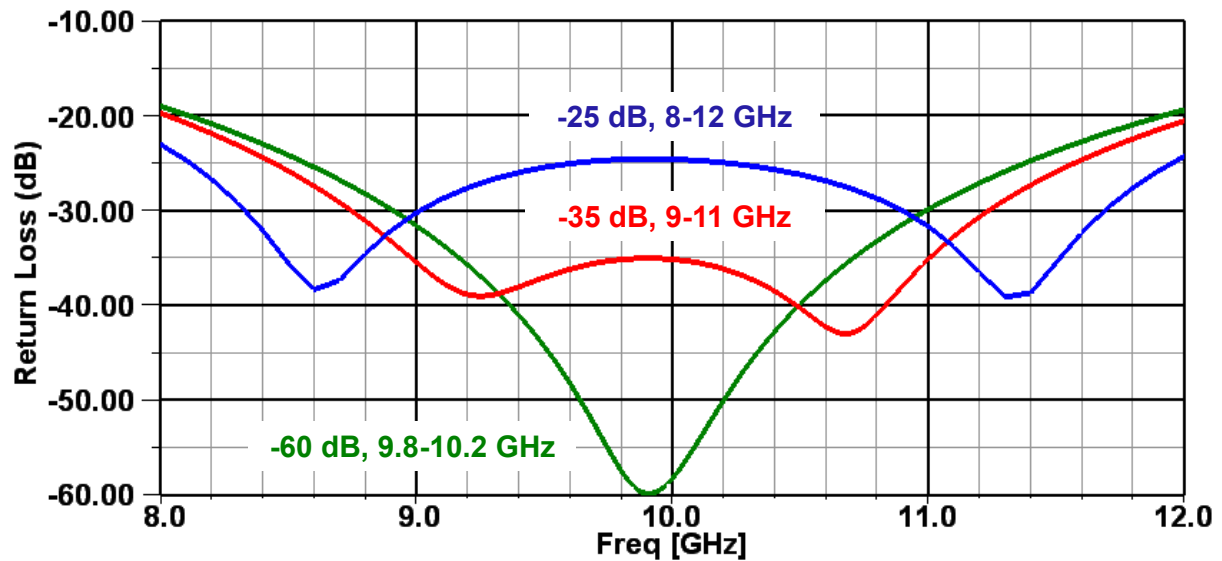
Robert Eisenhart

7

Surprise!

Try again

# Various Optimizations Over the Band



**You have to be fairly specific with the goals**

Robert Eisenhart

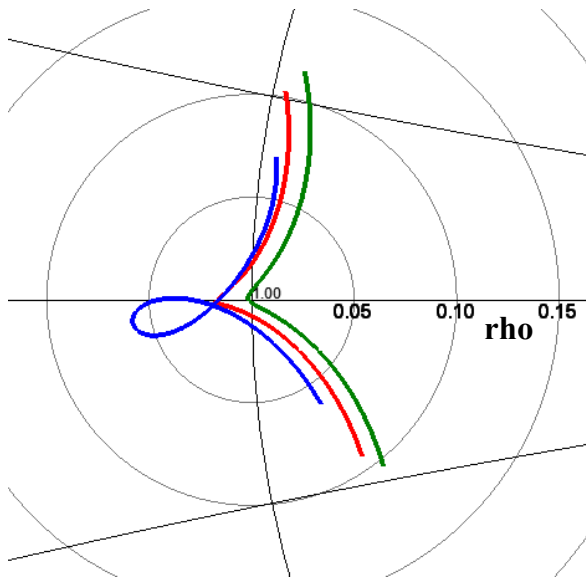
8

The cost goals for these curves use band edges as well as the center frequency with the goals to keep the performance symmetrical. Different results for different goals.

Smith Chart View of results



## Smith Chart plots for the three designs



-25 dB, 8-12 GHz

-35 dB, 9-11 GHz

-60 dB, 9.8-10.2 GHz

**Completing the picture of the match**

Robert Eisenhart

9

The curve of page 7 tried to optimize on a single frequency.

Works better using a small band about the desired center. (green curve)

Relax the match and tune for a wider bandwidth. This often requires repeated trys. (red curve)

Extending to a wider band. (blue curve)

Of course the Smith Chart gives another perspective.

Then summarizing

## Optimizing Summary

The optimizer is a significant part of the EM field/circuit simulator package. It allows the operator to stay in control of the design process, but be less involved in the repetitious, time consuming steps of trial and error design. It is a complement to the parameter sweep capability

**Takes HFSS from the Analysis domain into the Design domain**

Sometimes the best design is to use a hybrid approach. In this case we call it “Co-Simulation”.

# Introduce Co-Simulation

**HFSS (EM) ↔ Designer (Circuit)**

- Simply the use of multiple simulation programs to address a design problem, avoiding the limitations of one program and taking advantage of features of another.
- The common language between programs is the use of S-parameters, defining circuits blocks with properly defined Port characteristics.

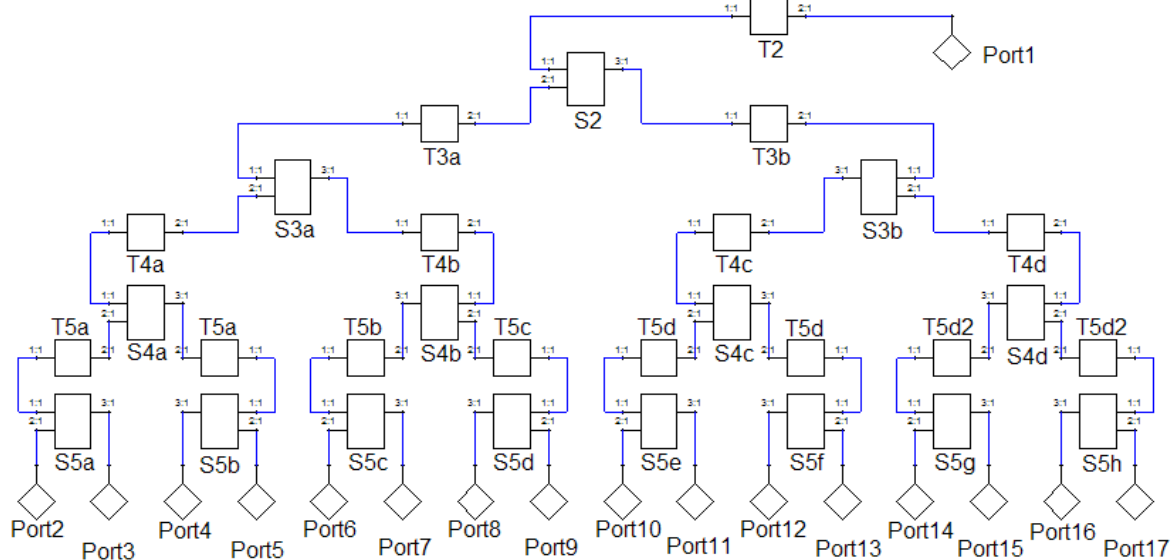
What is Co-simulation?

Having two different simulators working together, each doing what they do best, to come up with a solution. Naturally the programs have to communicate with circuit description and data.

The first example will address the issue of model size limitations in HFSS.

# Co-Simulation Example #1

Here are 30 different components connected together which were modeled independently in HFSS to represent the antenna feed circuit for 16-way power division



17 ports, 15 unequal splitters and 15 transformers (connected by 11 different transmission lines) are accurately modeled in Designer

Robert Eisenhart

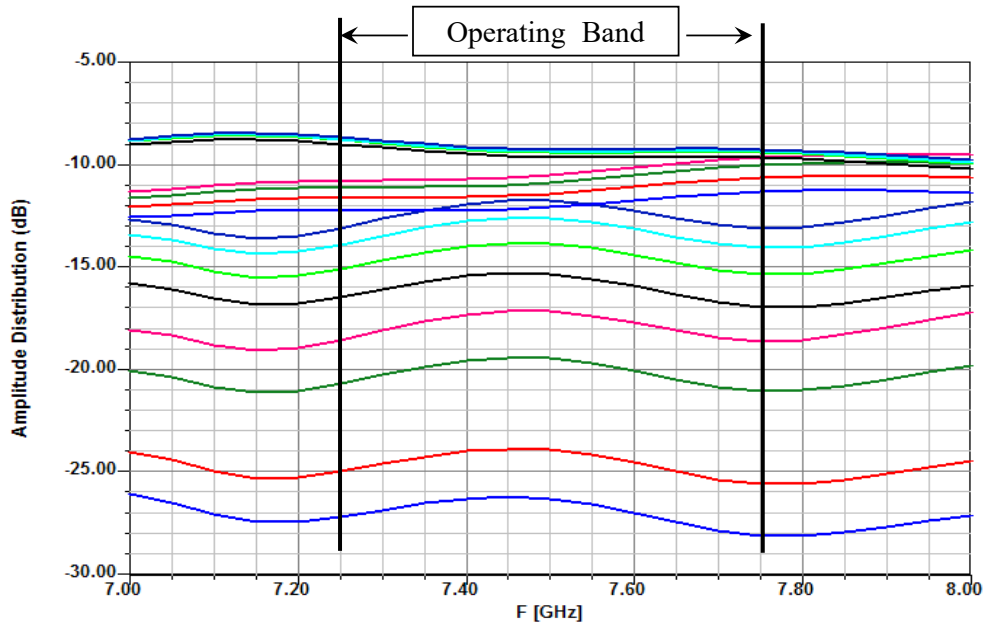
12

This example shows how we can build a large circuit by consolidating the many parts which have all been accurately characterized in the 3-D simulator, where this total circuit size is much too large for the 3-D simulator to handle as a single model.

