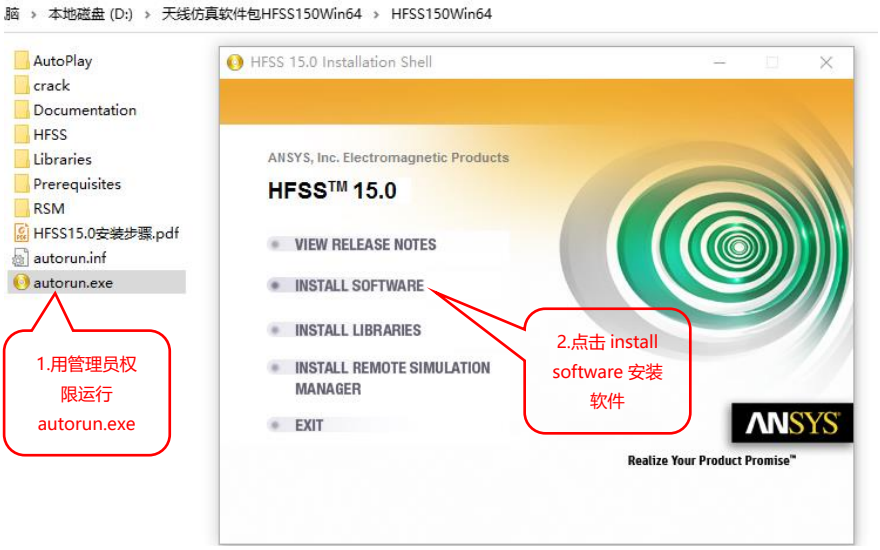
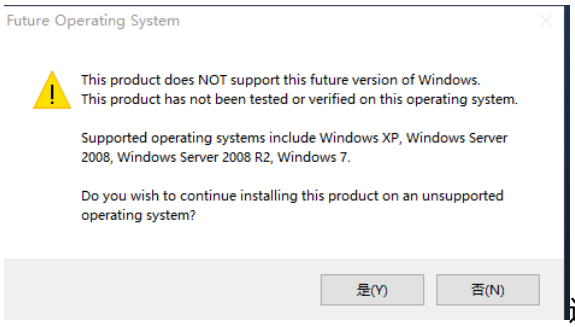
HFSS 天线设计仿真及 smith 圆图分析

作者:向仔州

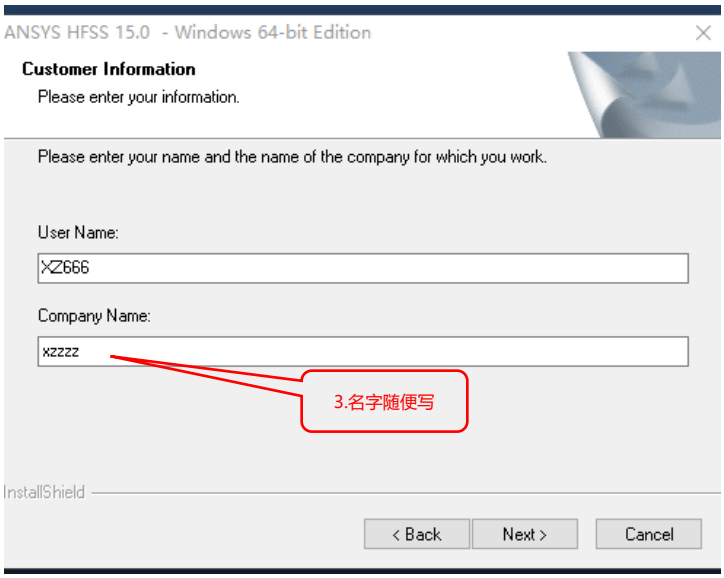
HFSS15.0 安装

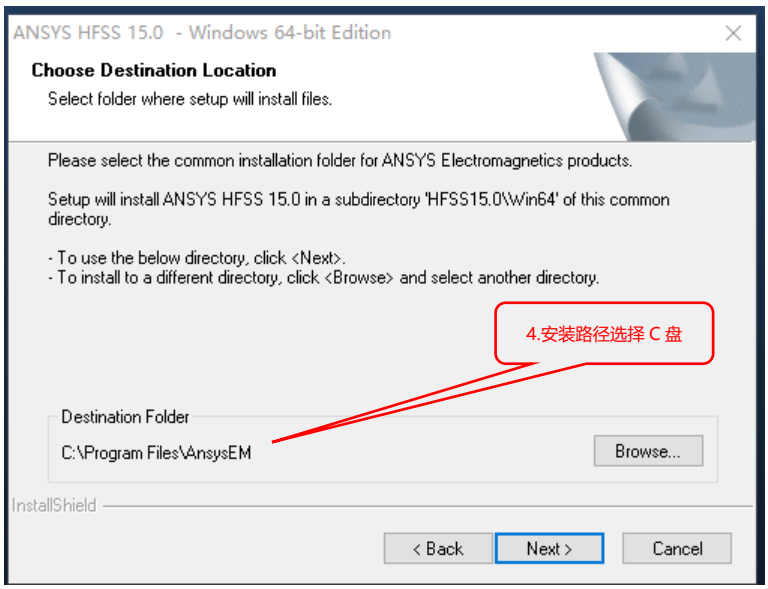
首先关闭 360，电脑管家等杀毒软件

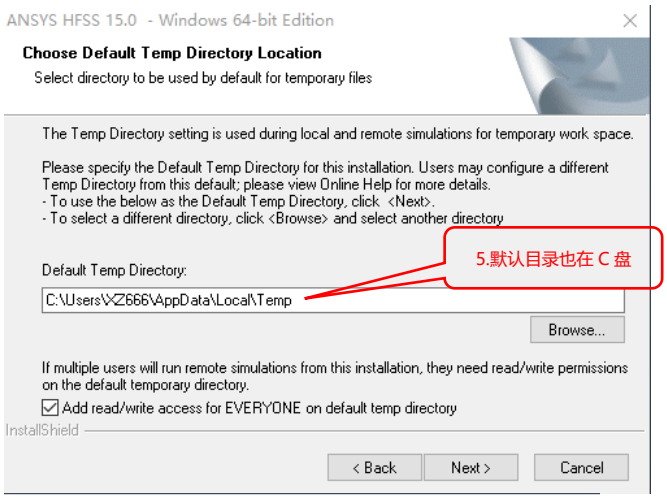


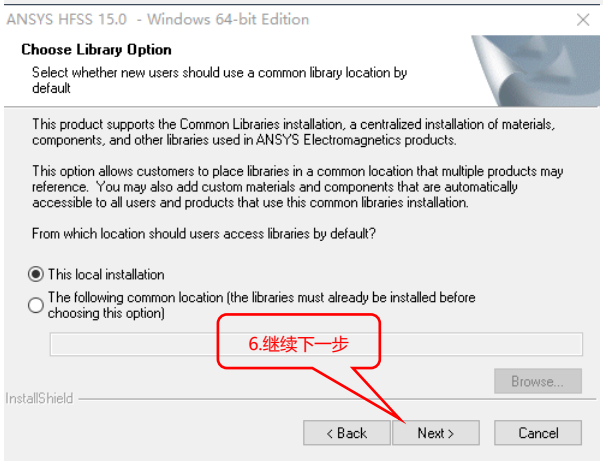


这里说软件只能在 win7 上安装，不理它，我们点击‘是’

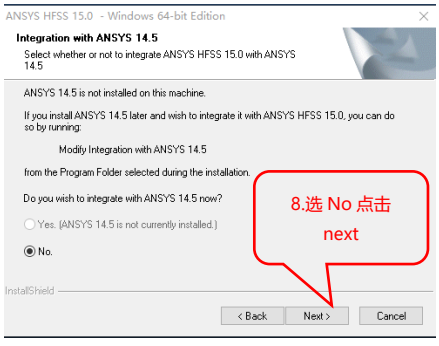


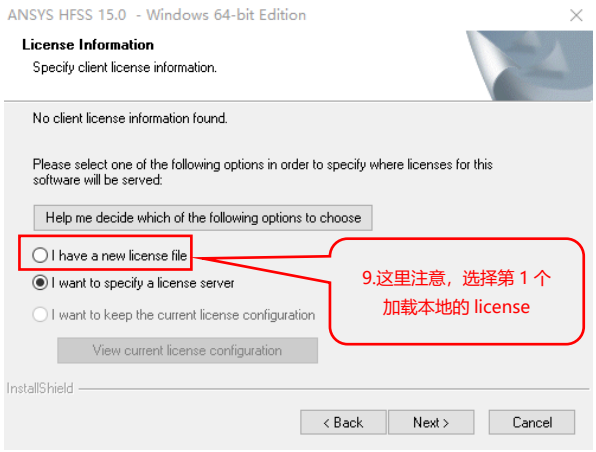






第 7 步也是选择 next

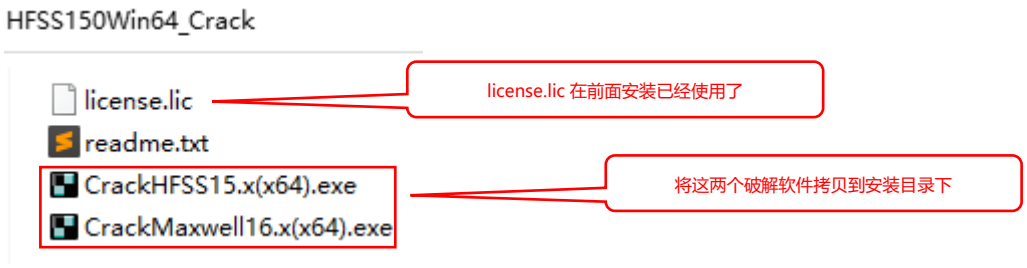


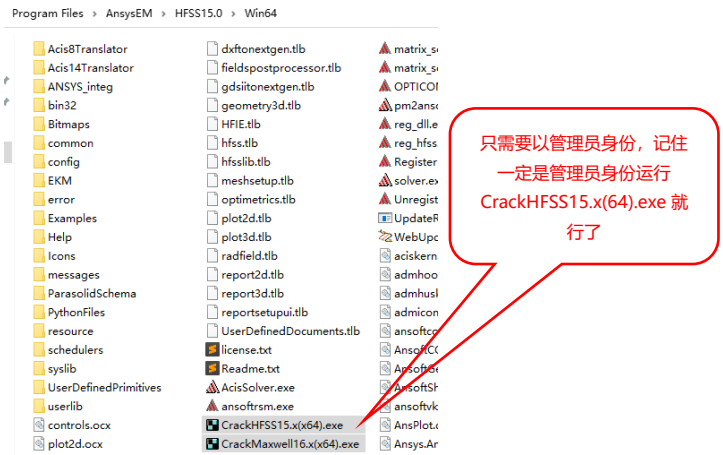


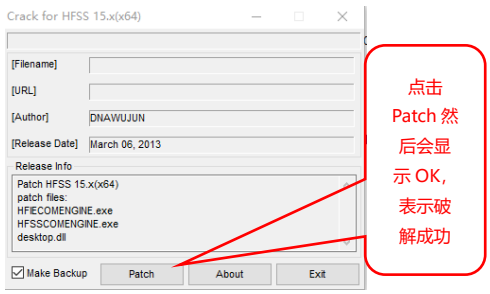
进入安装进度

软件破解方法

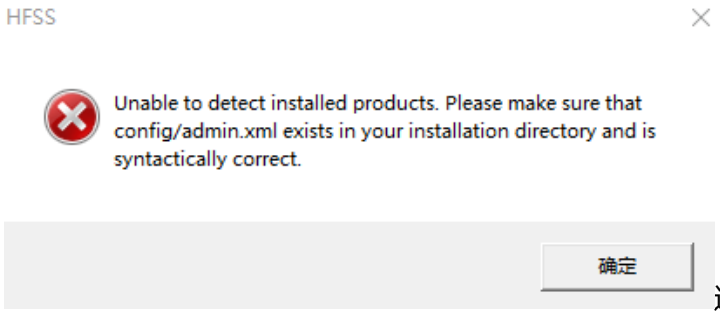
HFSS150Win64_Crack.zip 选择压缩包里面的破解文件





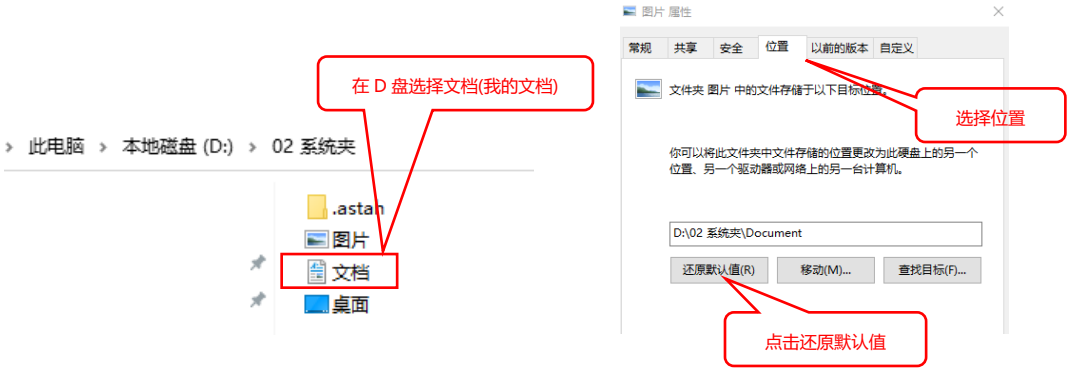


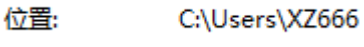
运行软件出现 unable to detect installed products please make sure that config/admin.xml 这个问题一定要谨记



运行 HFSS 出现了这个错误

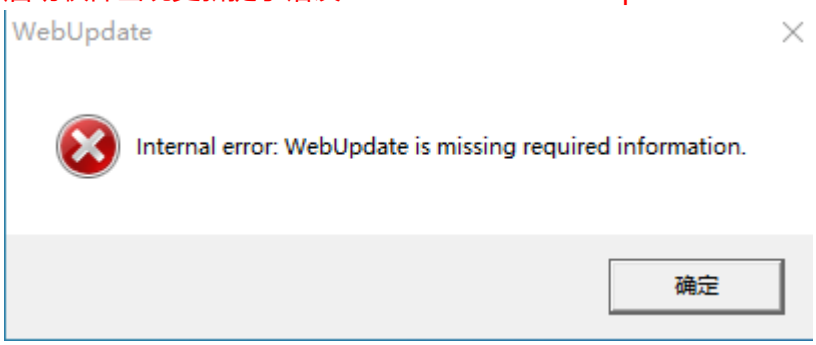
这个错误是因为在安装 windows10 系统的时候犯了个错误，把”我的文档”(windows10 可能也叫文档)默认安装在了 D 盘，但是 HFSS 软件官方默认是去 C 盘找“我的文档”（文档），所以造成了软件无法启动的情况。





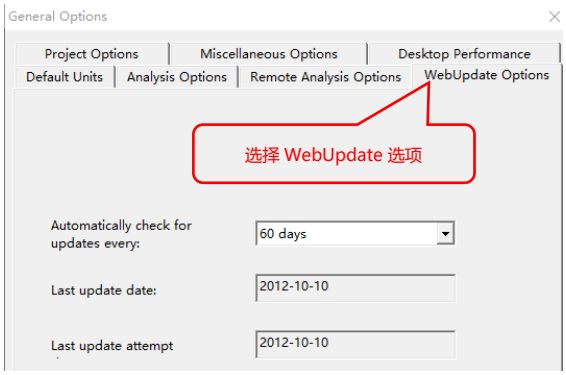
系统会显示还原到 C 盘位置，点击应用确定就可以了

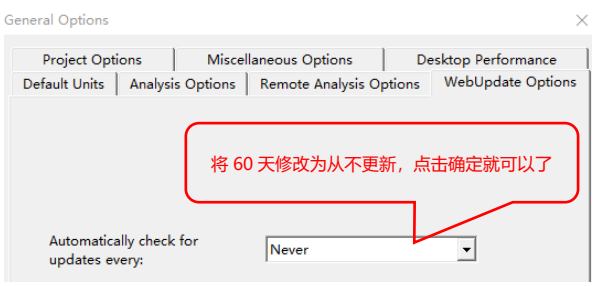
启动软件出现更新提示错误 internal error: WebUpdate is missing required information