This circuit distributes power from Port 1 to 16 ports (2-17) in a prescribed non-uniform excitation with over 20 dB variation across the 16 output ports.

So that we can now show the overall performance.

## Output Excitation for a line array (16 elements) from the input Port



The 15 splitters and 15 transformers are connected as S-parameter defined blocks to simulate the array feed with 17 ports.

Robert Eisenhart

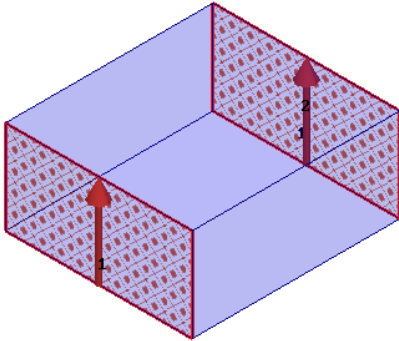
13

We're looking for a -20 dB variation from top to bottom, the result averages about -18 dB.

The second Co-Simulation example will take advantage of circuit simulation analysis speed over the relatively slow 3-D simulator speed.

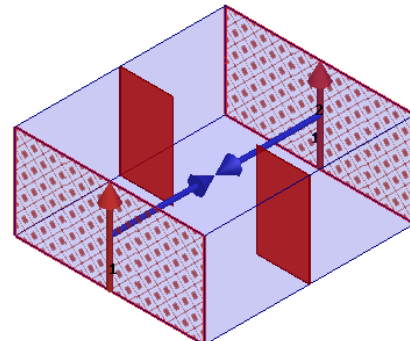
## Co-Simulation Example #2

Here the HFSS components were simulated for a number of parameter values, creating sets of S-parameters for each value



**Waveguide length  
as a variable  
500 - 700, 25 mil steps**

**Port reference  
at each end**



**Inductive element  
opening as a variable  
500 - 700, 25 mil steps**

**Port reference  
set at iris center**

Robert Eisenhart

14

This time we will construct a 3-resonator waveguide band-pass filter. There are two pieces linked from HFSS to Designer.

On the left is a variable length of waveguide

Modeled at nine different lengths with the ports at the ends.

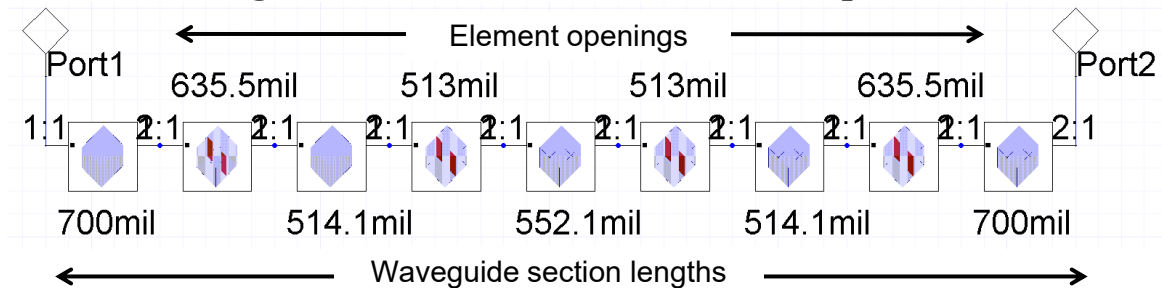
On the right is a section of WG with a shunt inductive element.

The opening in the element is modeled at 9 different sizes, and the port reference planes are de-embedded to the element center. All of the data for both pieces at all variable values is linked from HFSS to Designer.

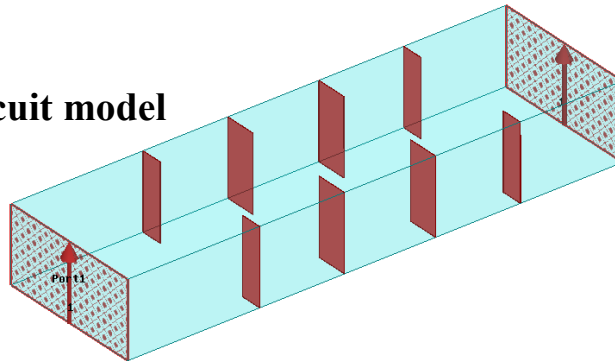
For length values of waveguide and inductive element opening values not simulated (or linked to HFSS), Designer uses interpolation to generate approximate values. Make up a "block diagram" type circuit representing the real 3-D circuit.

# Co-Simulation of a Waveguide Band Pass Filter

## A. Designer circuit model with linked components



## B. HFSS 3-D circuit model



Robert Eisenhart

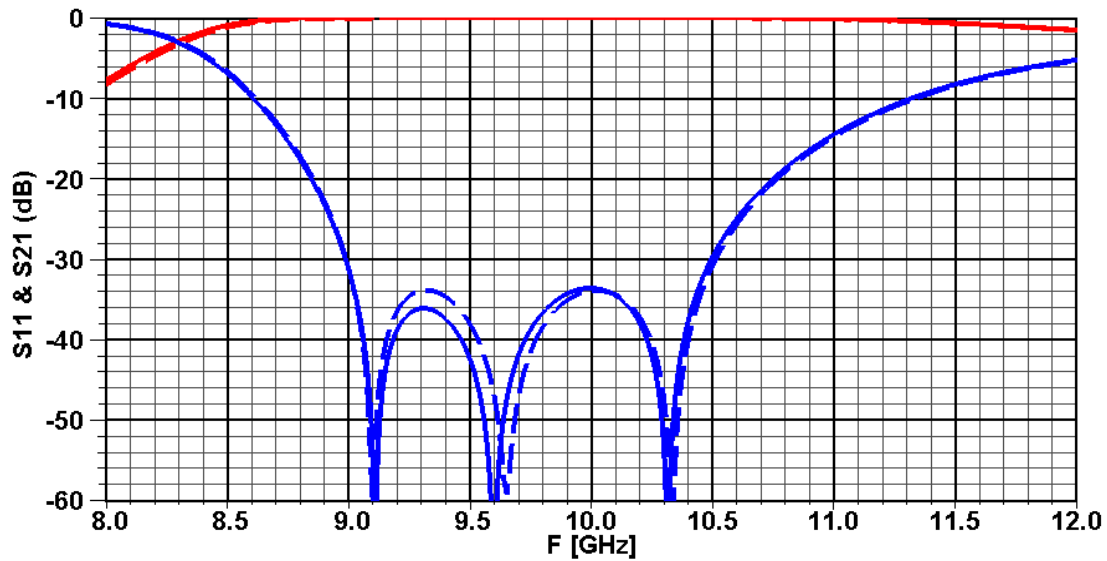
15

Here's the cascaded circuit block diagram representing the real 3-D circuit. Initially the circuit values are selected to approximate the filter ballpark values. Then an optimization is run with the advantage that the circuit simulator can run 100s of simulations in minutes.

The results of the optimization are then used to create a complete model in HFSS.

So let's compare the results for the two programs.

## Co-Simulation Results for a Waveguide Band Pass Filter



HFSS\* —————

Designer\*\* - - - - -

\* HFSS simulation with Designer defined dimension values

\*\* Designer simulation with HFSS defined S-parameter circuit blocks

Robert Eisenhart

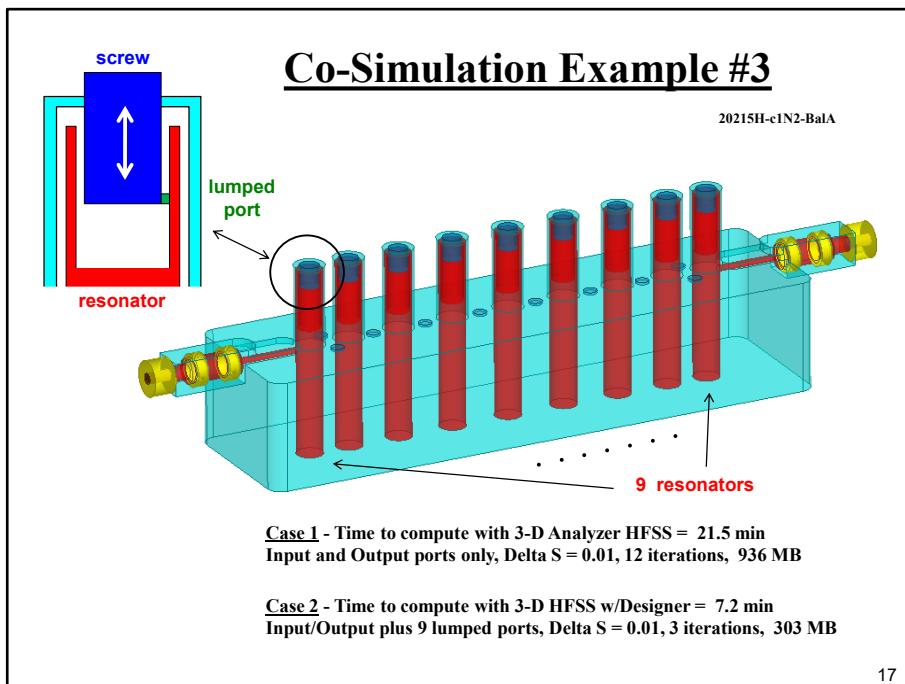
16

There is excellent correlation between the two models.

Before we leave Optimization I don't want to leave the impression that it is necessarily easy. Often selecting the starting values for the optimization run is the most difficult part, along with setting the cost function (previously addressed).

On to example #3.





This third co-simulation example actually has two tricks to it. Both rely on “seeding” a complex circuit model such as this 9 resonator filter with lumped (internal) ports. A lumped port is like a pair of probes which can be put anywhere within a model, which then acts just like a port. This filter is like a small box with coax ports at each end and nine resonators inside. Each resonator is attached at the bottom and open circuited at the top with high capacitance to the top walls. The resonators couple to each other in a precisely controlled manner.

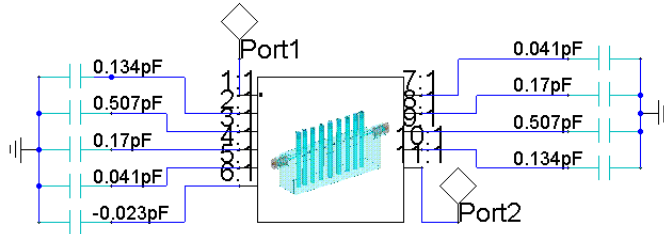
The capacitance at each top can be varied with a tuning screw by changing the length it sticks into the resonator end which has a cutout region. This model has a lumped port at the end of each tuning screw.

Without these extra lumped ports, as case 1, simulating the model in HFSS takes 21.5 minutes and results in a performance curve, shown in page 19.

However, with the resonant killing loading effect of the extra lumped ports as case 2, the simulation takes only 7.2 minutes. It may be 3 times faster but what do we gain?

Move on to the co-simulation part.

## Co-Simulation 11-Port Circuit Model



**Case 1** – Doesn't use Co-simulation

**Case 2** – All capacitor values set = 0 to simulate open circuits, (this effectively removes any influence from adding the lumped ports) essentially producing the same results as Case 1, only faster.

**Case 3** – All capacitor values are tuned with the optimizer in the circuit program, realizing 100s of simulations in just minutes, resulting in the values shown in the schematic above.

**Case 4** – Make the tuning screw length changes and simulate again as in case 1, using the relationship:

$$\Delta \text{ screw length (mil)} = \frac{\text{Tuned value(pf)}}{0.0083 \left( \frac{\text{pf}}{\text{mil}} \right)}$$

18

We haven't gained anything yet because we're not done. The next step is to link the model to Designer and to terminate all ports with capacitors.

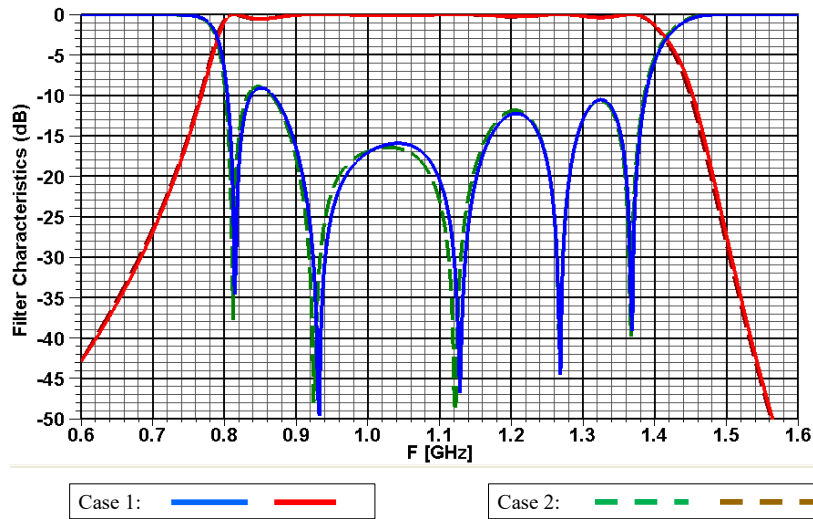
By setting all the 9 lumped port capacitors to very small values, and analyzing the circuit we get the same result as Case 1.

The second trick is the real benefit because now it is possible to run optimization on the complex filter (with respect to tuning screw depths) with the circuit simulator.

With the lumped ports placed at the ends of the screws, the capacitive values from the optimization can be easily converted to added screw length by using the capacitance per length for the impedance of the inserted screw.

Let's see the results from all this.

## Co-Simulation Cases 1 & 2 Comparison

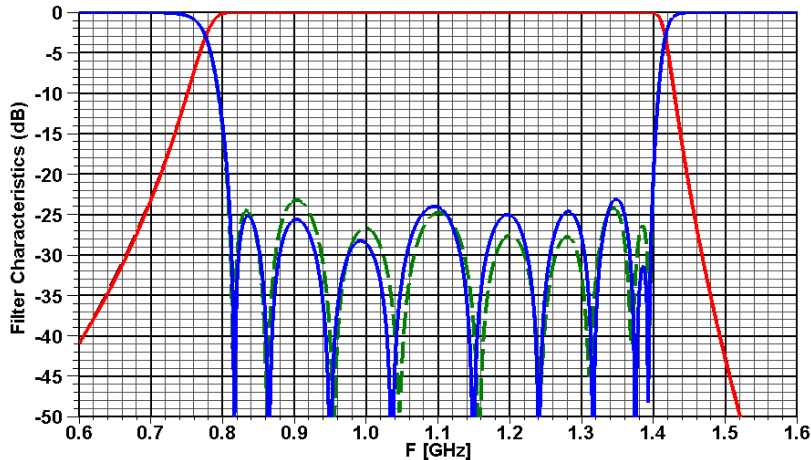


19

Here's the comparison for the two un-tuned cases, showing equality. That is, using the lumped ports approach gives the same answer as direct simulation, only faster.

How about the final design?

### Co-Simulation Cases 3 & 4 Comparison



20

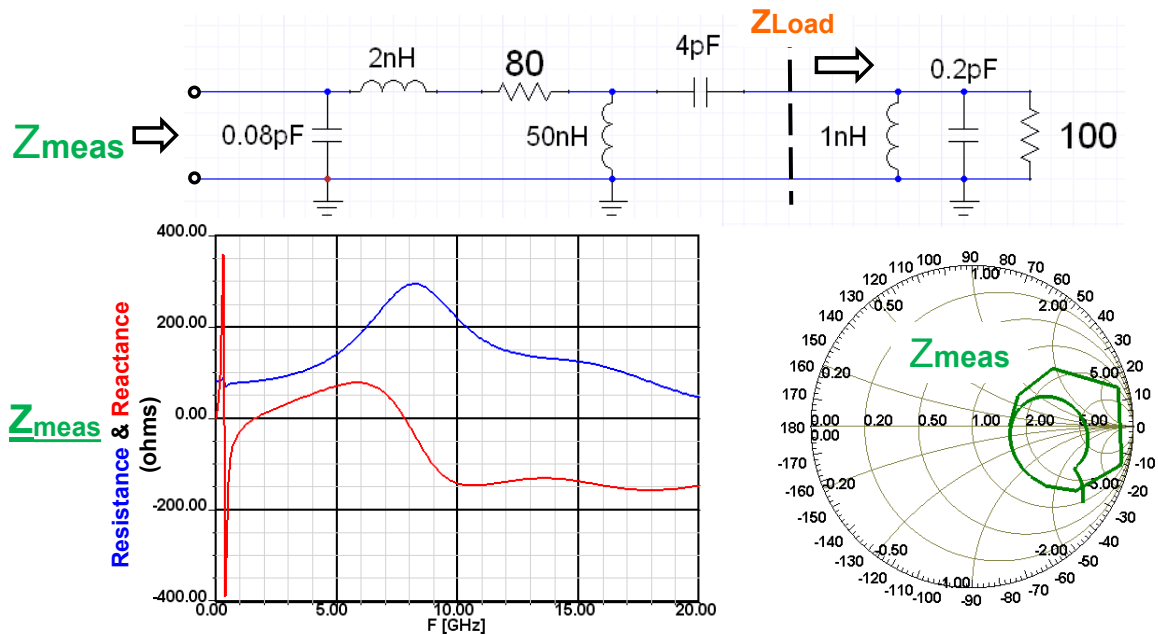
Case 3, performance in Designer with optimized capacitor values on the 9 lumped ports, and

Case 4, performance in HFSS with tuning screws of the nine resonators set to lengths that correspond to the optimized capacitors.