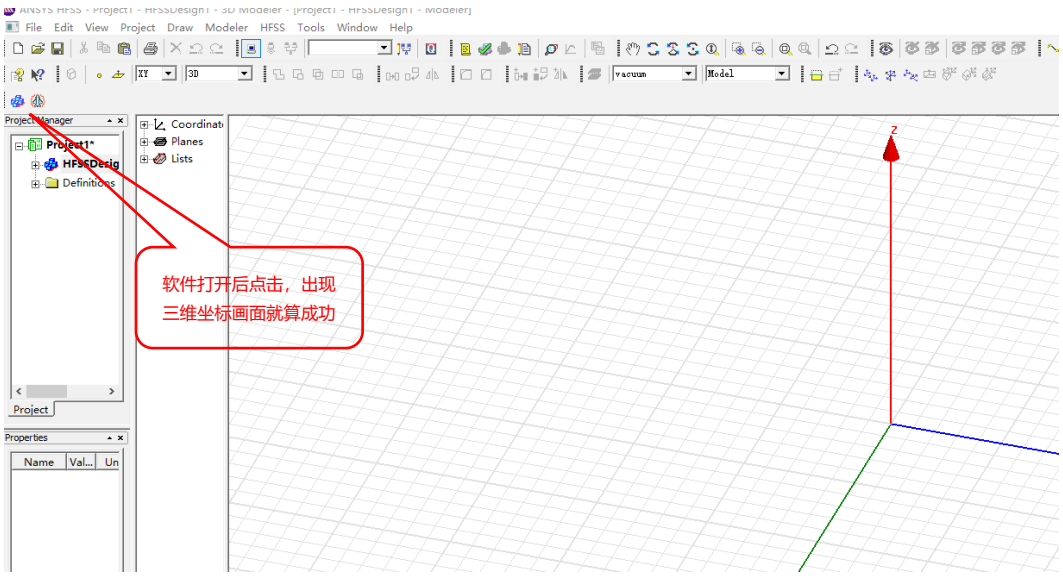


这个错误不影响软件使用，只是每次运行软件都会显示这个很烦。

在软件中，菜单栏上点：Tools》Options》General Options》

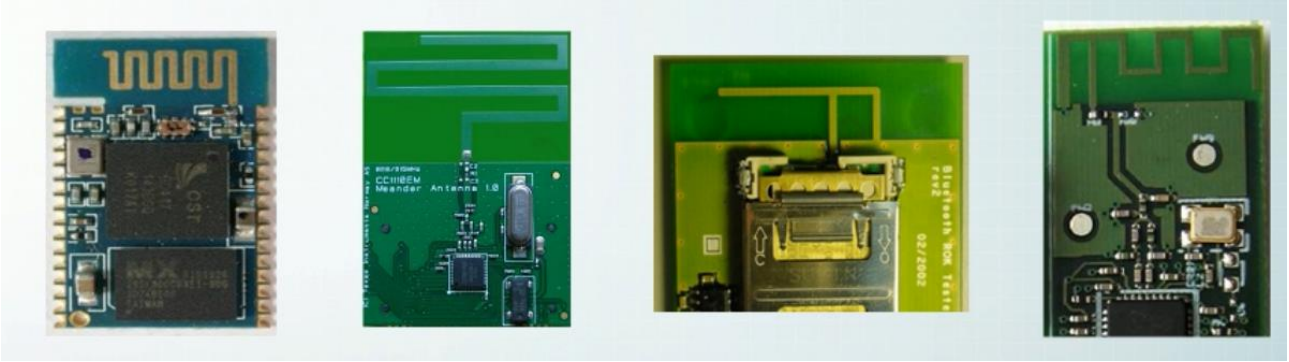






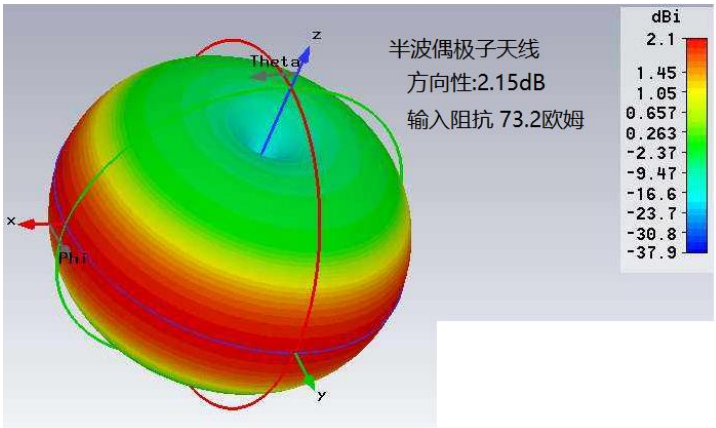
HFSS 里面的工程只能保存在英文路径的文件夹下。

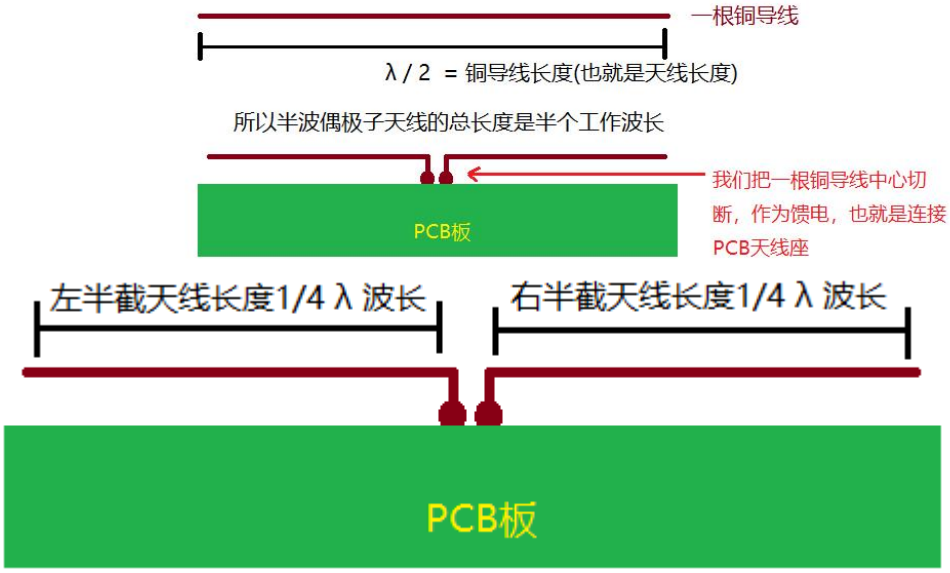
PCB 天线知识



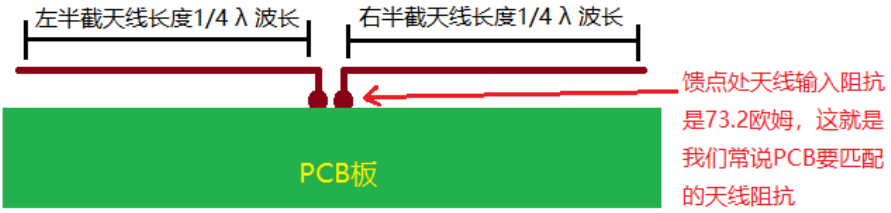
这 4 种都是 PCB 天线

天线的雏形如下



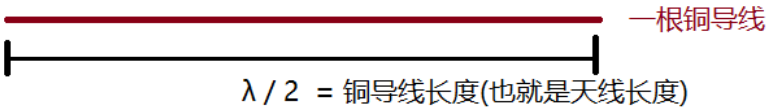


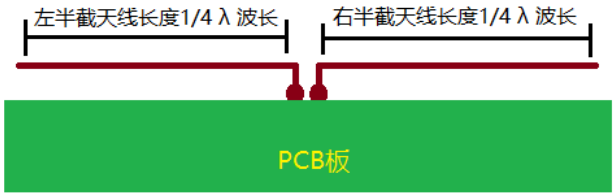
所以天线的长度与波长有关。



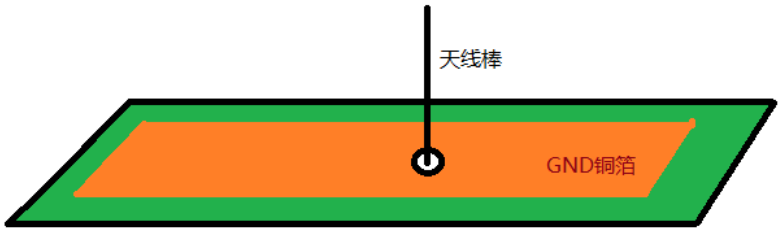
半波偶极子天线很长，因为根据前面波长公式就知道。

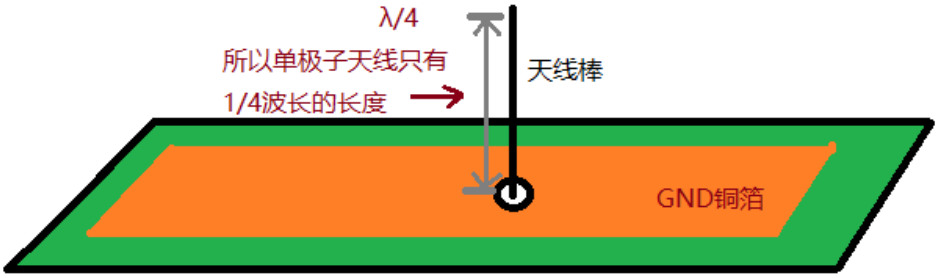
为了降低天线的长度，我们又发明了单极子天线





单极子天线就是将半波偶极子天线的一端取掉，用GND铜箔来代替



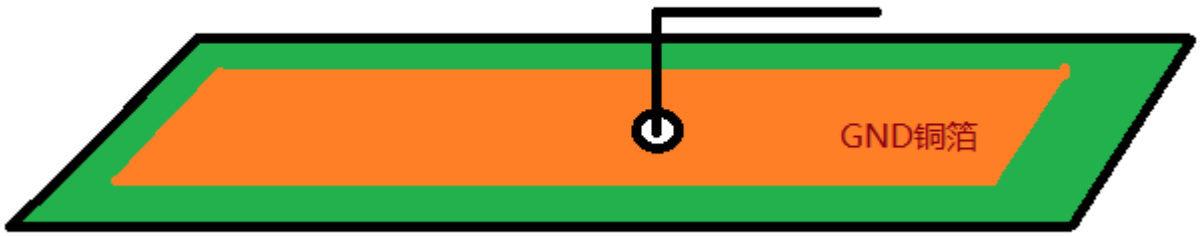


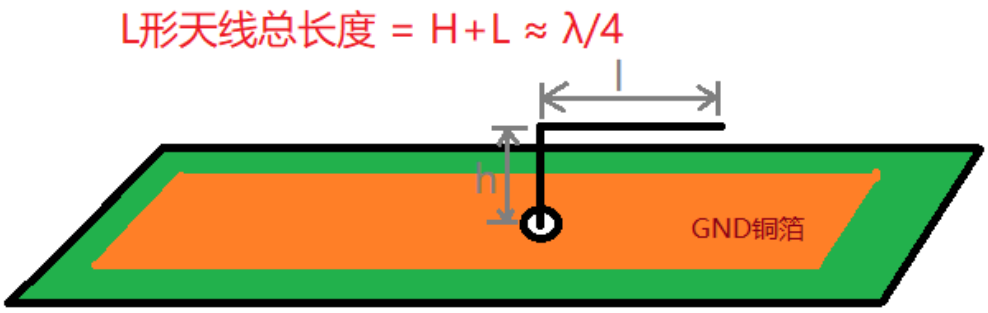

单极天线实物

因为只有 1/4 波长的长度，单极子天线就要短得多。只有偶极子天线长度的一半。
单极子天线方向性: 2.15dB +3dB，所以单极子天线比偶极子天线方向性多 3dB。
单极子天线输入阻抗只有偶极子天线的一半，37.6 欧姆。

在 PCB 上，单极子天线的长度还是太长了。

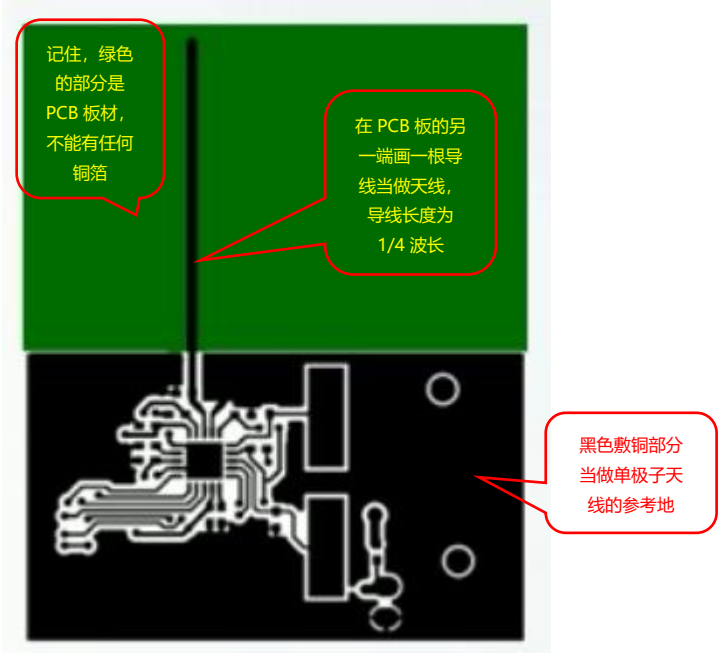
有一种办法，就是把单极子天线弯折一下，做成L形天线



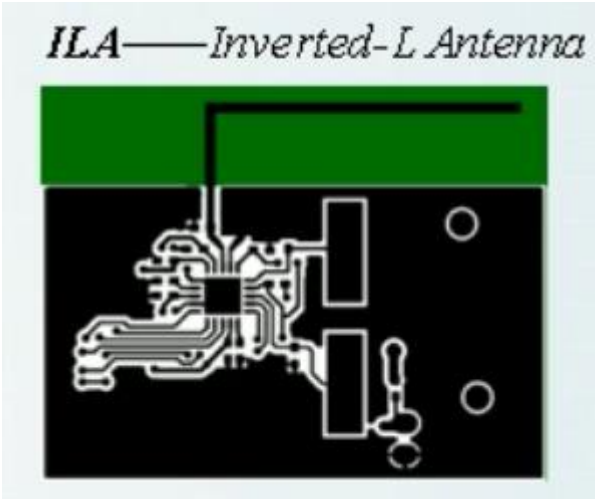


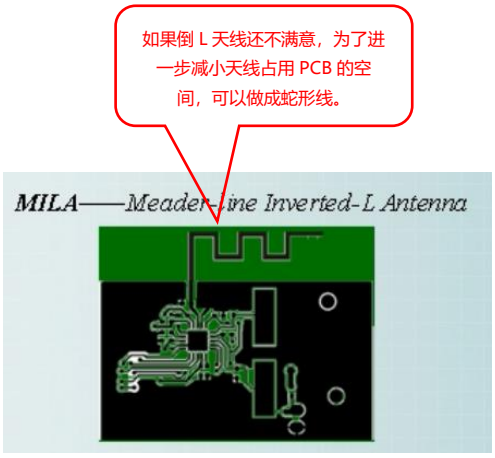
这种天线就是倒 L 天线，简称(ILA)

我们将以上单极子天线和倒 L 天线移植到 PCB 上

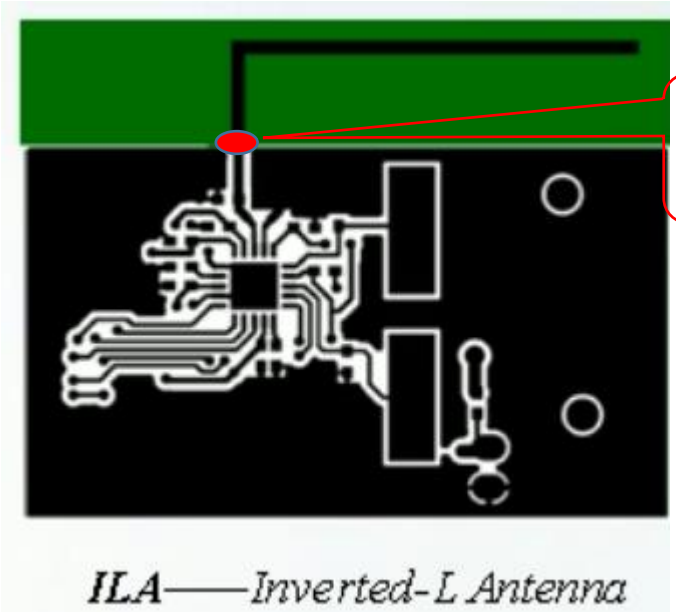


为了节省 PCB 空间，我们可以把 PCB 上的单极子天线弯折一下，做成倒 L 天线

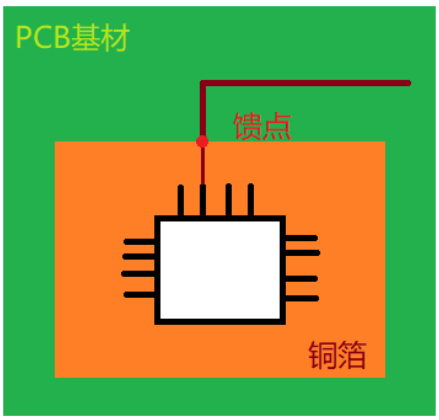




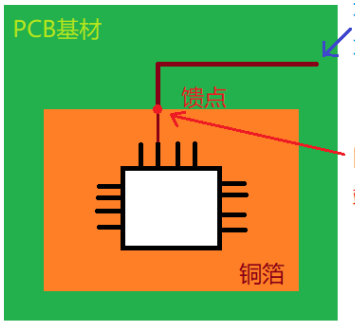
所以 PCB 上的倒 L 天线，直线，还是蛇形线，都是单极子天线的变形。



PCB 倒 L 天线阻抗调试方法，引入倒 F 天线

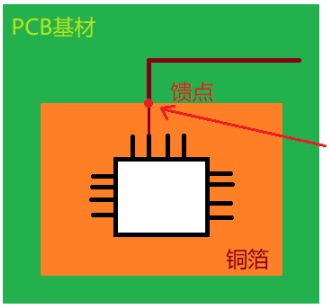


有可能连接芯片天线输出端的馈点没有50欧姆，也就是天线馈点没有50欧姆，无法和芯片输出端匹配

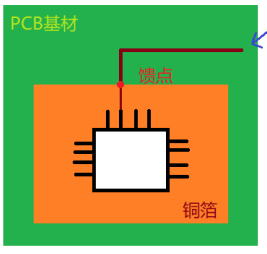


天线的末端，电压最大，电流最小

因为馈点是倒L天线的输入端，电流最大电压最小

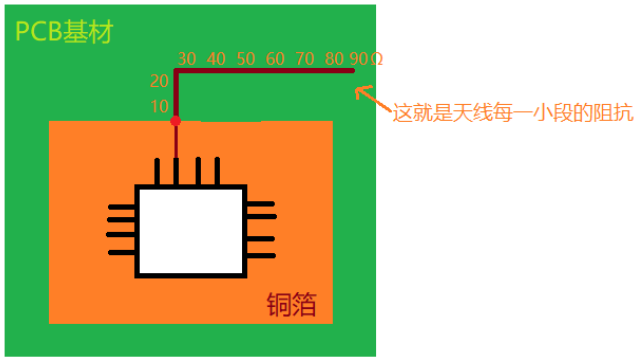


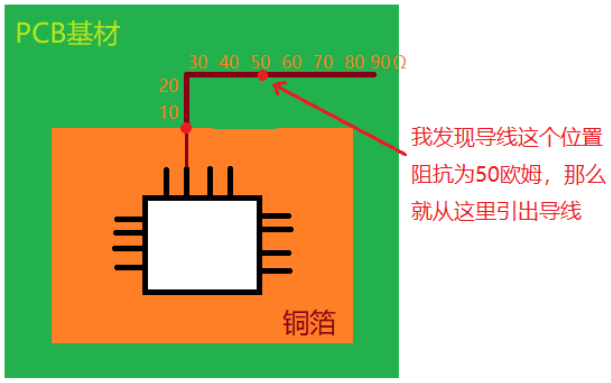
根据阻抗公式R = U/I
馈点，电压最小，电流最大，那么R阻抗就很小，所以馈点阻抗很小

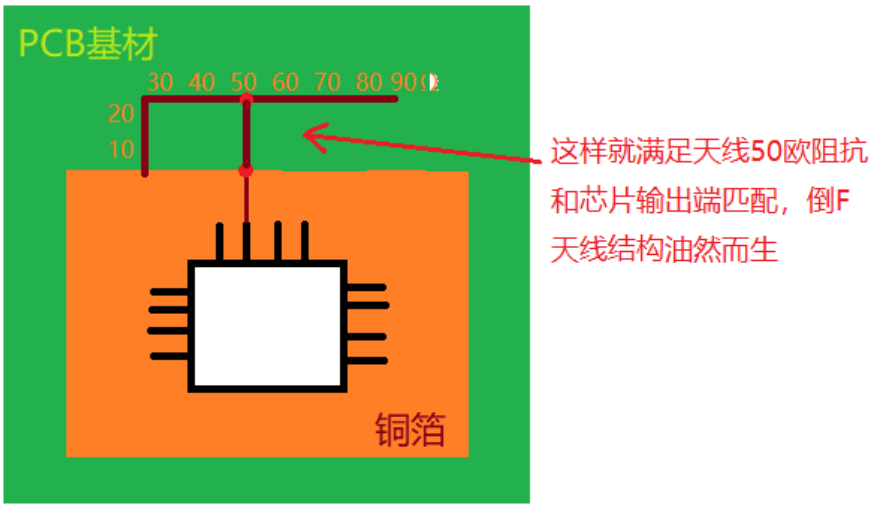


根据R=U/I，天线末端电压最大，电流最小，所以天线末端阻抗最大

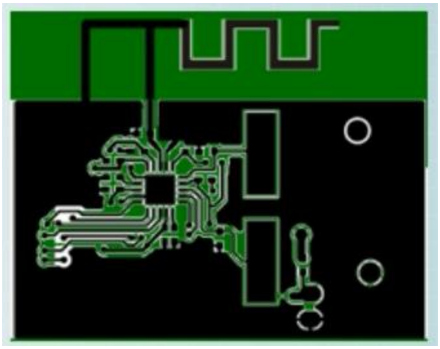
所以根据以上原理我们知道，天线的阻抗从馈点出来，到达天线末端，阻抗是逐步增大的。







如果要进一步减小天线尺寸



可以做出蛇形倒 F 天线

对于工作在 2.4G 频段的蓝牙或者 wifi 来说，它们天线都采用单极子天线变形结构。如倒 L 或者倒 F 天线

影响 PCB 天线的参数有如下:

PCB 板材介电常数，(通常 FR4 板材，Er = 4.4)

PCB 板厚

PCB 板参考地平面尺寸

PCB 板上，天线走线宽度

PCB 板上天线布线的位置，还有外壳因素。

工作在 2.4G 的蓝牙和 wifi PCB 天线仿真设计

Bluetooth/wifi 工作在 2.4Ghz 的 ISM 频段(2.4 ~ 2.4835Ghz)，中心频率取 2.45Ghz

现在给出 PCB 板信息:

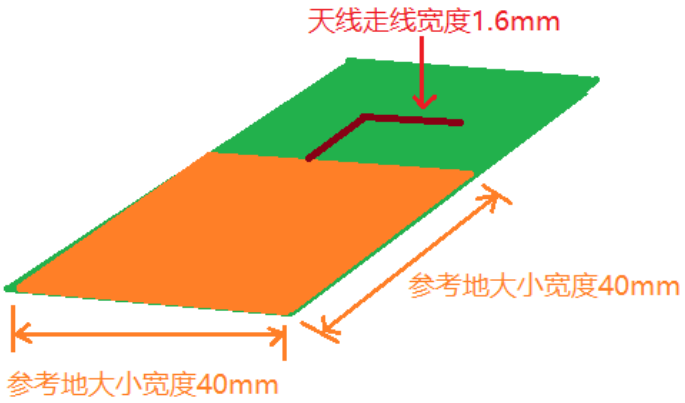
介质材料: FR4 , Er = 4.4

板厚: 1mm

天线走线宽度: 1.6mm

PCB 板尺寸: 65mm x 40mm

参考地大小: 40mm x 40mm



既然PCB天线走线宽度是1.6mm，那么PCB天线长度是多少呢?

因为天线导线底部处在PCB介质上，但是导线顶部面对的是空气

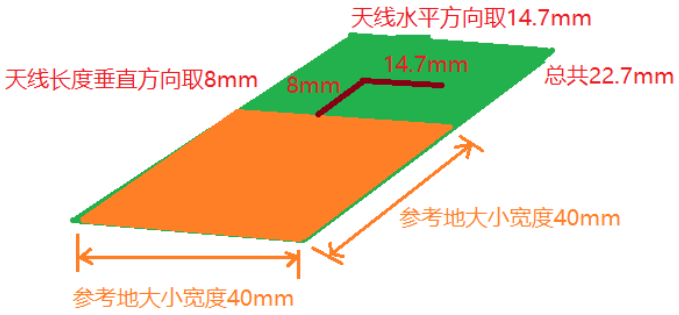
$$\text{那么空气部分的自由空间波长} = \frac{\text{光速}}{\text{工作频率}} = \frac{3 \times 10^8}{2.45 \times 10^9} = 0.122\text{米} = 122\text{mm}$$

我们只能取1/4波长 = 122mm / 4 = 30.6mm

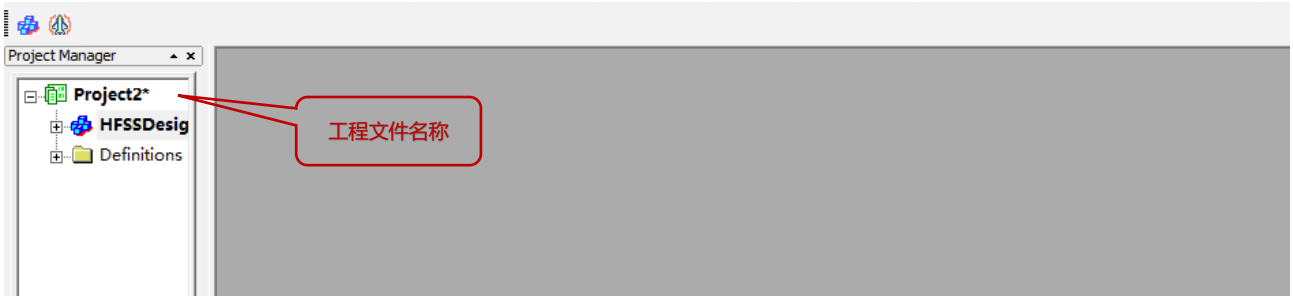
$$\text{介质波长} = \frac{\lambda\ (\text{自由空间波长})}{Er\ (\text{有效介电常数})} = \frac{30.6\text{mm}}{\sqrt{4.4}} = 14.58\text{mm}$$

PCB介电常数取根号

所以PCB天线长度 取自由空间波长到介质波长的中间 30.6mm到14.58mm之间，取22.7mm



1.打开 HFSS 软件

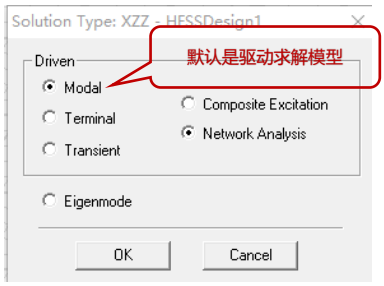


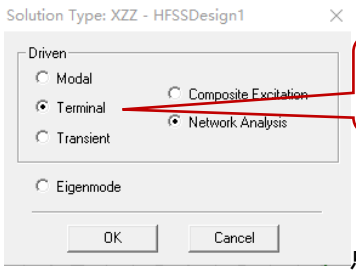
2.点击顶部 project->insert HFSS Design，创建 HFSS 工程

一般新建立的 HFSS 工程都是 DrivenModal 模式驱动求解类型。

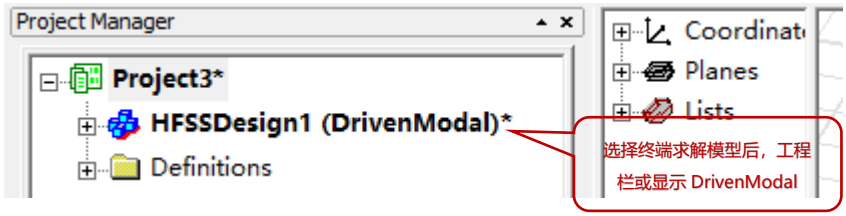
我们天线仿真需要终端驱动求解类型，但是 DrivenModal 模式驱动也是可以的，现在只用终端驱动求解。

3.点击顶部 HFSS->Solution Type

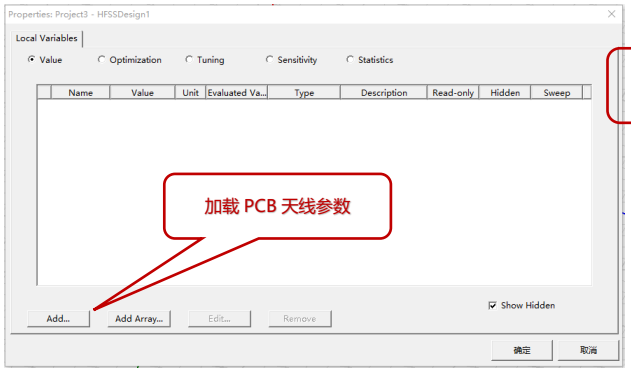




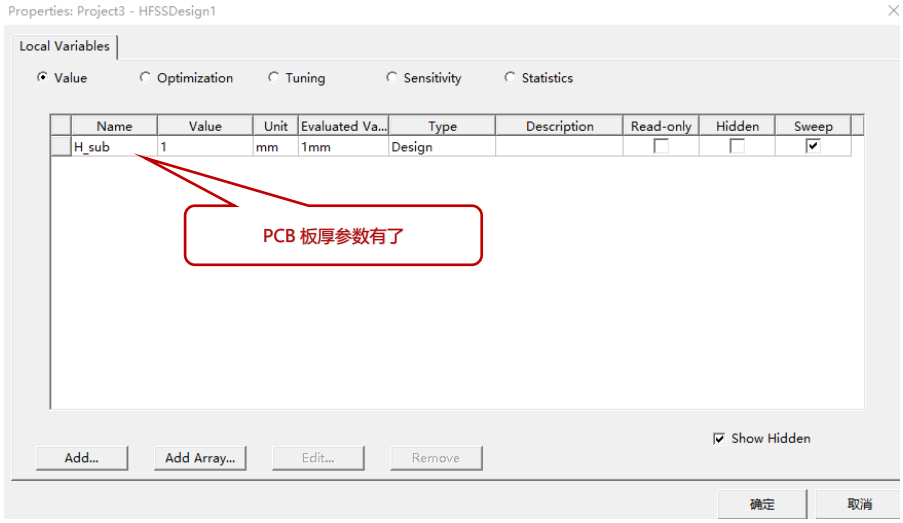
点击 OK



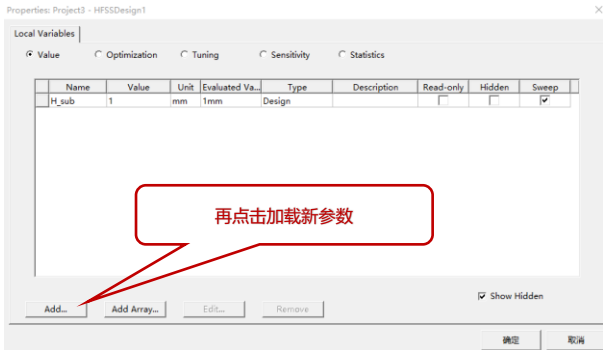
4.设置 PCB 天线一系列参数，选择 HFSS->Design Properties