Again there is good comparison, except this time we never would have arrived at the final tuned design if we had to optimize with HFSS alone.

One quick trick to determining a buried impedance.

# De-embedding from a measured or simulated $Z_{meas}$ to an internal “port” $Z_{Load}$



**De-embedding thru lumped elements works also!**

Robert Eisenhart

21

De-embedding is not just for lengths of transmission line.

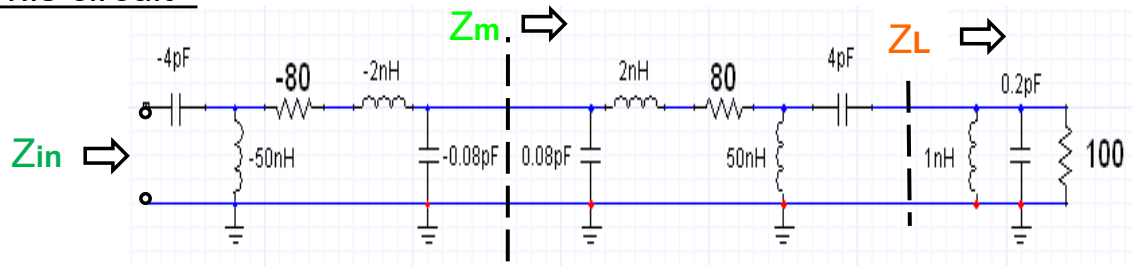
Say we measure  $Z_{meas}$  and want to know  $Z_{Load}$  and we do know the makeup of the circuit we're measuring through.

Here's  $Z$  measured but

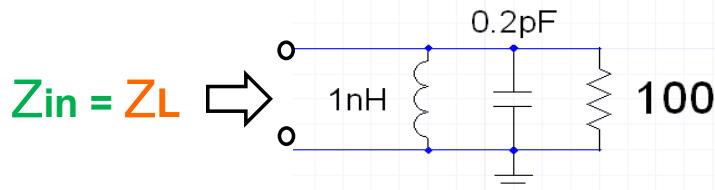
How to de-embed to  $Z_{LOAD}$ .

# De-embedding to the Load, $Z_{Load}$

This circuit -



Equals this circuit -



**Negative image the elements to be removed  
about the measurement plane**

Robert Eisenhart

22

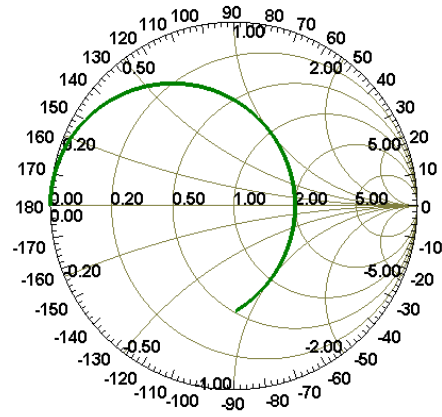
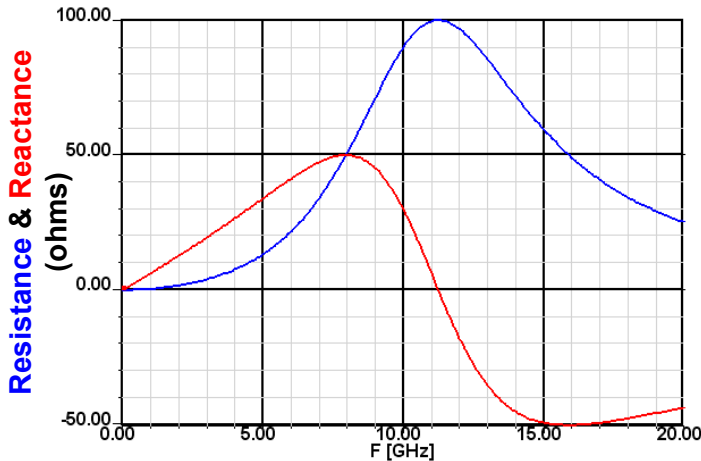
With your circuit simulator just add on the input, the negative mirror image of the through circuit and calculate the input impedance.

and you get the answer.

looking like

# Impedance of $Z_{Load}$

Resulting in -



**De-embedding is used to remove circuit elements that are present but don't allow direct measurement of what you want to see. Calibration is a form of de-embedding by characterizing the elements to be removed.**

Robert Eisenhart

23

The result we want

We've just done a sort of calibration.

Let's next look at the relationship of the Wave Impedance (377 ohms) and a TEM line impedance.

## Impedance Observations of a point in a Radiating Wave

Imagine you are a point in 3-D space as part of a radiating wave. The ratio between the E & H fields is the Wave Impedance = 377 ohms and the fields are normal to each other and perpendicular to the axis of propagation. You don't know whether you are in "free space" or somewhere inside a transmission line (TL) because you can't tell the difference.

However, if you are part of a TL, the integrated effect of all points within the boundary of the TL will result in an effective impedance, the "characteristic impedance" of the transmission line.

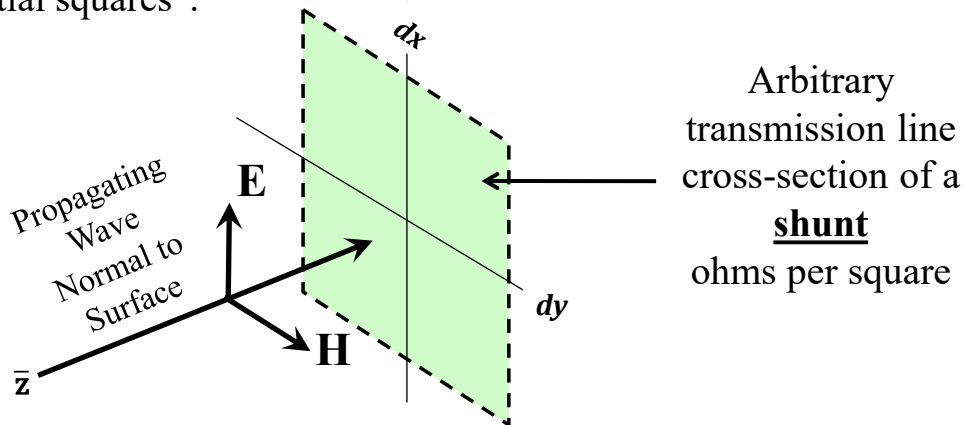
How can we determine a relationship between the wave impedance and the characteristic impedance?

Integrate the differential squares . . .



## Impedance Observations - discussion

Consider a differential square,  $dx$  by  $dy$  as the unit area of these fields, and the impedance is **377** ohms per square. Also consider combining these differential squares in series (along the E-field) and in parallel (along the H-field) to create a larger cross-sectional area within a TL boundary normal to the propagation. This is defining a transverse (shunt) cross-section of a “surface area containing differential squares”.