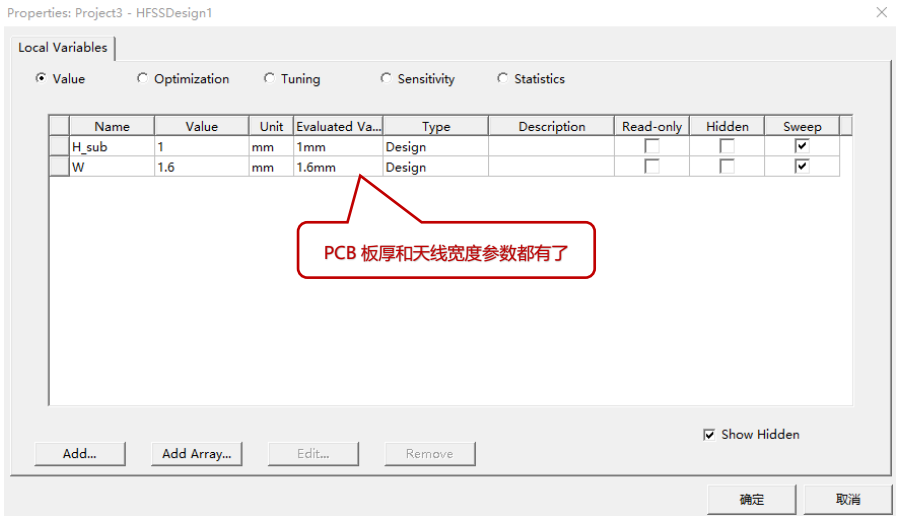


设置介质层的厚度： 因为 PCB 介质层厚度就是 PCB 板材厚度，这里 PCB 板厚取 1mm



设置天线走线宽度: PCB 天线宽度设置为 1.6mm





设置了 6 个变量

H_sub 变量：　设置的 PCB 厚度 1mm

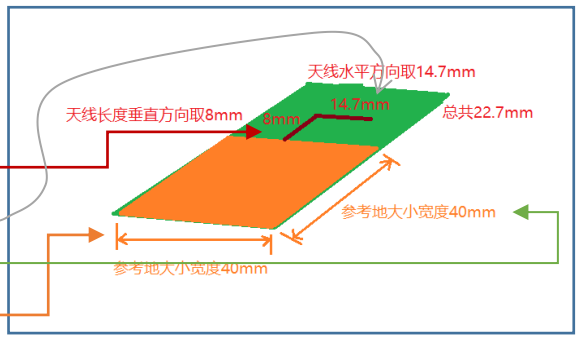
W 变量：　设置的 PCB 天线宽度 1.6mm

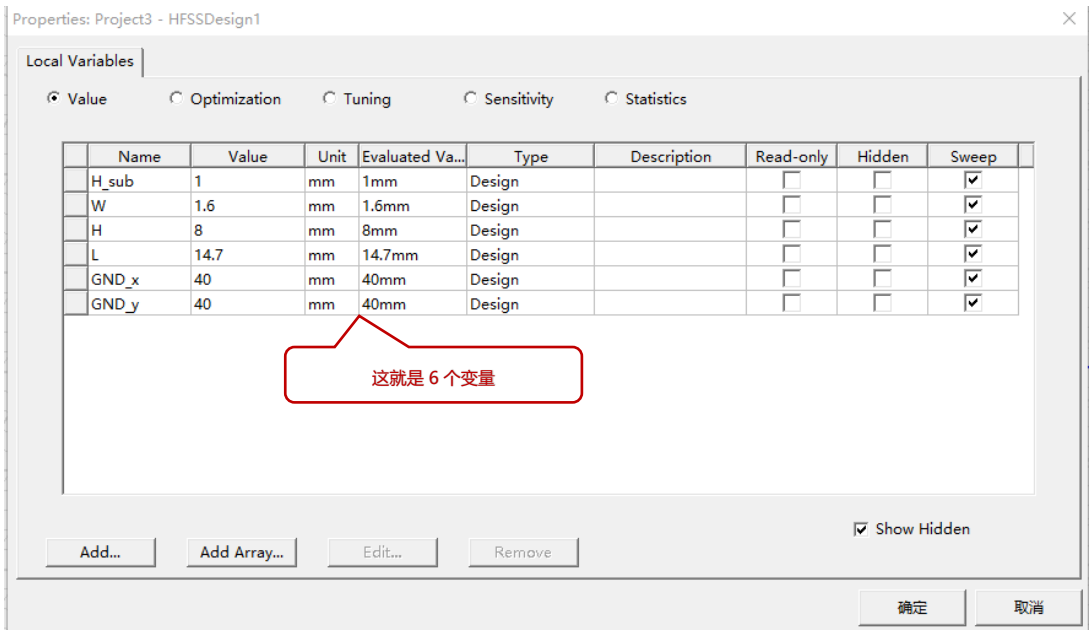
H 变量：　设置 PCB 天线垂直方向长度 8mm

L 变量：　设置 PCB 天线水平方向长度 14.7mm

GND_x 变量：　设置 PCB 参考地长度 40mm

GND_y 变量：　设置 PCB 参考地宽度 40mm

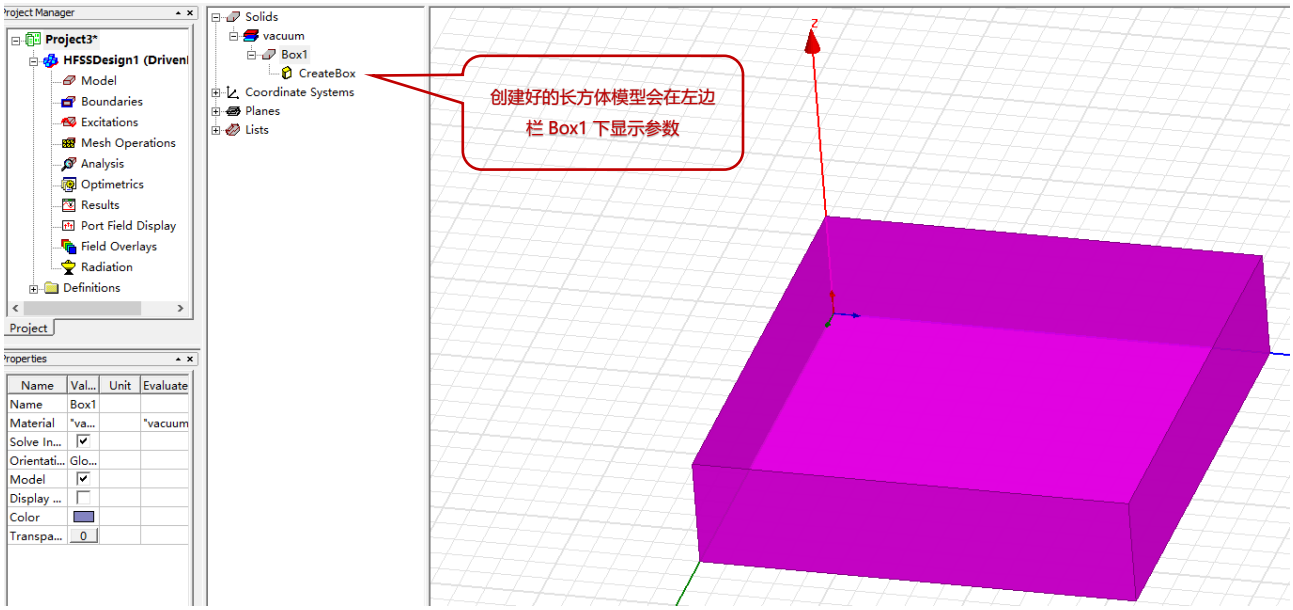


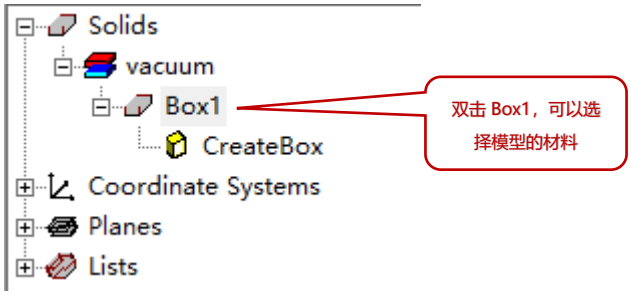


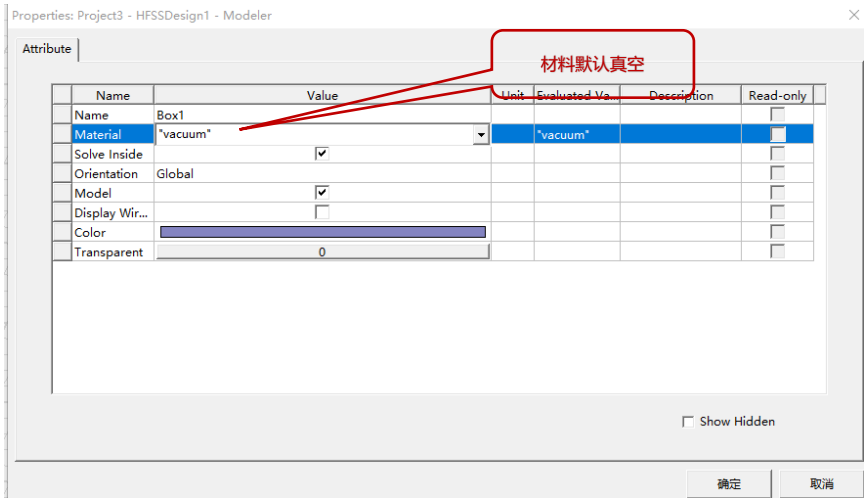
下面建模的时候就使用这 6 个变量来表示模型的长宽高等参数

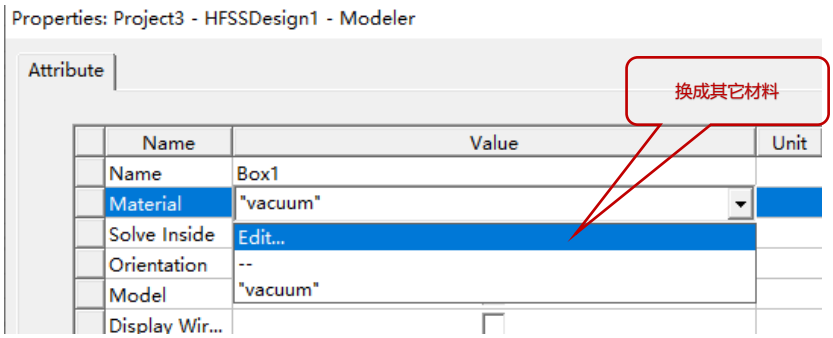
创建天线模型

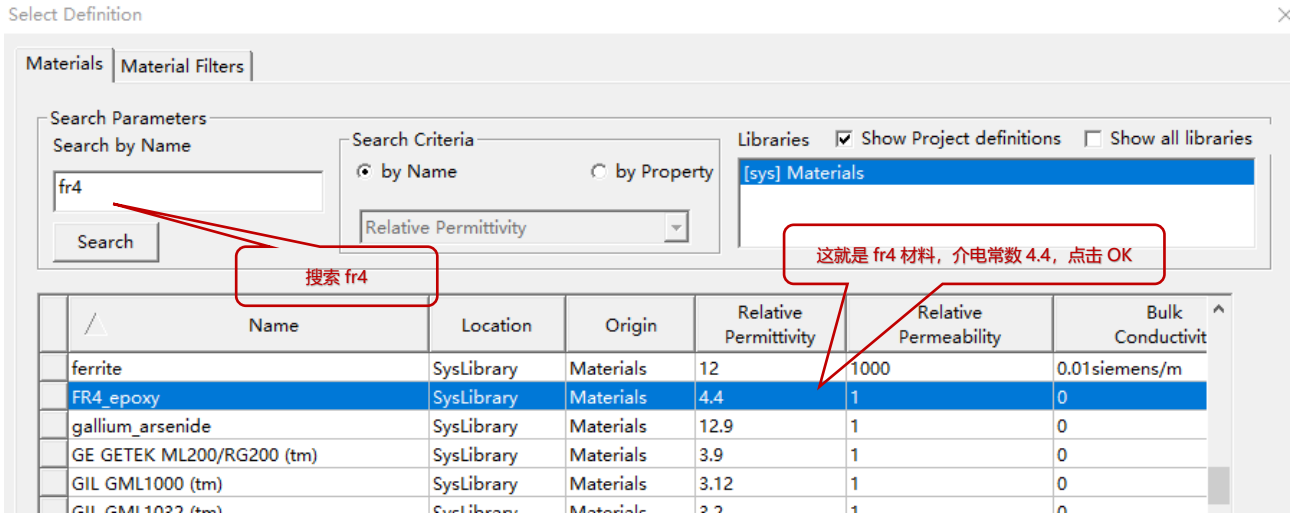
选择右上角长方体模型来当做介质层(PCB 板厚)

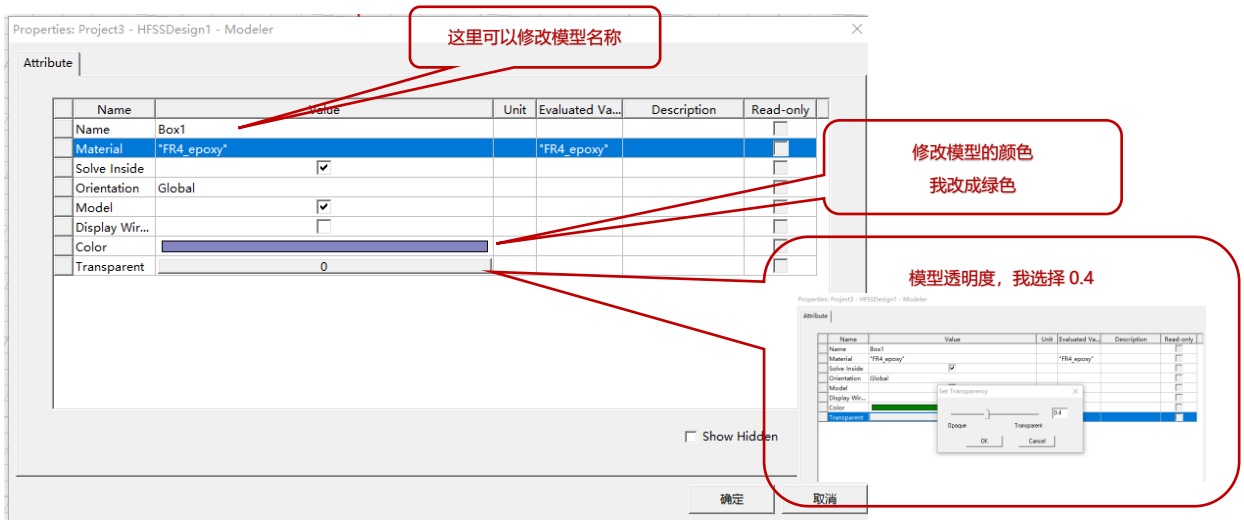

Project Manager
Project3*
HFSSDesign1 (Driven
Model
Boundaries
Excitations
Mesh Operations
Analysis
Optimetrics
Results
Port Field Display
Field Overlays
Radiation
Definitions
Project
Properties
Name Val... Unit Evaluate
Name Box1
Material "va... "vacuum
Solve In...
Orientati... Glo...
Model
Display ...
Color
Transpa... 0
Solids
vacuum
Box1
CreateBox
Coordinate Systems
Planes
Lists
创建好的长方体模型会在左边栏 Box1 下显示参数


Solids
vacuum
Box1
CreateBox
Coordinate Systems
Planes
Lists
双击 Box1，可以选择模型的材料

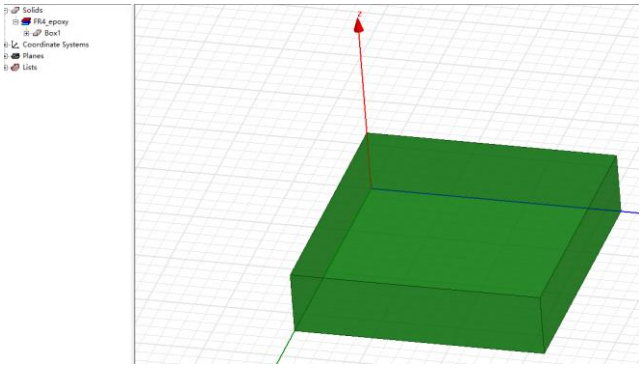

Properties: Project3 - HFSSDesign1 - Modeler
Attribute
材料默认真空
Name Value Unit Evaluated Va... Description Read-only
Name Box1
Material "vacuum" "vacuum"
Solve Inside
Orientation Global
Model
Display Wir...
Color
Transparent 0
Show Hidden
确定
取消


Properties: Project3 - HFSSDesign1 - Modeler
Attribute
换成其它材料
Name Value Unit
Name Box1
Material "vacuum"
Solve Inside
Orientation
Model
Display Wir...
Edit...
--
"vacuum"

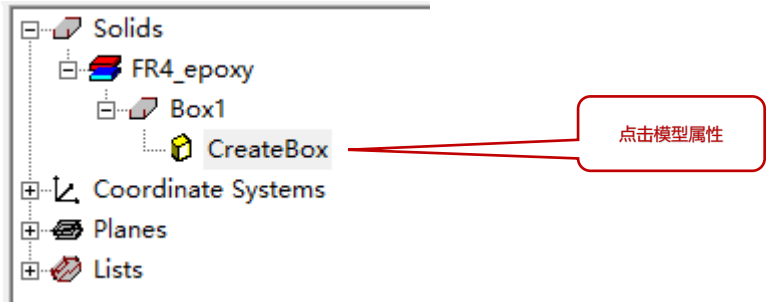


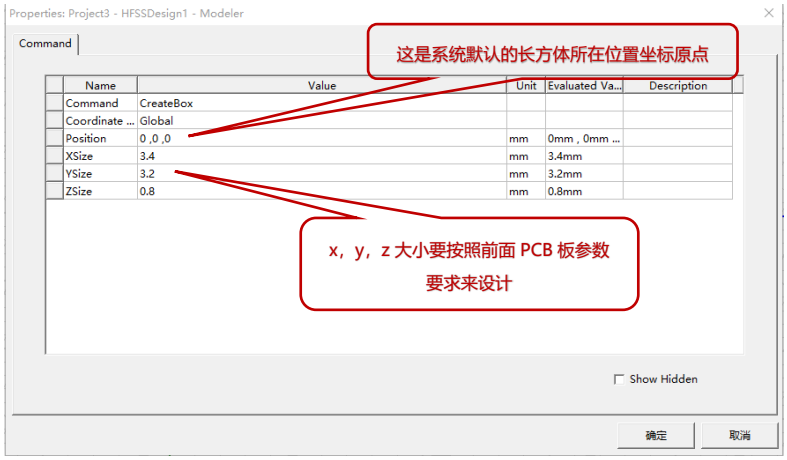


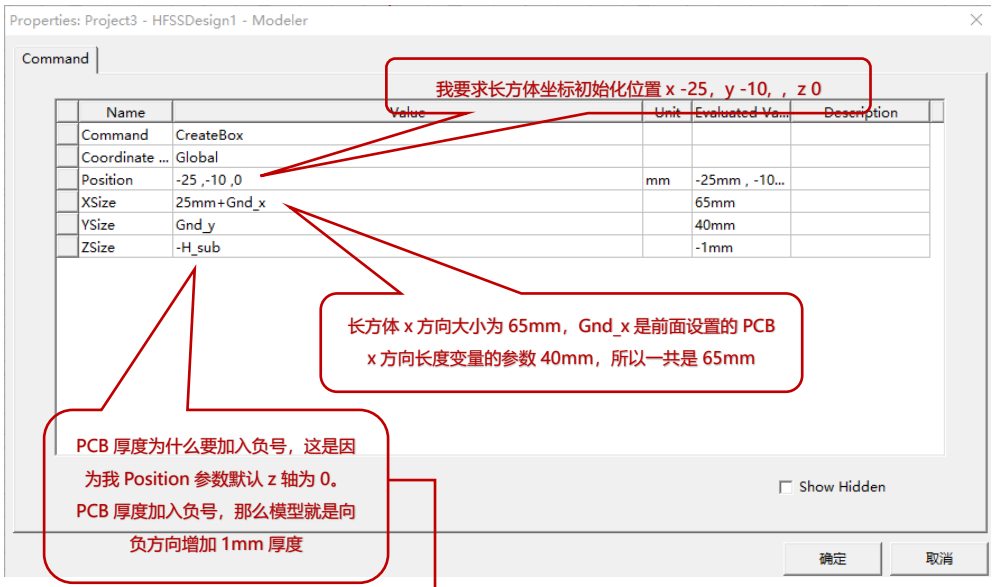
最后点击确定

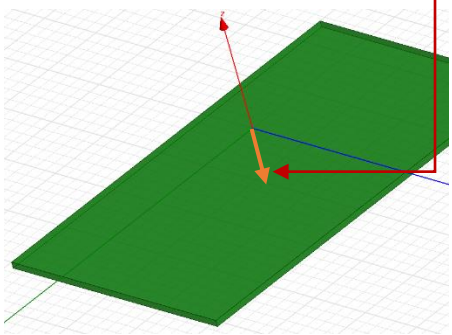


设置模型属性



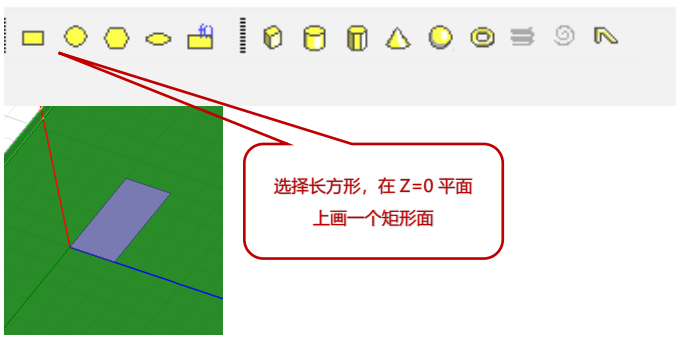


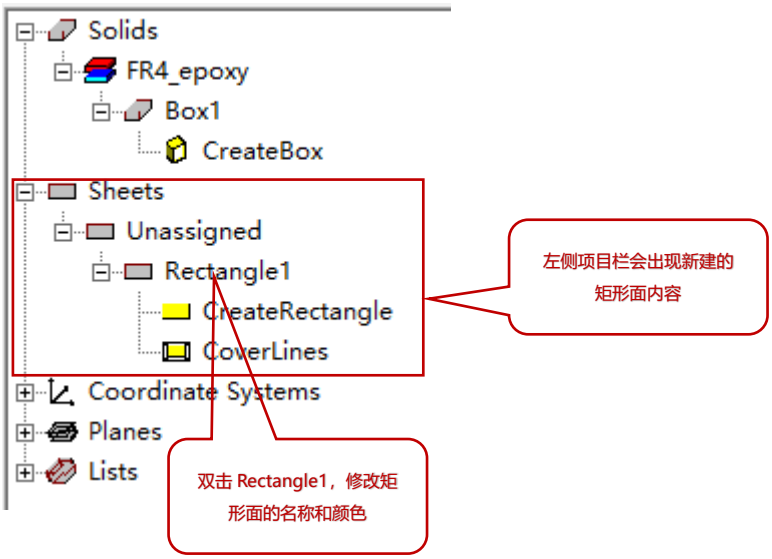


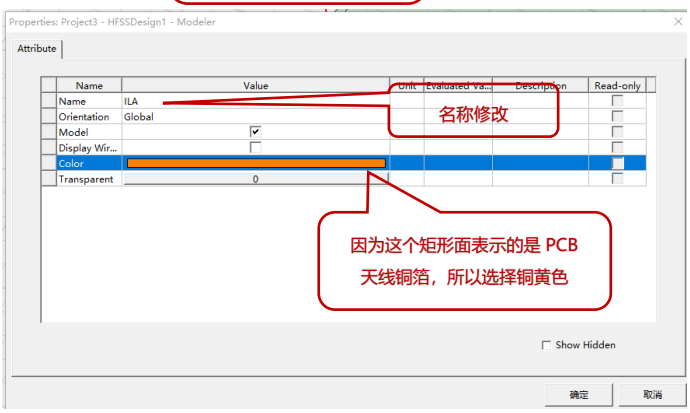


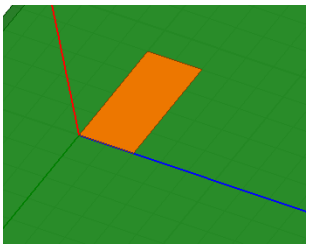
这样 PCB 板材就建立好了

在 PCB 板材上创建天线走线部分




Solids
FR4_epoxy
Box1
CreateBox
Sheets
Unassigned
Rectangle1
CreateRectangle
CoverLines
Coordinate Systems
Planes
Lists
左侧项目栏会出现新建的矩形面内容
双击 Rectangle1，修改矩形面的名称和颜色