Robert Eisenhart

25

Determine a transmission line cross-section

## Impedance Observations - continuing

Think of a series ohms per square resistor in a printed circuit. This is the same concept but as a shunt ohms per square, only it is not a resistor but a relationship between E & H. The resulting impedance value as many unit squares are combined in unusual shapes like various transmission lines will be determined by the dimensions of the TL. This makes sense, as the impedance should only be a function of the cross-sectional dimensions, i.e. the

***SF***  $\equiv$  ***Shape Factor***

The TEM Characteristic Impedance is directly related to the Free Space Impedance

$$Z_c \rightarrow Z_o \frac{SF}{\sqrt{\epsilon_r}}$$

**Where:**

$$Z_o = 120\pi \text{ ohms} = (377) \quad \text{and} \quad \sqrt{\epsilon_r} = \text{dielectric constant}$$

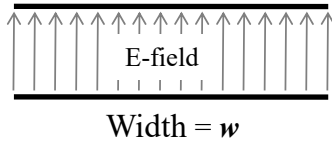
Consider a simple parallel plate TL

# Impedance of a Parallel Plate Line?

( $\epsilon_r=1$ , and not including fringing fields)

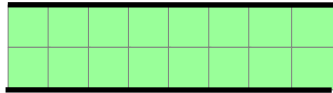
$$\text{Shape Factor} = \frac{h}{w}$$

$$Z_c = 377 \frac{h}{w} \Omega$$



Height =  $h$

Given a uniform electric field of width =  $w$  and height =  $h$ , calculate impedance by considering series and parallel squares of 377ohms/square



Height = 2

$$Z_c \approx 377 \frac{2}{8} = 94.25 \Omega$$

**This simple concept can be applied to various types of TEM lines to approximate the characteristic impedance**

Robert Eisenhart

27

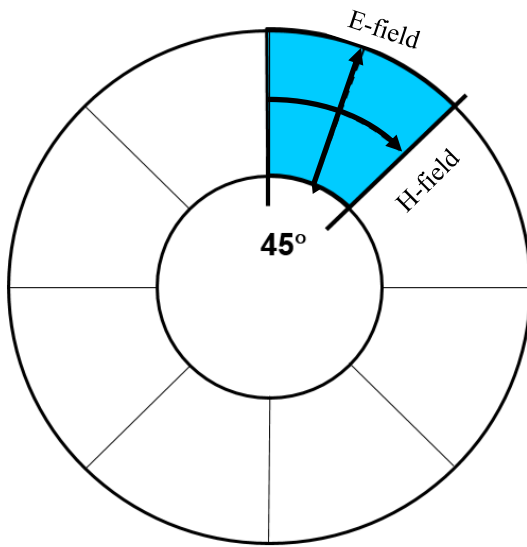
we have the Shape Factor as  $h/w$ .

Breaking the cross-section up into squares we have 8 in parallel with 2 in series.

Therefore the shape factor =  $1/4$ , and the  $Z_c = 94.25$  ohms. Fringing fields would add some parallel squares to in effect lower the impedance a bit.

Let's apply this to something a little more complex in shape.

# Impedance of a “50 ohm” coax airline?



**1.** Consider separating the field region of a 50 ohm air coax into 8 approximately equal “**squares**” of equal width and height. This is then 8 parallel regions of 377 ohms. Therefore, the characteristic impedance is:

$$Z_c = 377 \left( \frac{1}{8} \right) = 47.1 \, \Omega$$

**Add 377 ohms/square in series along the E-plane,  
and in parallel along the H-plane**

Robert Eisenhart

28

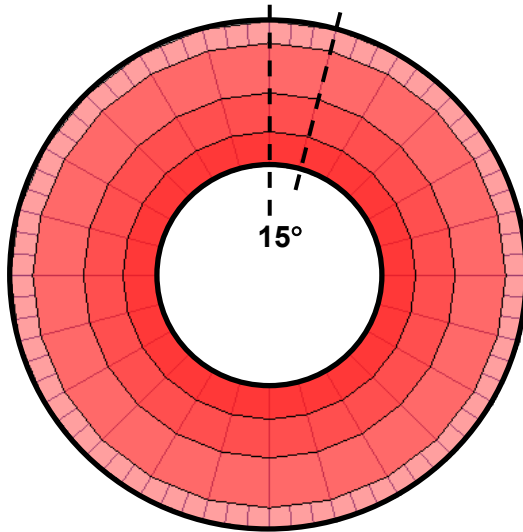
We'll calculate this  $Z_c$  two ways with two rough approximations.

Here's our approximate square, with eight in parallel, so we calculate 47 ohms for the impedance. (based upon the orientation of the E field)

There will be no fringing fields because the structure is closed.

However, let's try to get better looking squares by making them smaller.

## Impedance of a “50 ohm” coax airline?



2. Next, consider separating the same field region of this coax into 24 equal pie sections ( $15^\circ$ ) in parallel. Then each section will be 3 squares in series plus 3 squares near the outer edge in parallel. Therefore, the characteristic impedance is:

$$Z_c = 377 \frac{(3.33)}{24} = 52.3 \, \Omega$$

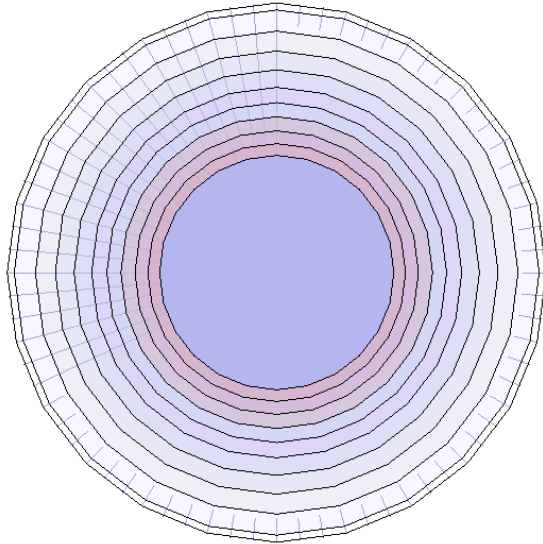
That is counting squares on a coaxial line where one  $15^\circ$  section is  $3 + 1/3 = 3.33$  squares. For 24 parallel sections.

**Both approaches give good estimates for the line impedance**

Using smaller differential regions does a bit better.

Both approaches are not bad, and, using smaller differential elements will produce a more accurate estimate as in the limit the error goes to zero.

## Going with smaller “squares” for the Coax



As the differential squares go to zero area, the estimate gets more accurate

Try counting squares on a coaxial line where one  $5^\circ$  section is  $9 + 1/2 = 9.5$  squares. For 72 parallel sections.  **$Z_c = 377 * (9.5/72) = 49.7$  ohms**

In the limit for the small differential square the estimate error goes to zero.

The point of all this is that I want someone to be able to look at the cross-section of a line and have a feel for the level of impedance, rather than just getting it from a formula.

From Section 2, TEM lines, page 9, here are some other Shape Factors on the next page

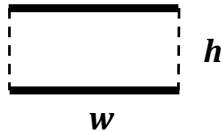
# Common Shape Factors

The Characteristic Impedance of a TEM line is scaled to the Free Space Impedance with modification by a Shape Factor and the material of the line. The Shape Factor is based only on the cross-sectional dimensions of the line. The material influence is by the line dielectric constant. So finally:

$$Z_c \rightarrow Z_o \frac{SF}{\sqrt{\epsilon_r}}$$

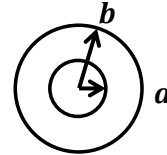
Parallel Plate

$$SF = \frac{h}{w}$$



Coax

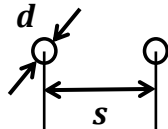
$$SF = \frac{\ln\left(\frac{b}{a}\right)}{2\pi}$$



Twin Lead

for  $\frac{s}{d} \geq 5$

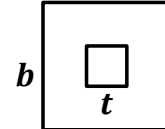
$$SF = \frac{\ln\left(\frac{2s}{d}\right)}{\pi}$$



Square-ax

for  $\frac{b}{t} \geq 2$

$$SF = \frac{\ln\left(0.9259 \frac{b}{t}\right)}{2\pi}$$

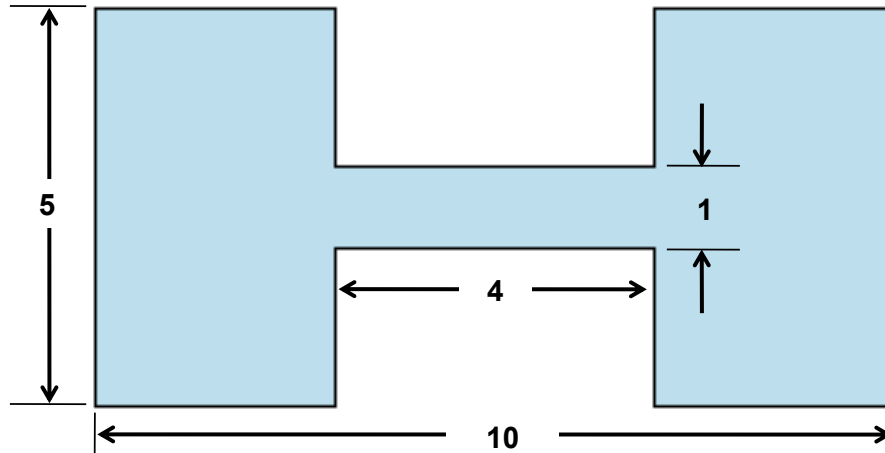


So let's try to calculate the  $Z_c$  of a ridge waveguide in the upper part of the band?

# Impedance Estimation?

Engineering is about estimation and approximation

Can anyone give a ballpark value for  $Z_c$  at 3x cutoff frequency  
for this double ridge waveguide?



**$Z_c = 75, 150, 300, 500$  ohms ?**

Robert Eisenhart

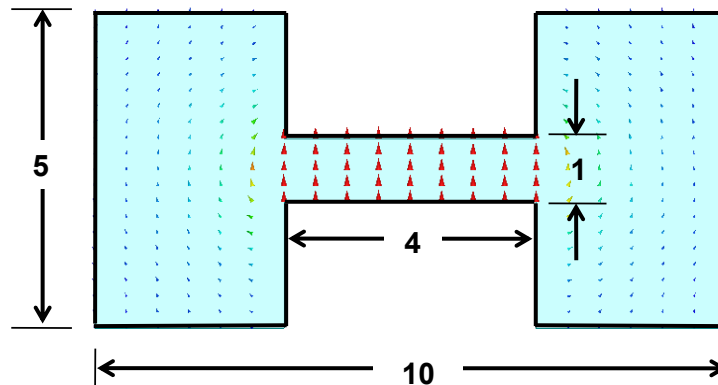
32

The answer is -



# Impedance Estimation

$Z_c$  at 3x cutoff frequency?



$$Z_c \approx 377 \frac{1}{4} < 94.25 \Omega$$

**Actual  $Z_c = 78 \Omega$   
including fringing**

Robert Eisenhart

33

Most of the fields are concentrated between the ridges so let's calculate for that region alone, ignoring fringing.

The fringing always adds a few "Squares" in parallel so the actual value is a little lower, and at 3 times cutoff it is almost a TEM line.

Next consider a discussion on Waveguide Simulation for large antenna arrays, which is a name for isolating mutual coupling effects in array antennas long before computers were around.

# Large Scanned Arrays

## Problem:

When scanning an array beam by changing the excitation of the elements, the resultant change of the mutual coupling also changes the match of the elements.

## Initial Hardware Solution (1963-1993):

Once upon a time there was an array analysis technique called **Waveguide Simulation**. This approach was complicated, hardware intensive, time consuming, very limited in results, but it was the only way to characterize a scanning array, **without building it first!**

## Enter Electromagnetic Simulation (HFSS 1993):

This discussion will address the concept that led to the **Linked Boundary Conditions** (LBC, i.e. Master/Slave) which HFSS uses today to describe large scanned arrays. My purpose is to provide an understanding of the basis behind Waveguide Simulation.

Robert Eisenhart

34

First, some WG Simulator background.

Early papers in 1963 Antennas and Propagation Transactions.

1995 first public presentation using HFSS- 1995 IEEE AP-S Symposium, Newport Beach, CA. Eisenhart and Park, Phased array scanning performance simulation,.

Chapter 10, *Finite Element Software for Microwave Engineering*, 1996 John Wiley & Sons.

# **Initial Waveguide Simulation Technique**

## **Purpose:**

To provide the *Active Element Pattern*: the element pattern as modified by the mutual coupling effects of the other array elements while scanning the mainbeam.

## **Issues:**

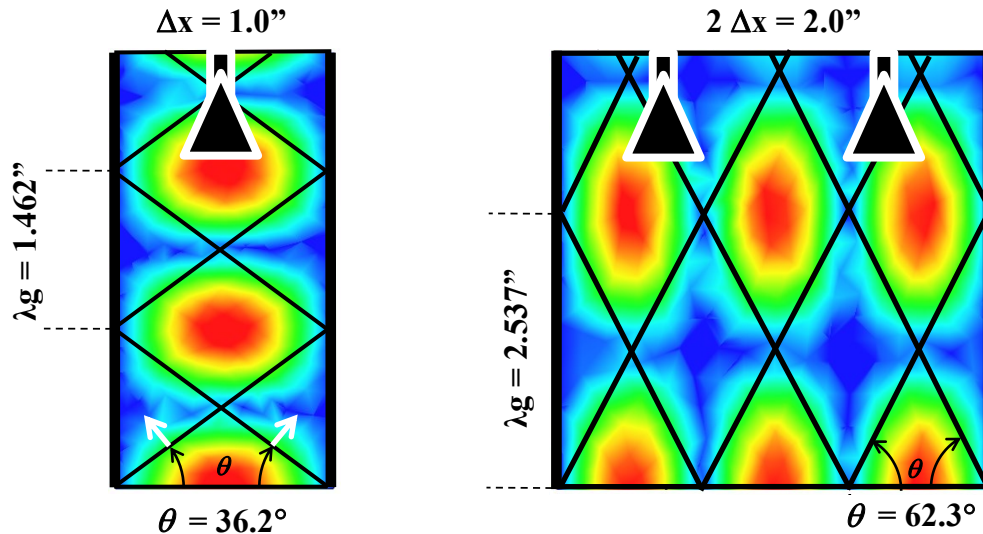
1. The technique is limited to a uniformly excited infinite array (approximation to a large array)
2. Only certain scan angle/frequency combinations can be simulated (does not include E-plane scan)
3. The maximum scan angle simulated is set by the element spacing.
4. Higher order modes are needed to simulate smaller scan angles (separate hardware is needed for each higher mode scan)
5. Hardware is very complex

This was the only way of defining the element match as a function of scan angle.

What are the limitations etc.

How does the Waveguide Simulation technique work?

# Waveguide Simulation Description



Two interfering waves at angle  $\pm \theta$ , as a result of the reflections off the side walls.

$$\theta = \sin^{-1} \left( \frac{m \lambda_o}{2M\Delta x} \right) \quad \text{where} \quad \lambda_o = 1.18'', \text{ then}$$

with  $m \equiv \text{mode \#}$  and  $M \equiv \# \text{ of cells}$

$m$	$M$	$\theta$
1	1	36.2
1	2	17.2
2	2	36.2
3	2	62.3

Robert Eisenhart

36

Consider a simple waveguide with dimensions set by the array lattice. The fundamental TE<sub>10</sub> mode is the superposition of two waves bouncing back and forth between the two sides at  $\pm \theta$ , with the two wave-fronts indicated by the lines.

And we see how the fields fit in the WG, at a snapshot in time.

If we put an antenna element as a termination at the end of the waveguide and determine the reflection coefficient, it will represent the match for an incident wave at  $\theta$ . Being reciprocal, this will also be the input match for the element when in an array scanned at  $\theta$ . And knowing rho we can now determine the active element pattern value for that  $\theta$ , only one scan angle. Now all we need are a lot more scan points.

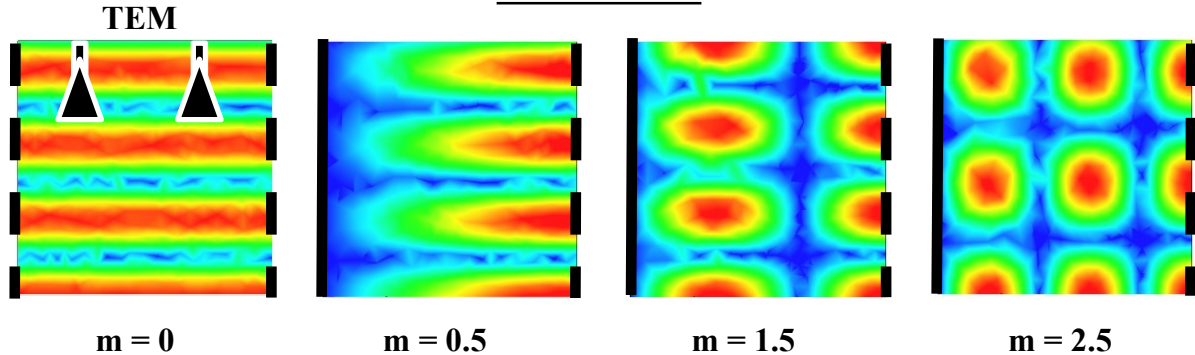
If we double the width of the WG, such a cell would hold two elements, and up to three modes can now propagate at the corresponding angles with this double-wide cell.

So sees that we have a summary of the four modes (ie scanning angles) that can now exist. We see that the single WG is redundant if we build the larger cell. These waveguide configurations dictate only the scanning angles and have nothing to do with the actual antenna elements, one per cell.

Along comes EM Simulation, and while simultaneously simulating multiple modes it also gives us a new boundary, the Magnetic wall, not available with just hardware.

# Waveguide Simulation Using HFSS



All cells  $2\Delta x$  wide



Two interfering waves with sidewall boundaries:

Again  $\theta = \sin^{-1}\left(\frac{m\lambda_o}{2M\Delta x}\right)$  but now “m” can be zero

and also have “half values” because of the Magnetic walls.