Properties: Project3 - HFSSDesign1 - Modeler
Attribute
Name
Value
Unit
Evaluated Va...
Description
Read-only
Name
ILA
Orientation
Global
Model
Display Wir...
Color
Transparent
0
名称修改
因为这个矩形面表示的是 PCB 天线铜箔，所以选择铜黄色
Show Hidden
确定
取消

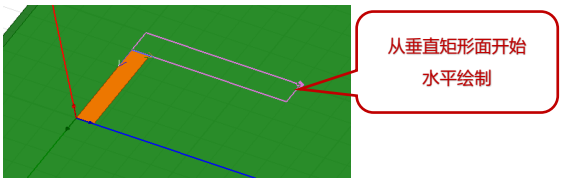


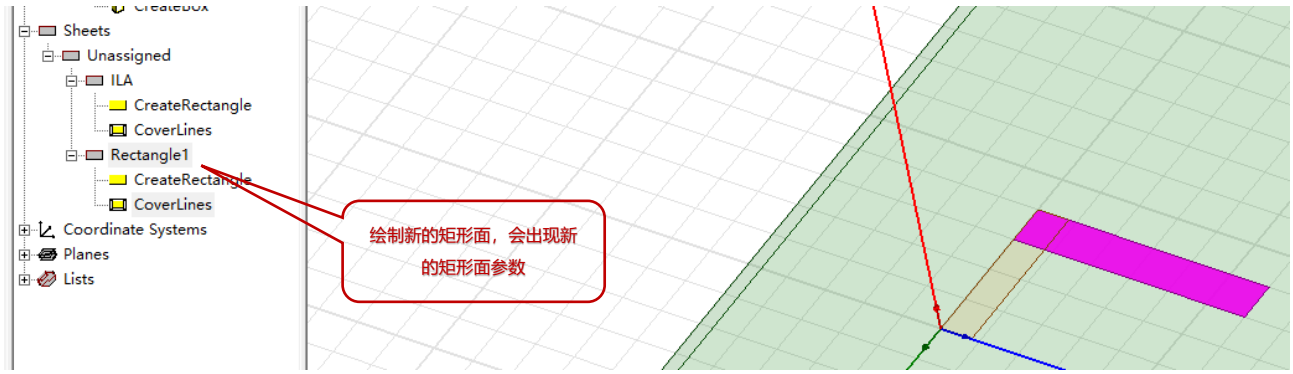

Sheets
Unassigned
ILA
CreateRectangle
CoverLines
双击矩形属性，设置矩形面长宽高，其实设置方式和前面设置 PCB 板材一样

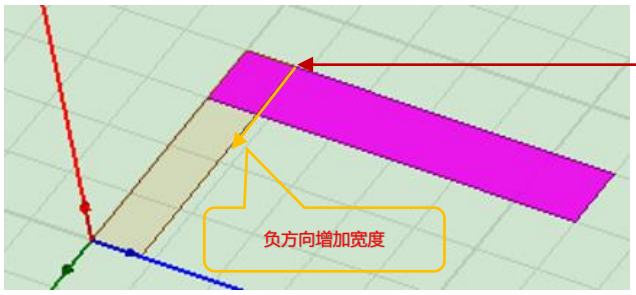

Properties: Project3 - HFSSDesign1 - Modeler
Command
因为是创建的 PCB 天线铜箔的垂直部分，所以只需要设置垂直方向线长 8mm。
这里为-8mm 是因为要求矩形向负方向画 8mm 长度
Name
Value
Unit
Evaluated Va...
Description
Command
CreateRectangle
Coordinate ...
Global
Position
0 ,0 ,0
mm
0mm , 0mm ...
Axis
Z
XSize
-H
-8mm
YSize
W
1.6mm
导线宽度 1.6mm

绘制水平面天线

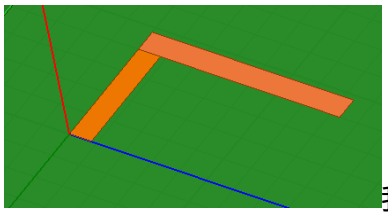

和前面一样，选择矩形


从垂直矩形面开始水平绘制

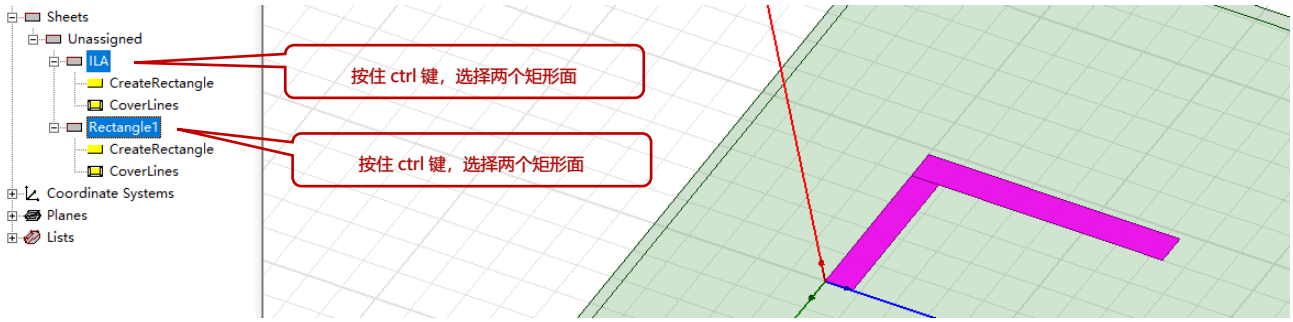


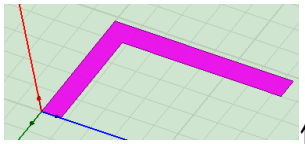


合并矩形面



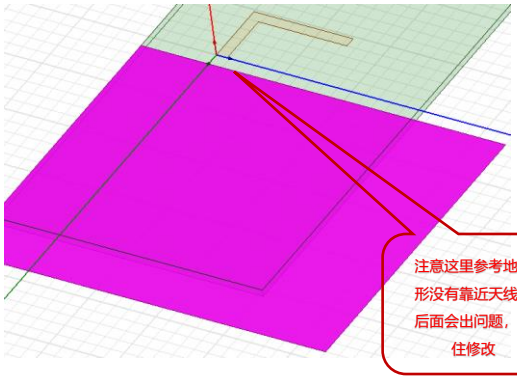
我发现天线的垂直和水平矩形面不是一个整体，有黑线分割。

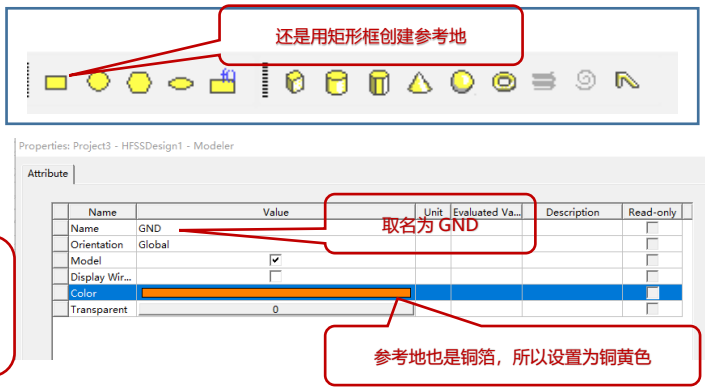


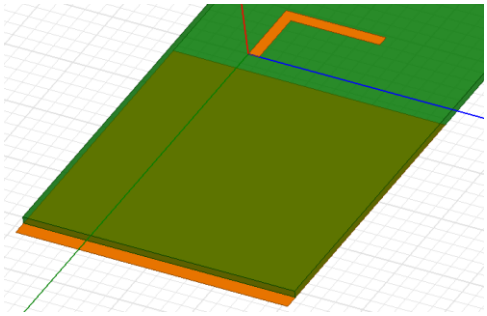


你看 和上面相比，两个矩形框没有黑色分割线了。

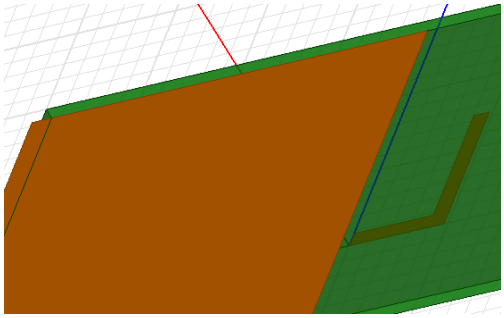
创建参考地



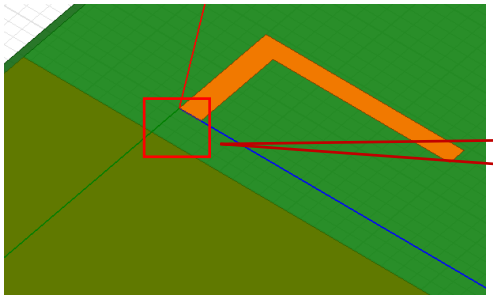


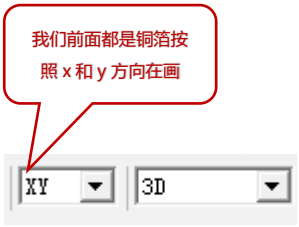


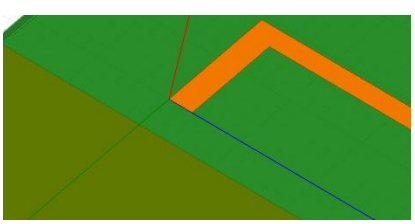
天线在介质(PCB 基板)上表面

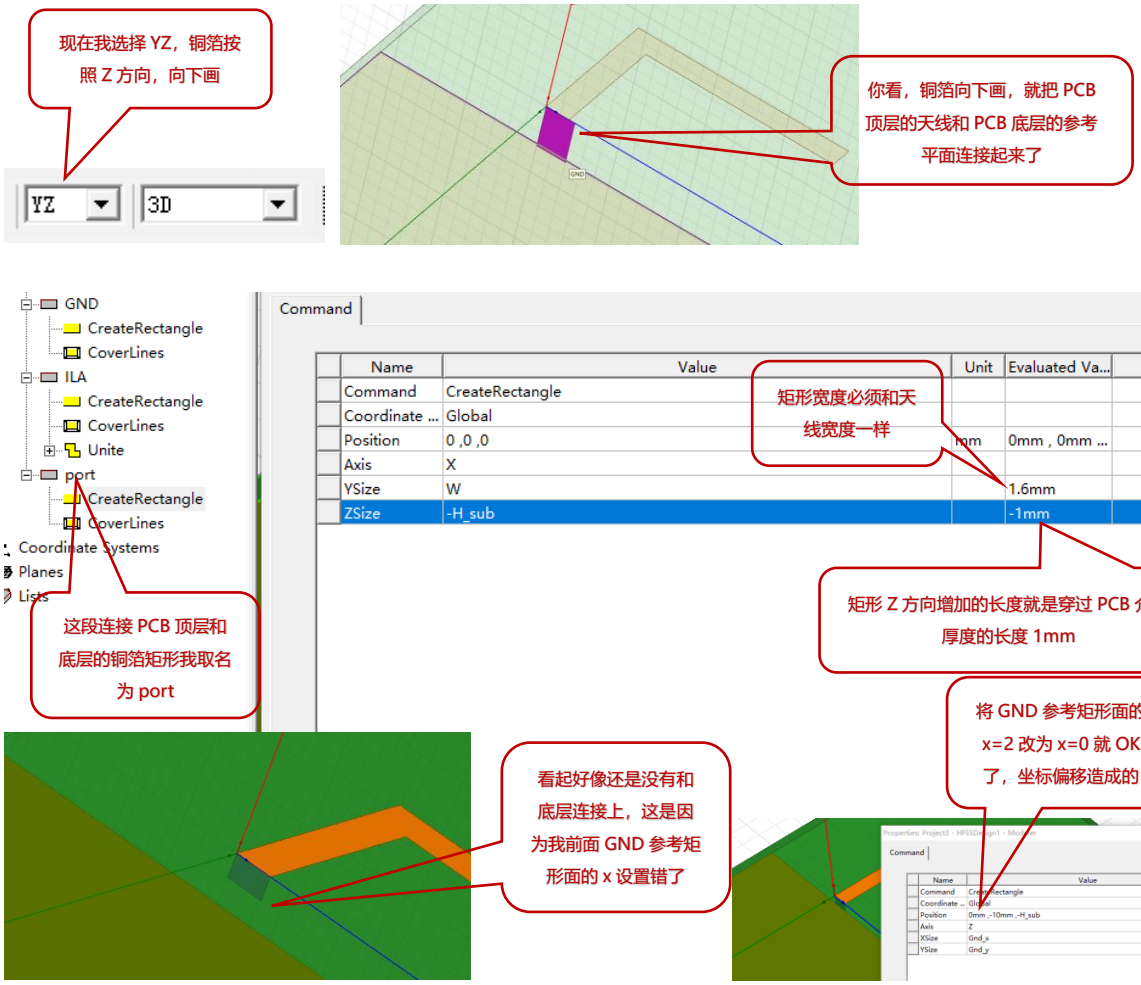


参考地在(PCB 基板)下表面



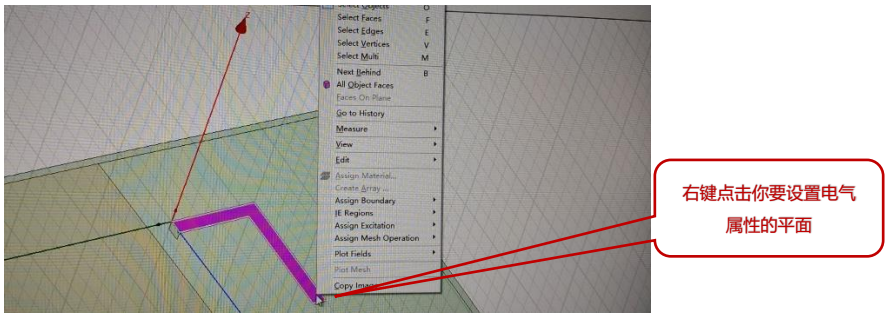


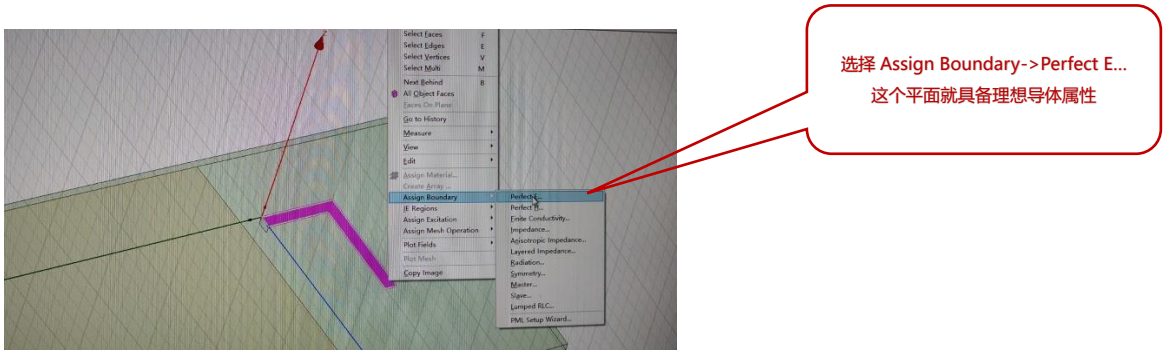


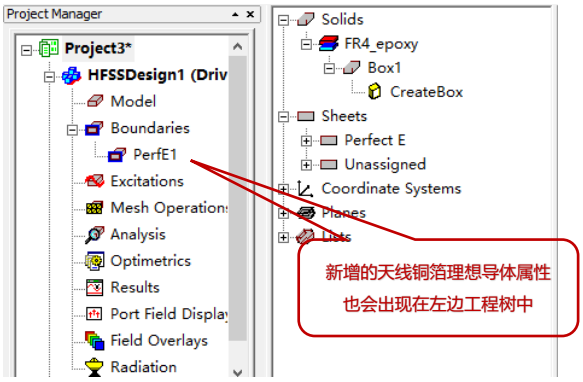


给模型形状增加电气特性

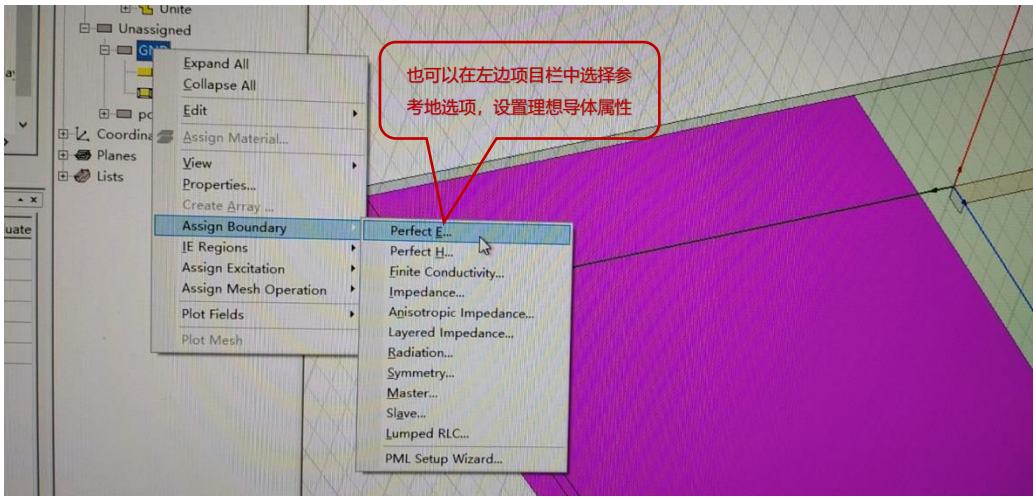
上面几节都只是单纯的画模型形状，这些形状并没有产生电气属性，下面给这些形状加电气属性

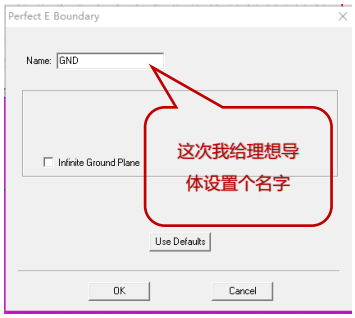


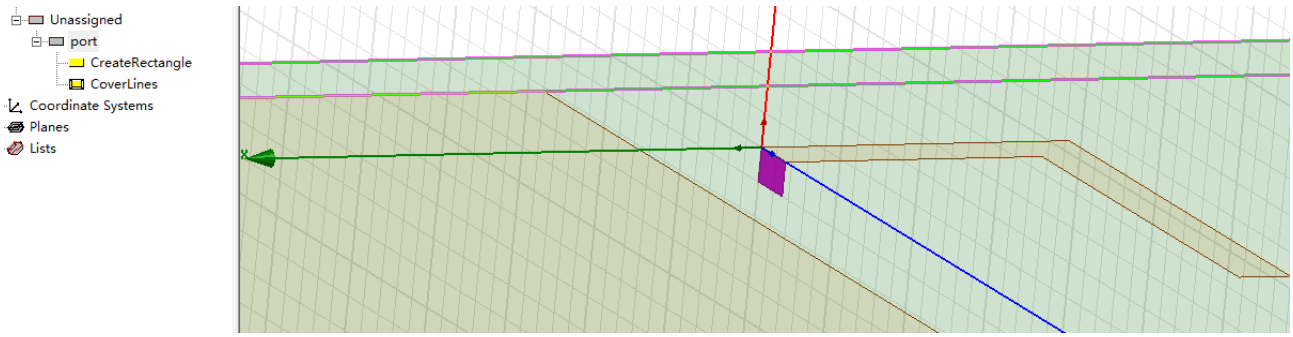




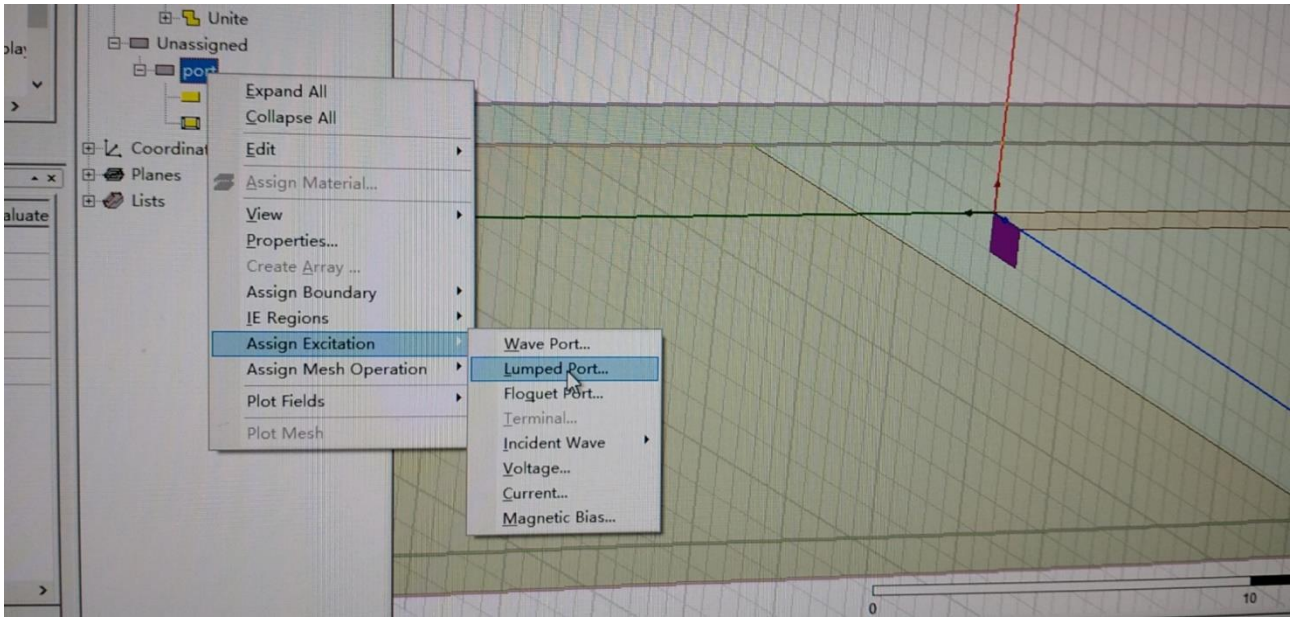
设置参考 GND 也为理想导体属性

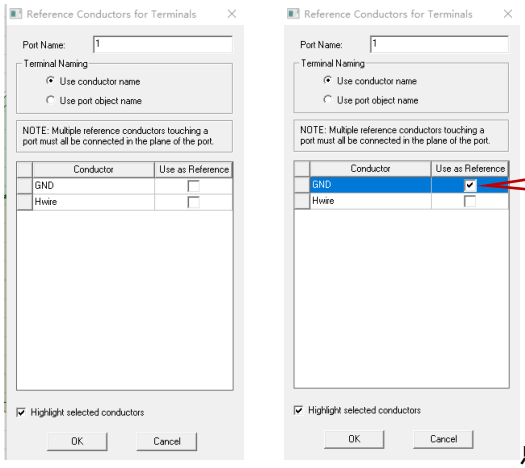






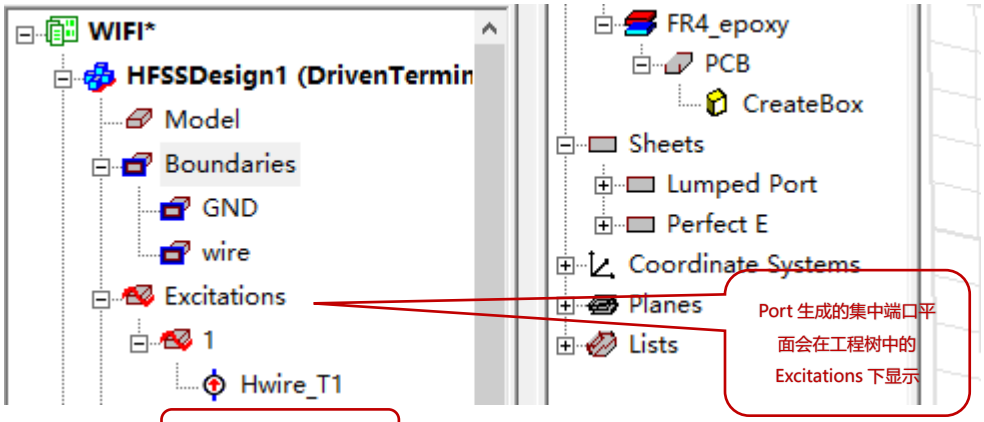
给 Port 连接顶层和底层的矩形图形也要设置电气特性，但是这个特性是集中参数端口 Lumped Port



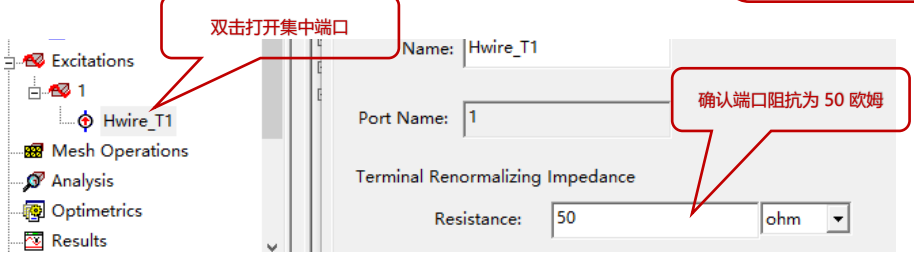


选择 port 为 GND，就表示 port 这块矩形区域也是参考地的平面

点击 OK



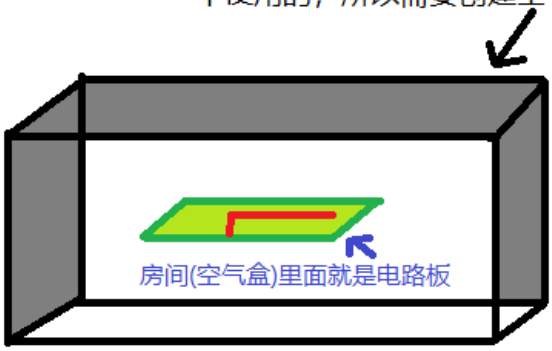
Port 生成的集中端口平面会在工程树中的 Excitations 下显示



确认端口阻抗为 50 欧姆

创建空气盒

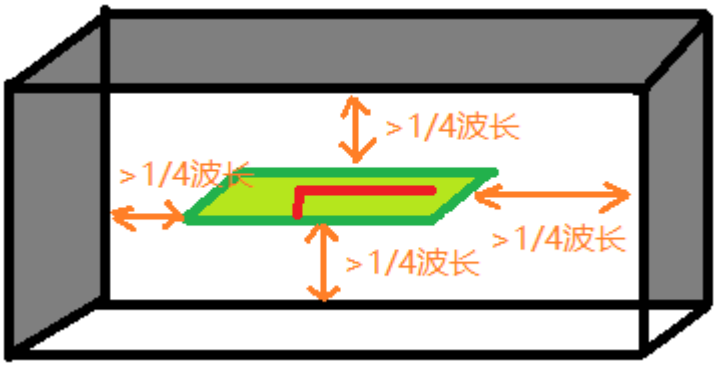
空气盒其实就是房间，我们PCB都是暴露在空气中使用的，所以需要创建空气盒假设为房间



所以我们空气盒设置的介电常数就是空气的介电常数1.0左右

这就是模拟真实环境仿真

空气盒四边的表面和天线的距离要大于 1/4 自由空间波长



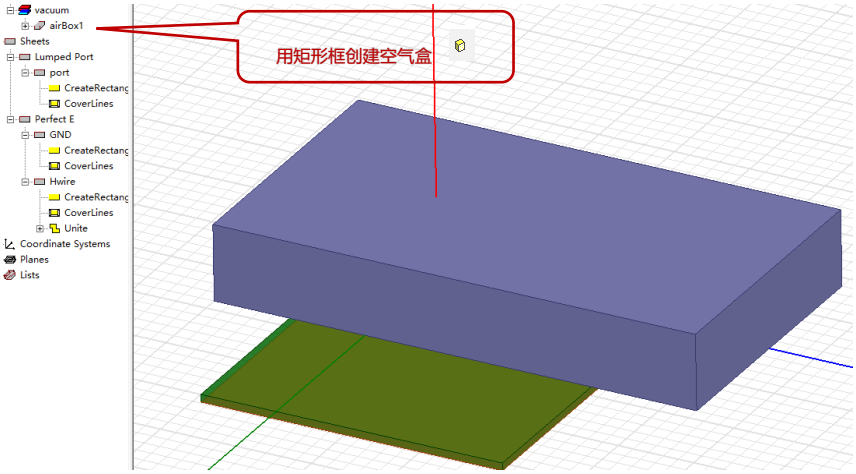
当前天线工作频率是 2.45GHZ，对应的 1/4 波长是 30.6mm

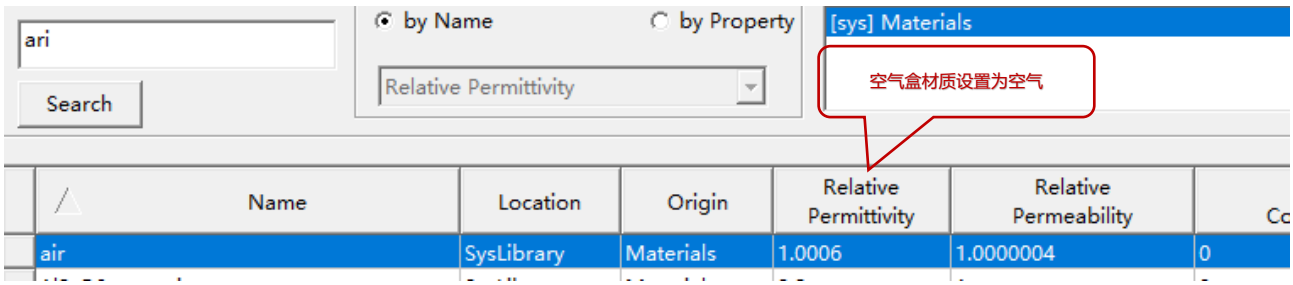
$$\text{那么空气部分的自由空间波长} = \frac{\text{光速}}{\text{工作频率}} = \frac{3 \times 10^8}{2.45 \times 10^9} = 0.122\text{米} = 122\text{mm}$$

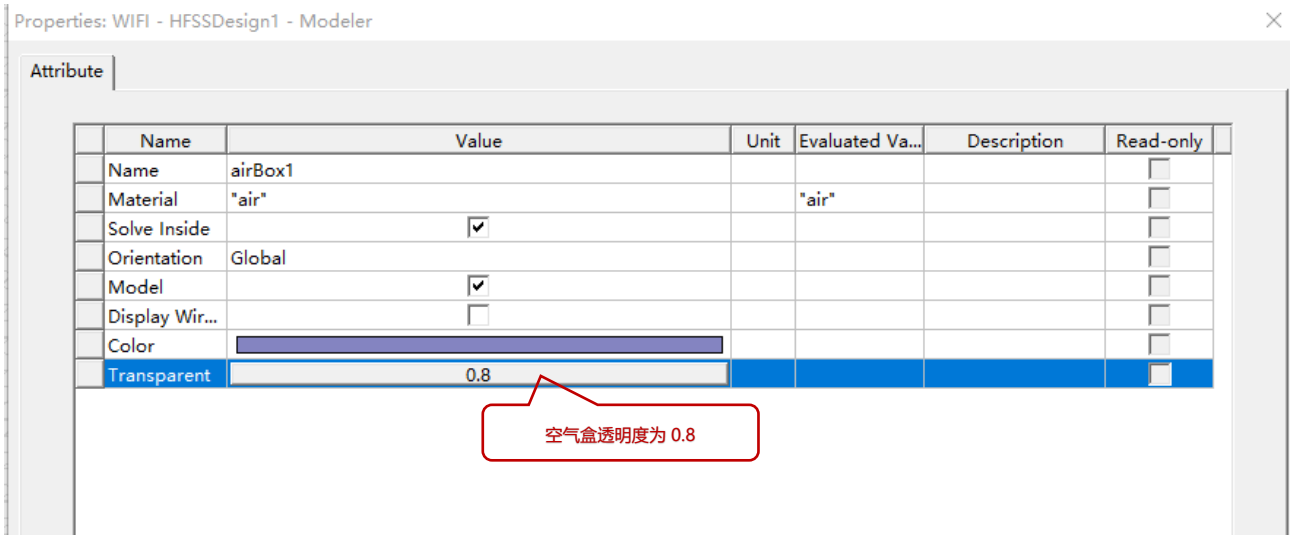
我们只能取1/4波长 = 122mm / 4 = 30.6mm

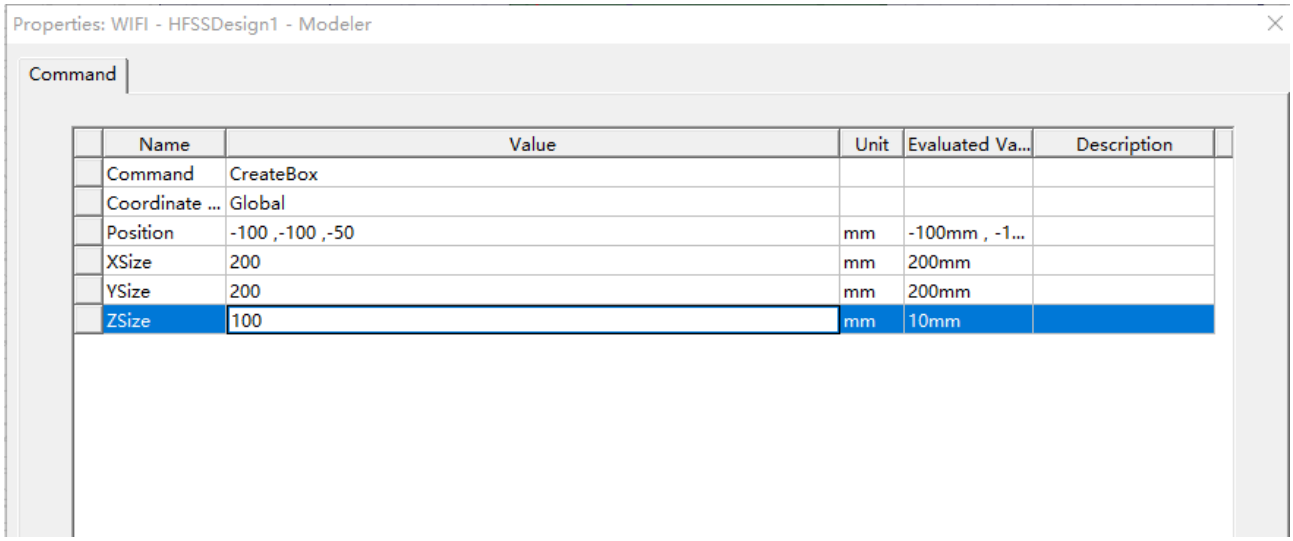
所以我们空气表面和天线的距离要大于 30.6mm 。

我们创建个 200mm x 200mm x 100mm 的矩形空气盒，这个空气盒每一面与 PCB 天线都 >30.6mm

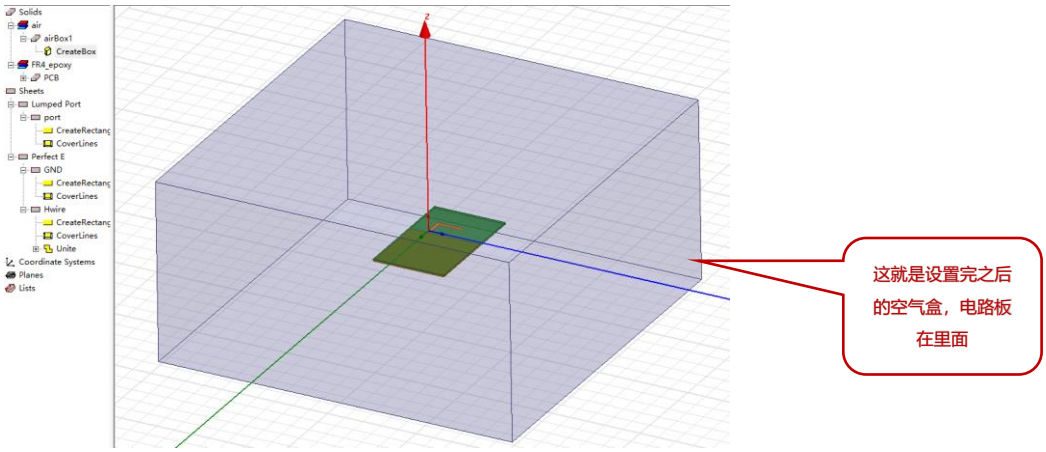



ari
Search
by Name
by Property
Relative Permittivity
[sys] Materials
空气盒材质设置为空气
Name
Location
Origin
Relative Permittivity
Relative Permeability
air
SysLibrary
Materials
1.0006
1.0000004