Magnetic Wall   
Electric Wall 

The resulting angles are in between the angles from the all metal waveguide multi-cell on the previous page

$m$	$M$	$\theta$
0	2	0
0.5	2	8.5
1.5	2	26.3
2.5	2	47.5

Robert Eisenhart

37

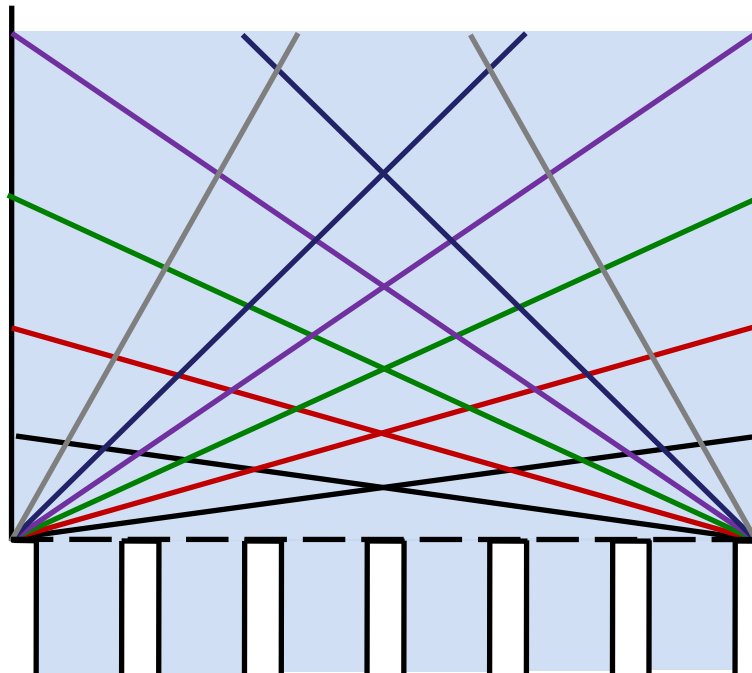
Taking advantage of the Mag walls we immediately are able to simulate the TEM, i.e. broadside or 0 deg scan condition which was not possible with the hardware only simulation. (This is also the missing E-plane scan)

This double cell would take two antenna elements. Now, also using one Mag wall and one Electric wall we can get intermediate scan angles relative to the all metal walls.

Note that we picked up 4 new scan angles

And, since HFSS gives us all mode info at once, why not do a larger multi-cell simulation?

# Scan Angles in a Multi-Cell Waveguide



Waveguides representing 6 antenna elements

<u>Mode</u>	<u>Angle</u>
	0.0°
	4.2°
<b>TE<sub>10</sub></b>	<b>8.5°</b>
	12.8°
<b>TE<sub>20</sub></b>	<b>17.2°</b>
	21.6°
<b>TE<sub>30</sub></b>	<b>26.3°</b>
	31.1°
<b>TE<sub>40</sub></b>	<b>36.2°</b>
	41.6°
<b>TE<sub>50</sub></b>	<b>47.5°</b>
	54.2°
<b>TE<sub>60</sub></b>	<b>62.3°</b>
	73.5°

Robert Eisenhart

38

Here we have a 6-cell model, which gives 6 scans and reflection data (S-matrix), all in one simulation resulting from 6 modes.

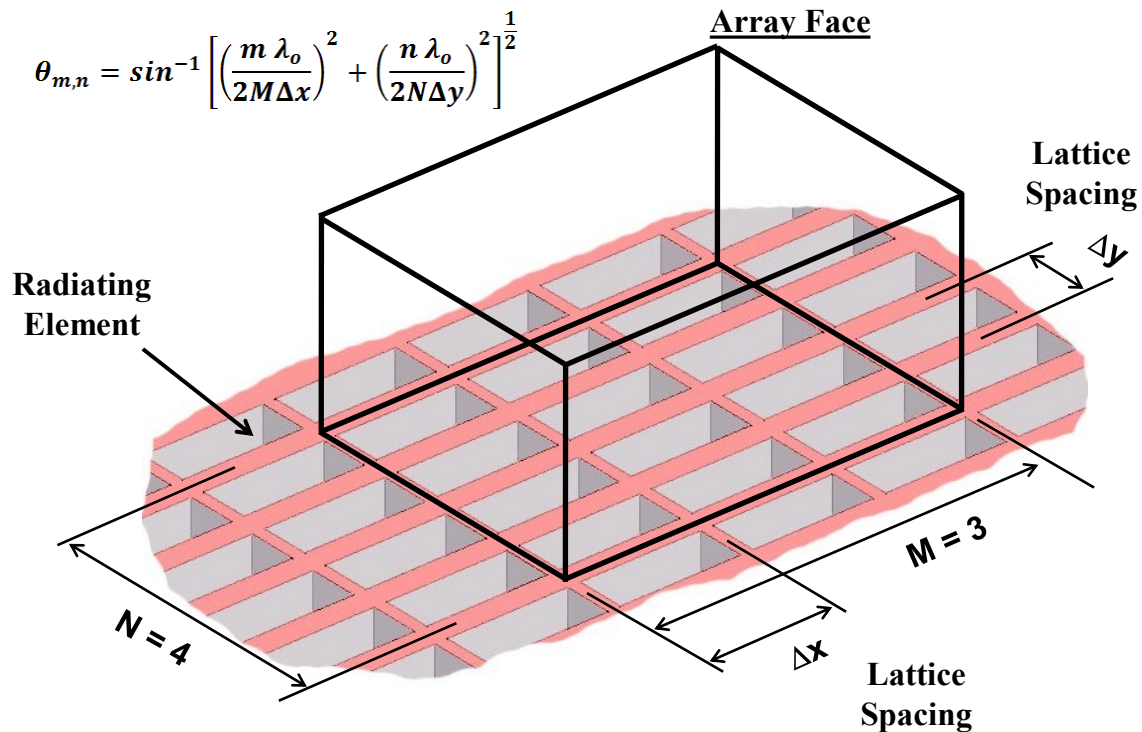
And using Mag walls will also give the 0 deg scan plus 7 additional scan values, for a total of 14 data points. And all we did was simulate two different multimode waveguides with different boundary conditions.

You can see how this concept can be extended to more modes if needed. The above examples deal only with the H-plane scan.

The same approach can be used to scan in the E-plane and all other planes.

Scans off the principle axes require a two dimensional multi-cell as seen in the next page.

# Scan Angles for Off Axis Modes



Robert Eisenhart

39

You can imagine how complicated this hardware would be with 12 antenna elements and a mode launcher for an off axis mode ( $m \& n \neq 0$ ).

Unfortunately, hardware could only be designed to deal with one mode at a time so a separate piece of specialized hardware was made for each freq/scan point of interest.

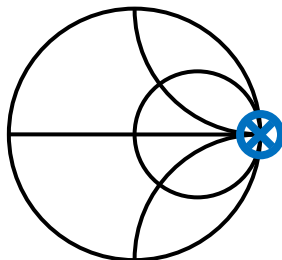
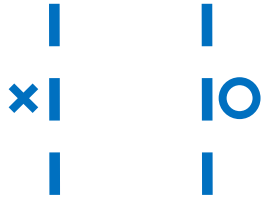
Now that we understand “Waveguide Simulation” and how HFSS makes it a lot easier. . . What about Master/Slave boundaries?

Consider going back to the two Mag wall case.

# Scan Angles Master/Slave Waveguide

## Magnetic Wall

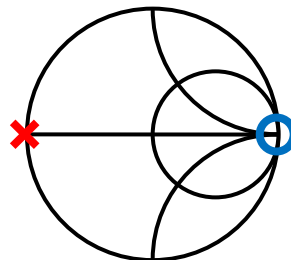
For a phase shift of:  
 $0^\circ, 360^\circ$



We get a Scan  $\theta$  of:  
 $0.0^\circ, 36.2^\circ$

## Electric Wall

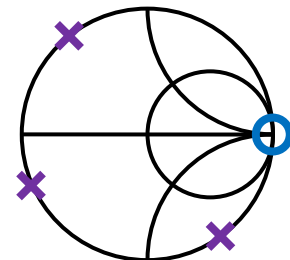
For a phase shift of:  
 $180^\circ$



We get a Scan  $\theta$  of:  
 $17.2^\circ$

## Reactive Wall

For phase shifts of:  
 $X1, X2, X3$



We get Scan  $\theta$ s of:  
 $Y1, Y2, Y3$

Robert Eisenhart

40

Start with the same single unit cell defined by the array lattice and for now just consider the H-plane scan.

Imagine the two walls as reactive surfaces as shown on the Smith chart, the circle for the right wall and the cross for the left wall. As magnetic walls they both are open circuits at the right side of the chart at infinite reactance. With respect to the Smith chart, they are either 0 or 360 degrees apart.

Resulting scan angles are 0 and 36.2 degrees.

If we change the left wall to electric, the reactance is at zero and shown by the red cross.

With a phase shift now of 180 deg, the resulting scan angle is 17.2 deg.