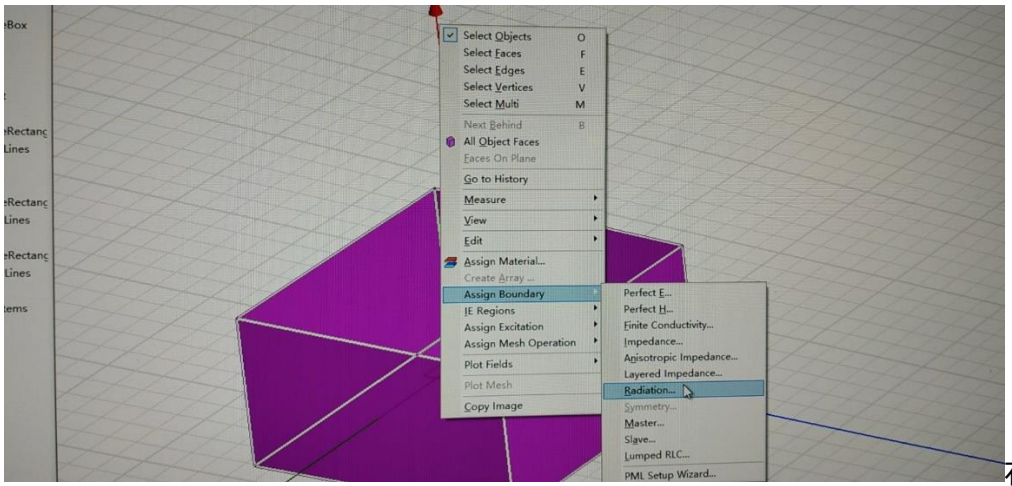

Properties: WIFI - HFSSDesign1 - Modeler
Attribute
Name
Value
Unit
Evaluated Va...
Description
Read-only
Name
airBox1
Material
"air"
"air"
Solve Inside
Orientation
Global
Model
Display Wir...
Color
Transparent
0.8
空气盒透明度为 0.8


Properties: WIFI - HFSSDesign1 - Modeler
Command
Name
Value
Unit
Evaluated Va...
Description
Command
CreateBox
Coordinate ...
Global
Position
-100 ,-100 ,-50
mm
-100mm , -1...
XSize
200
mm
200mm
YSize
200
mm
200mm
ZSize
100
mm
10mm

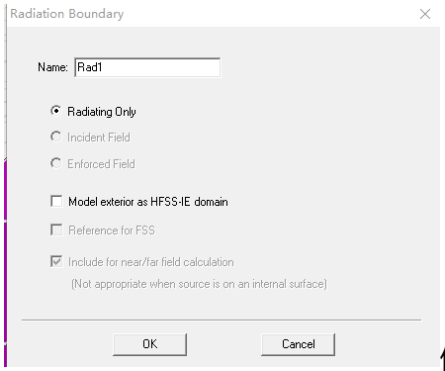
空气盒坐标及尺寸设置


这就是设置完之后的空气盒，电路板在里面

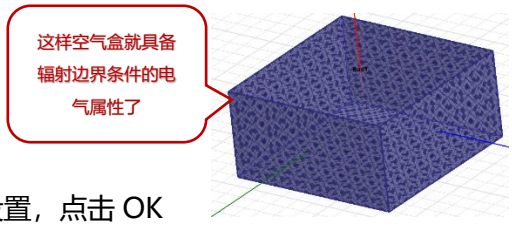
给空气盒增加电气属性



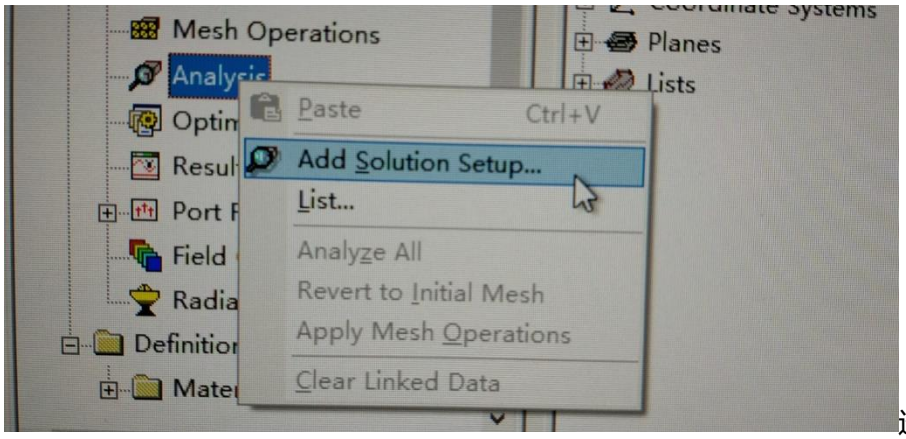
右键空气盒本身



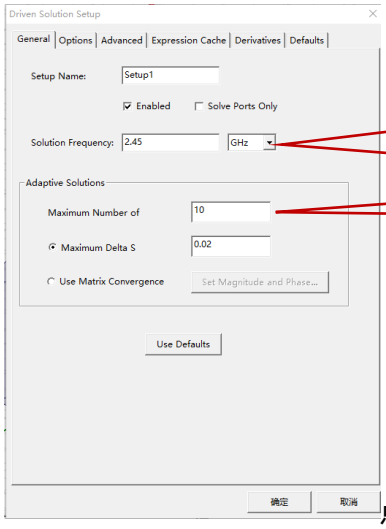
保留默认设置，点击 OK



设置求解分析参数

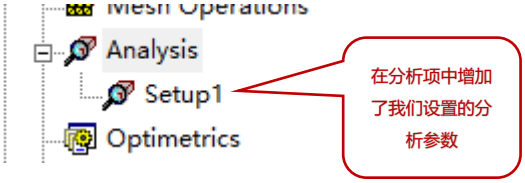


选择分析



因为天线是 2.45Ghz 频段的 wifi 天线，所以这里选择 2.45Ghz

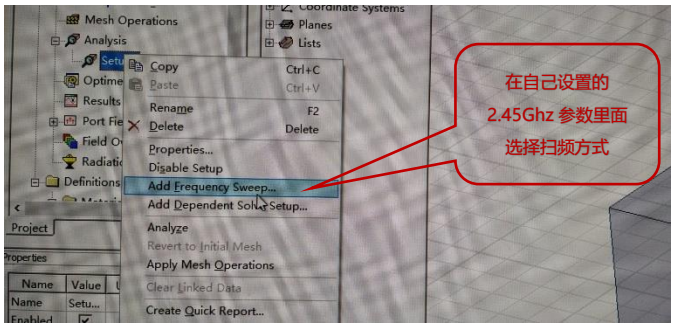
最大分析次数我们设置为 10



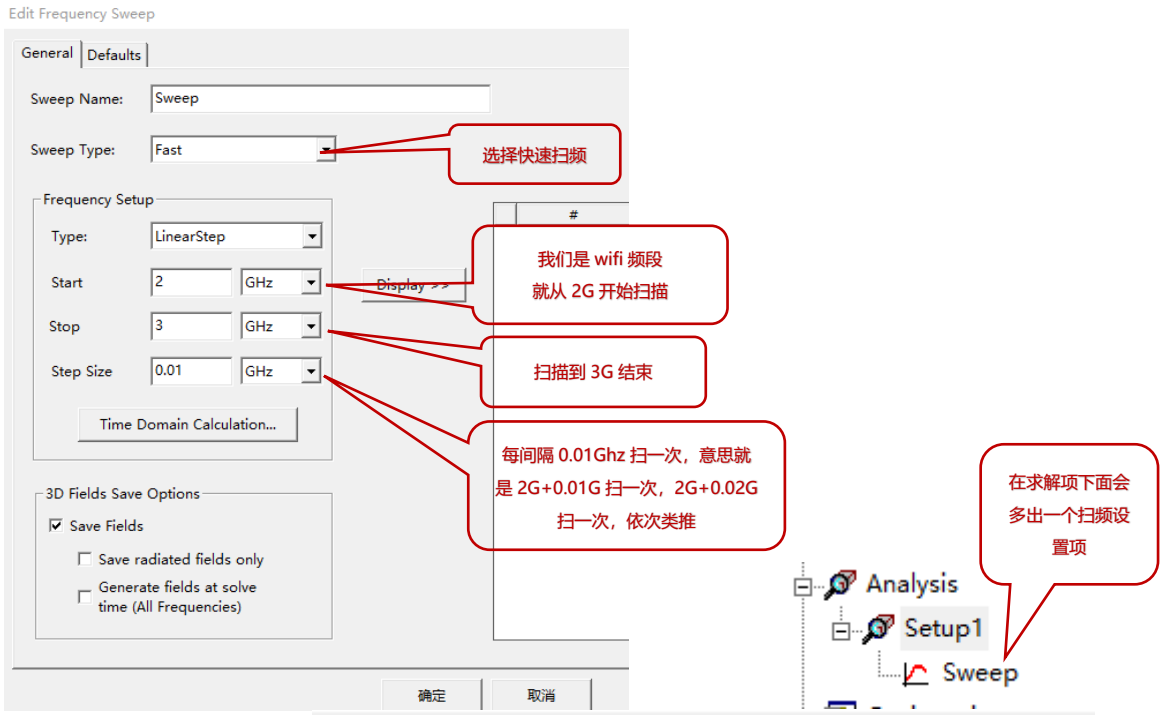
在分析项中增加了我们设置的分析参数

点击确定

添加扫频设置项



在自己设置的 2.45Ghz 参数里面选择扫频方式



选择快速扫频

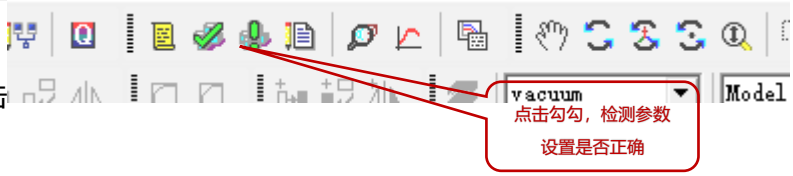
我们是 wifi 频段就从 2G 开始扫描

扫描到 3G 结束

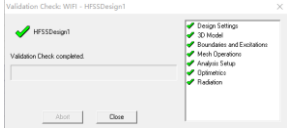
每间隔 0.01Ghz 扫一次，意思就是 2G+0.01G 扫一次，2G+0.02G 扫一次，依次类推