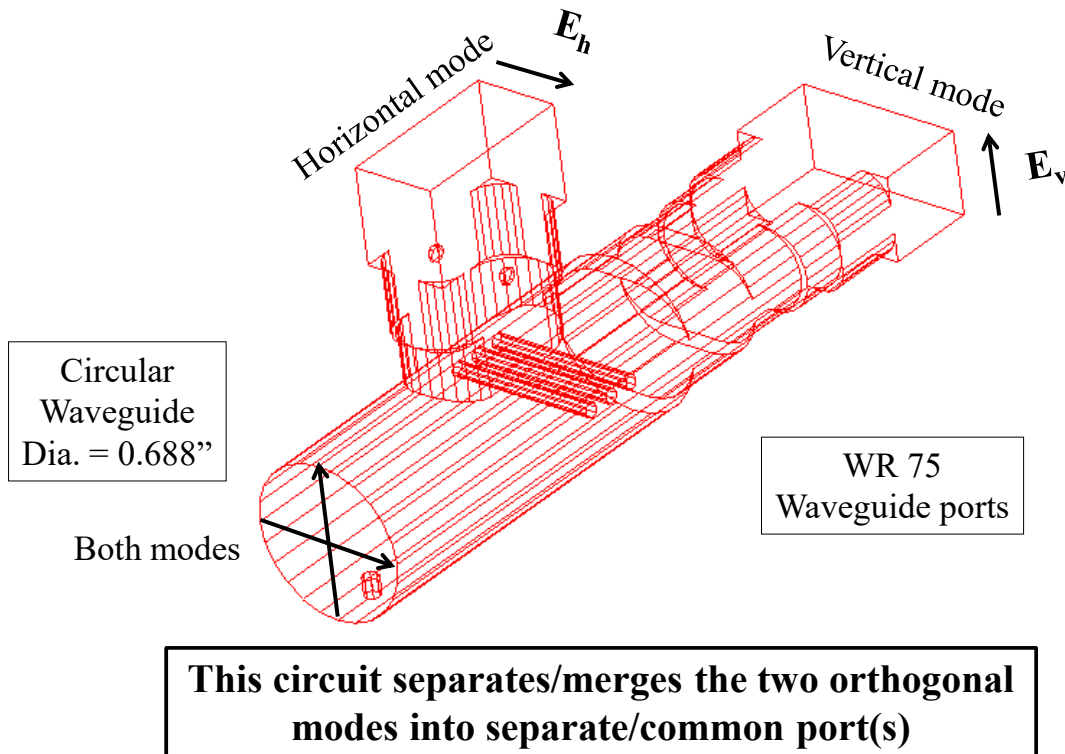
Now suppose we could mathematically construct the arbitrary reactive wall shown in the third case, for any reactive value around the edge of the Smith chart.

One would suspect that these reactive values would lead to intermediate scan angles, and in fact, allow continuous scanning from 0 to the maximum scan possible. Voila, you have what HFSS calls Master/Slave.



Let's look at an orthomode transducer.

# Ku-band Orthomode Transducer



Robert Eisenhart

41

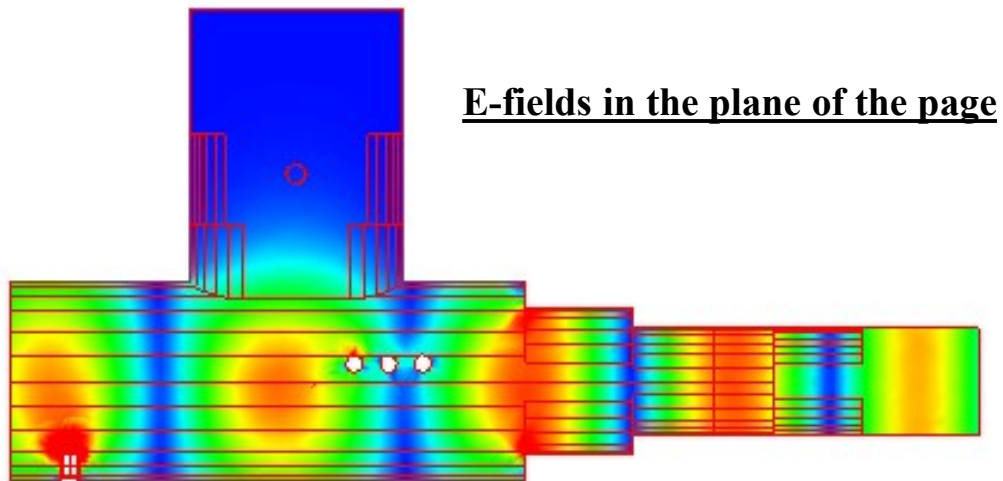
Horizontal mode, defined by the Electric field orientation.

Vertical mode

What would we get out of the linear ports if we put a circularly polarized mode in the circular WG port? Sine varying amplitude.

Look at the vertical mode

# Vertical Mode E-Fields



**Snapshot of the Vertical mode ( $E_v$ ) passing through**

Robert Eisenhart

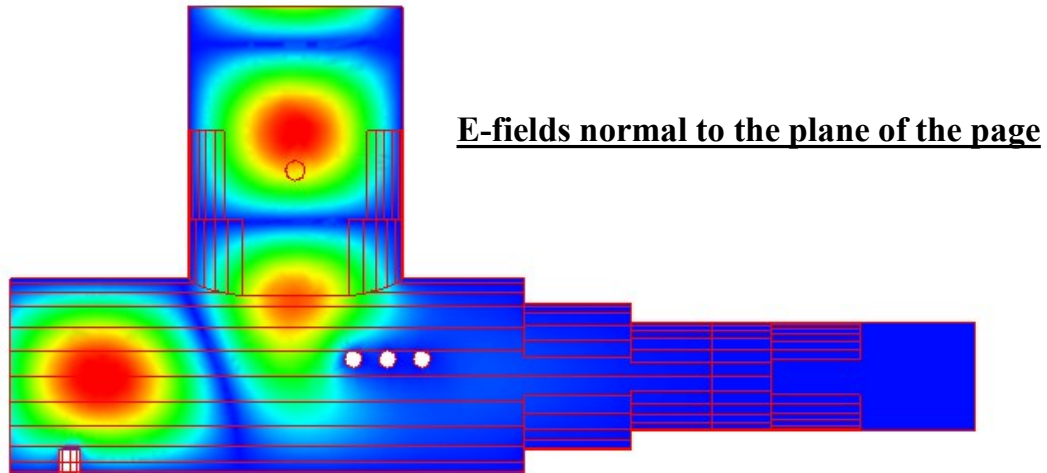
42

This mode's fields are cross polarized to the orientation of the upper waveguide so they are below cutoff and cannot propagate in that WG.

Also this mode is not affected by the mode reflector for the horizontal mode, see the three small circles that are horizontal bars in the waveguide.

And the horizontal mode?

# Horizontal Mode E-Fields



**The Horizontal mode is redirected by the mode filter into the upper WG**

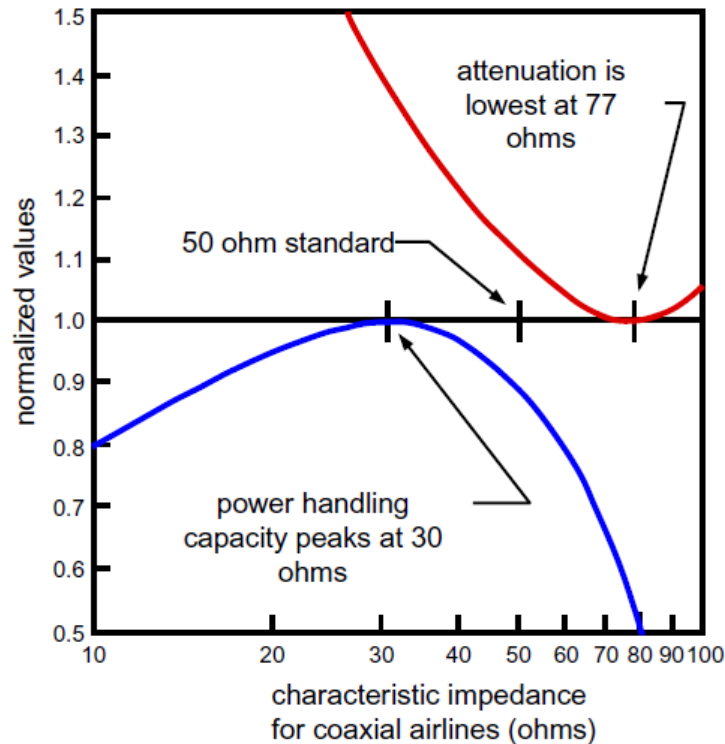
Robert Eisenhart

43

Viewing the fields is always a good check to make sure everything is working as expected. Note this mode is reflected by the three bars.

Ever wonder why 50 ohm coax is the standard line impedance?

# Why 50 ohms for Coax?



Robert Eisenhart

44

The value 50 ohms is a compromise between attenuation and power handling.

Normalized to a fixed outer diameter - we see the power peaks at 30 ohms and the loss is minimum at 77 ohms.

(Power is considered limited by a max E-field on the center conductor where the density is max.)

**Logic will get you from A to B.**

**Imagination  
will take you everywhere.**

Robert Eisenhart

45

Innovation is the step up in Engineering