在求解项下面会多出一个扫频设置项

仿真参数设置完毕，点击

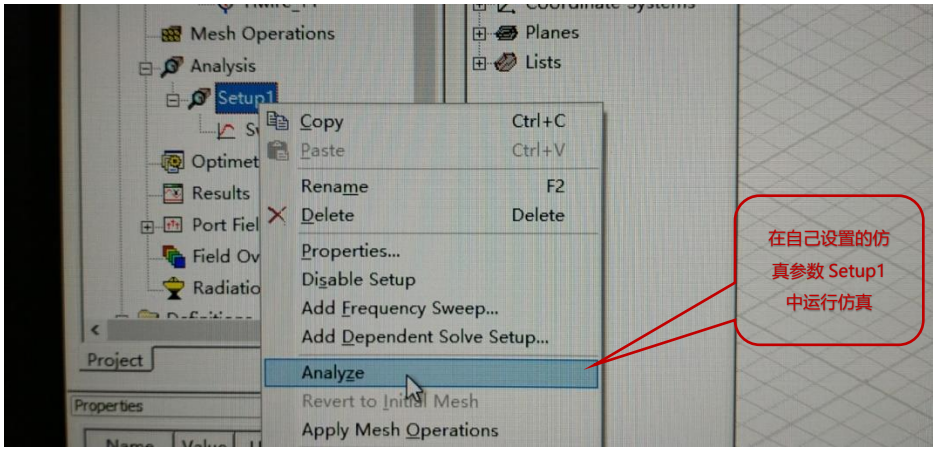


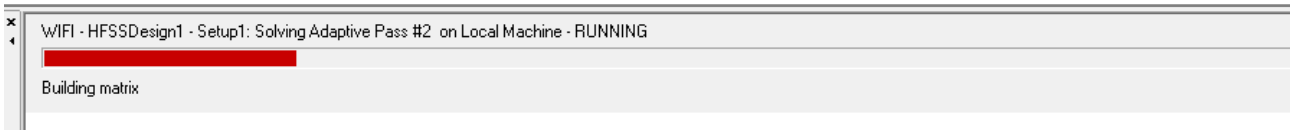
点击勾勾，检测参数设置是否正确



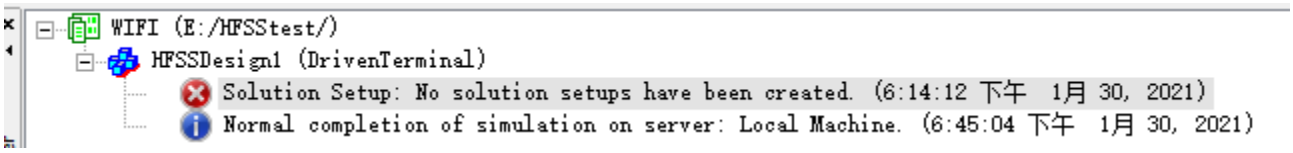
全部显示绿勾，证明所有模型和仿真参数设置正确。下面开始仿真。

开始仿真



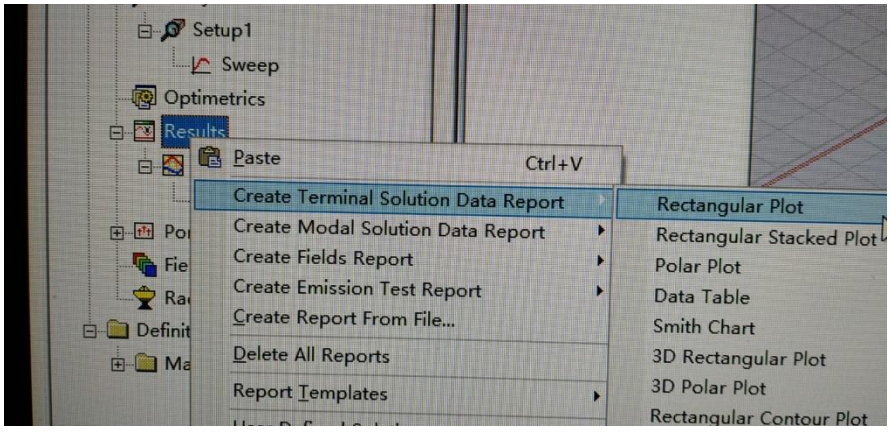


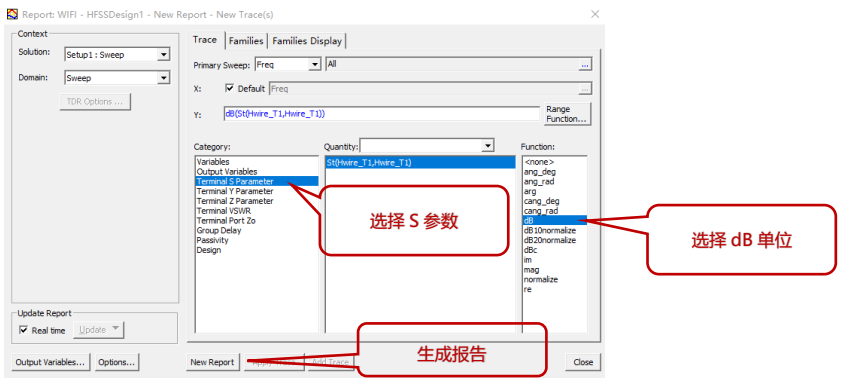
仿真运行时会显示进度条



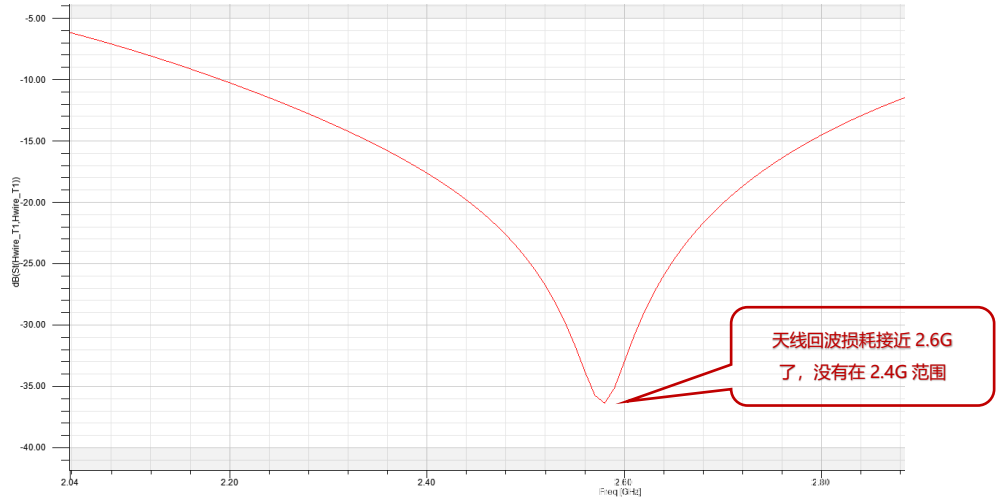
仿真完成

获取仿真结果的报告



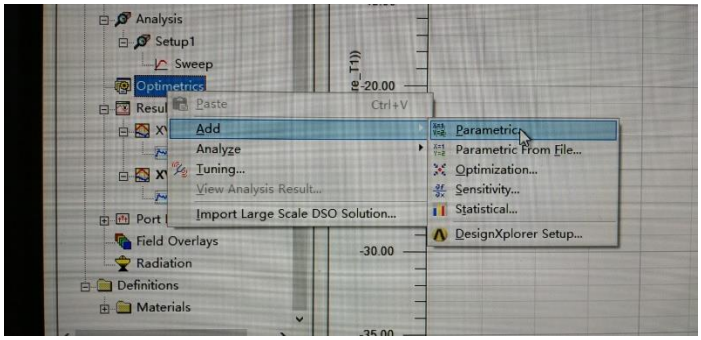


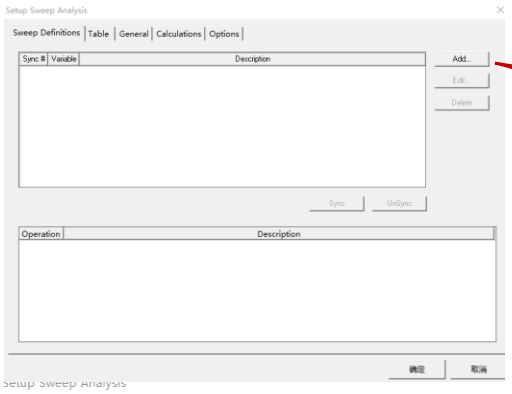
回波损耗分析



回波损耗在高于谐振频率 2.4Ghz 的地方，证明天线的长度短了需要增加天线长度。天线越长对应的谐振频率约低。

优化天线功能：用参数扫描，，扫描出 N 多条回波损耗曲线，方便选择符合谐振频率的回波损耗曲线





添加要修改的变量参数

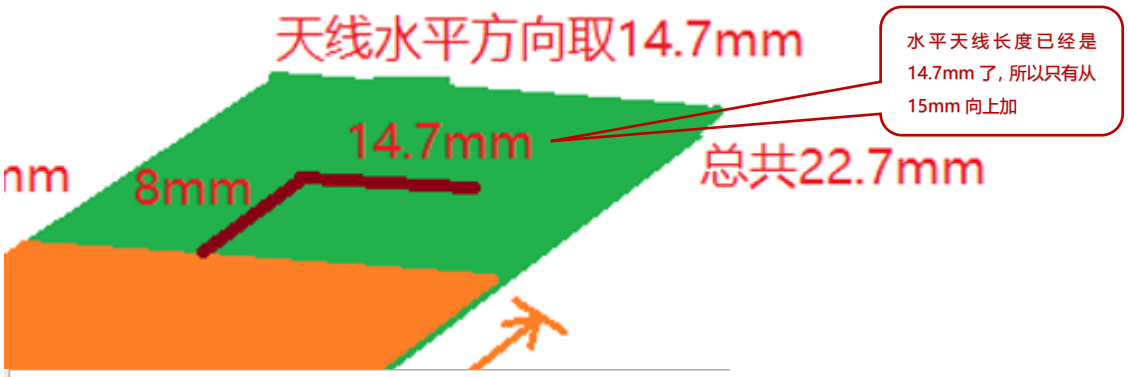
这就是前面定义变量的好处，我想修改什么变量这里就可以修改，选择水平天线长度

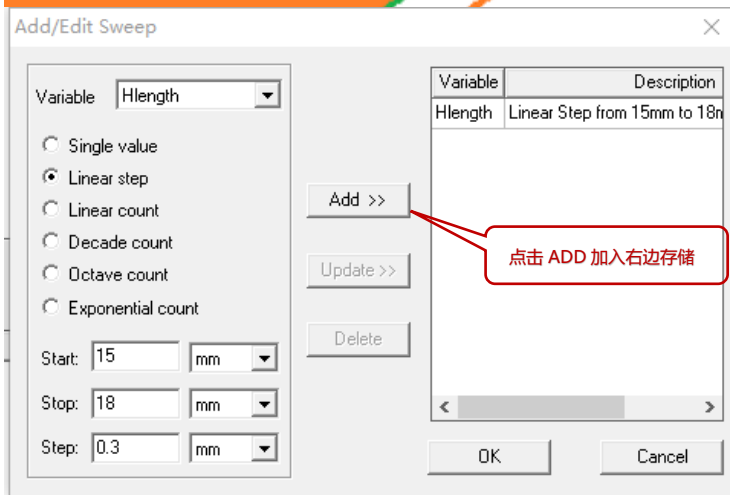
选择默认的线性扫描

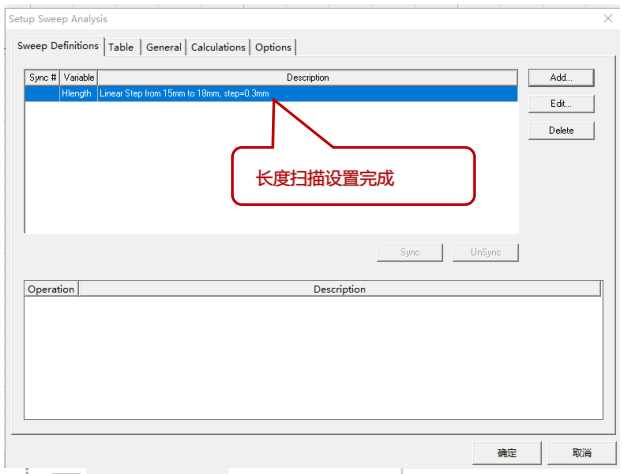
从 15mm 天线长度开始扫，一直扫到 18mm 天线长度

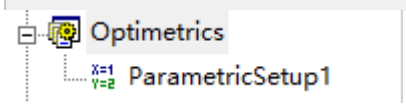
每增加 0.3mm 扫描一次

为什么天线长度要从 15mm 开始增加，这是因为我修改的是水平天线长度

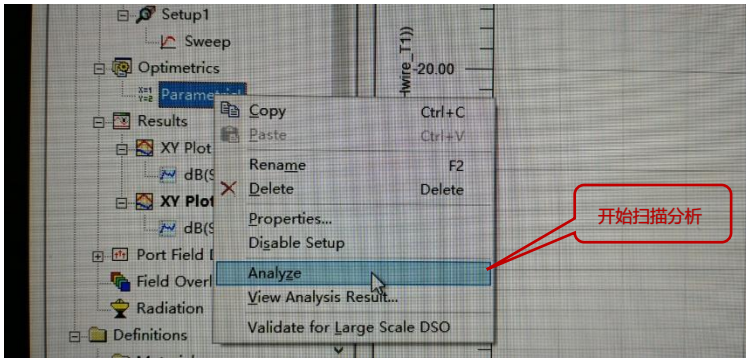






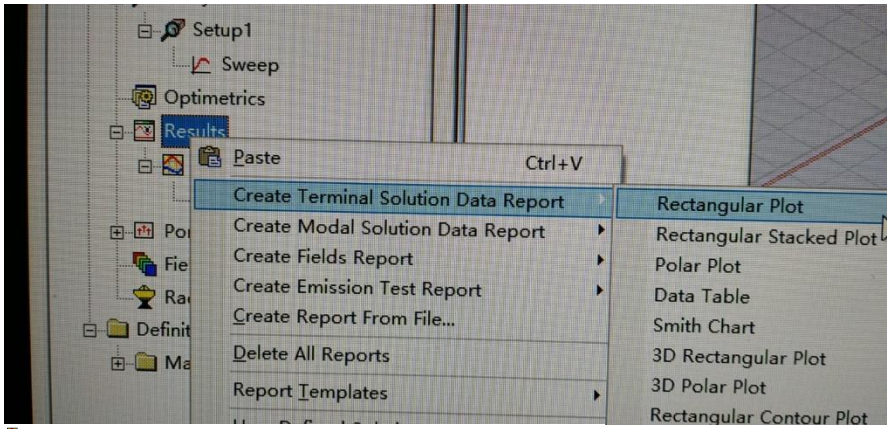


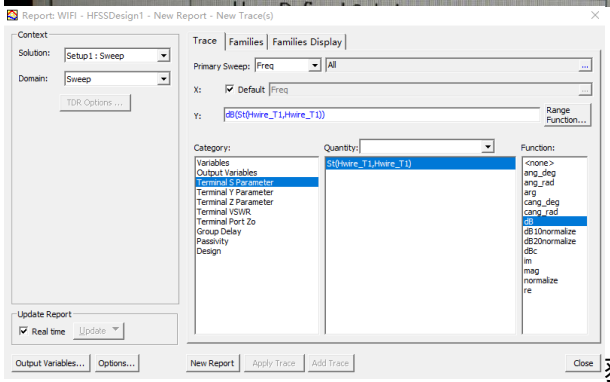
设置了天线长度变量的扫描，这里会出现参数项



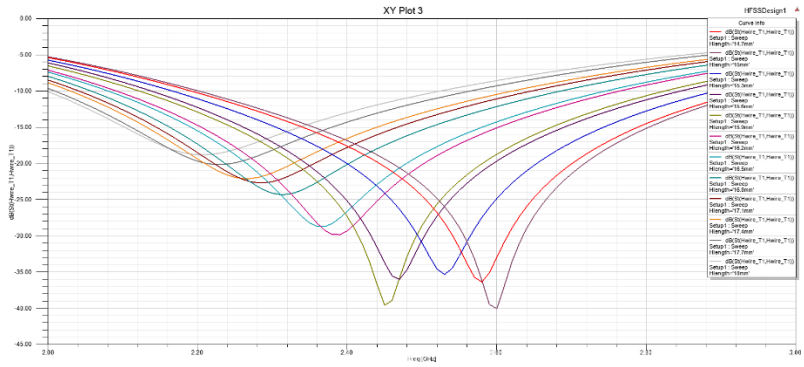
这将是个漫长的仿真过程。

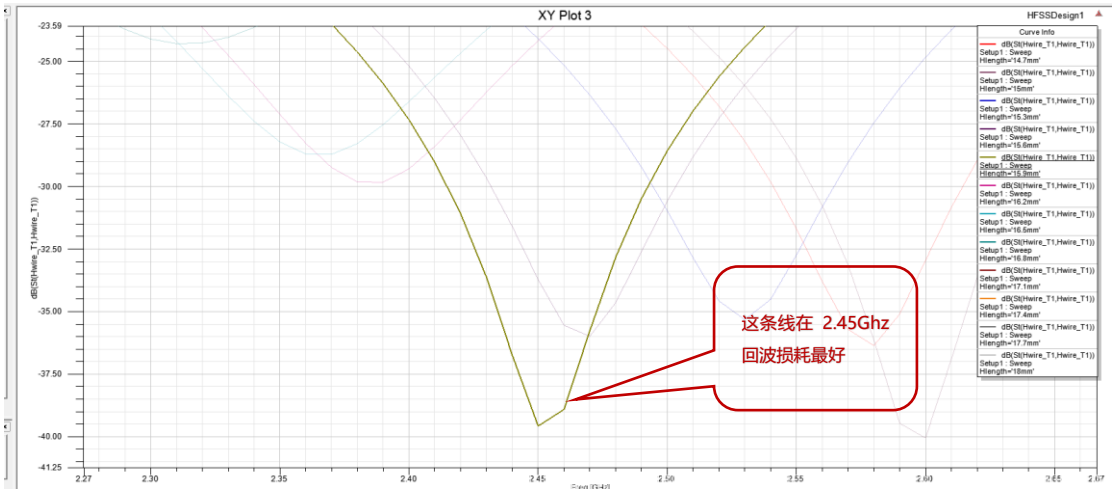
获取扫描长度的报告



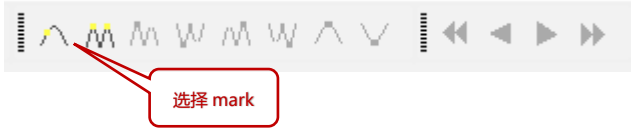


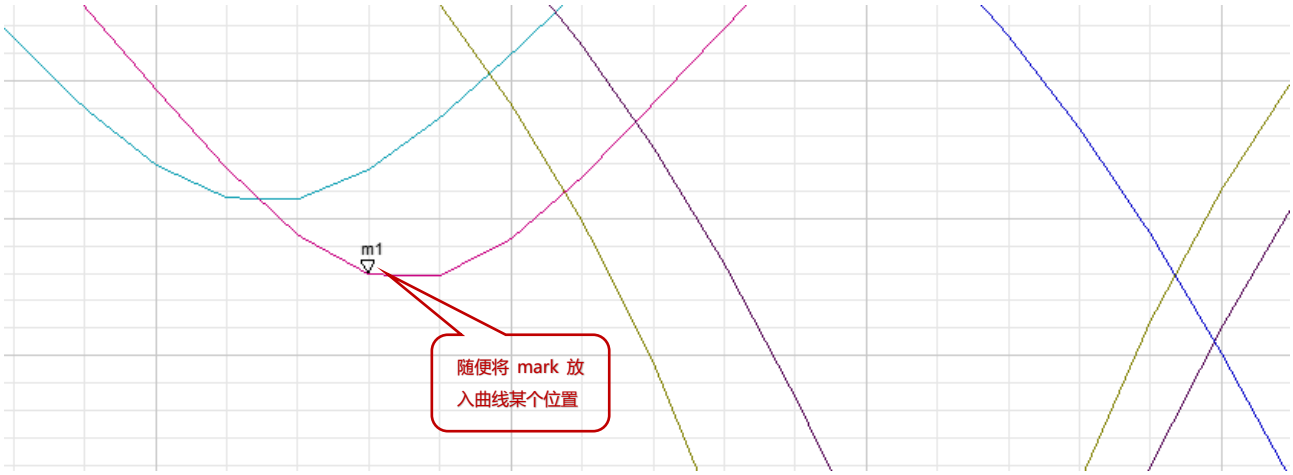
获取报告方式和前面一样。

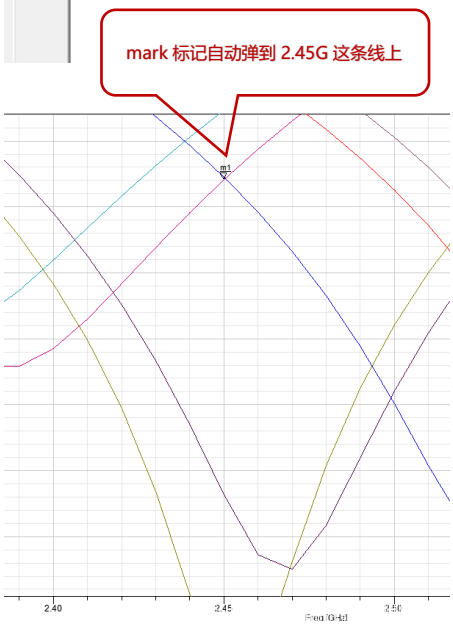




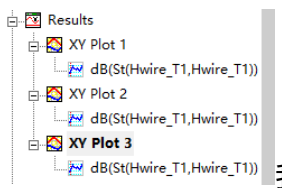
如果你找不到 2.45Ghz 位置，你可以先加一个 mark 标记





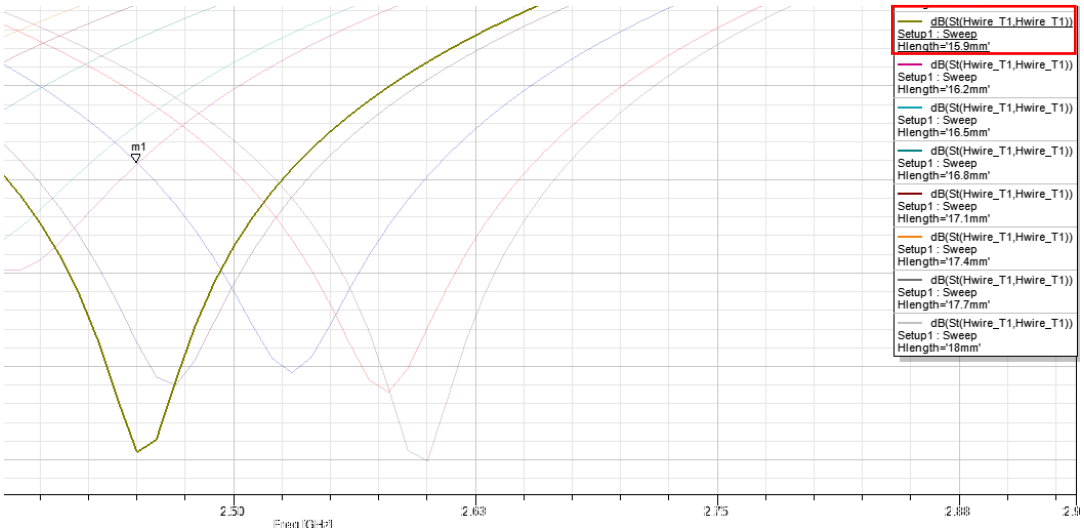


我们从多条曲线也能看出来，随着频率增加天线长度在变短。

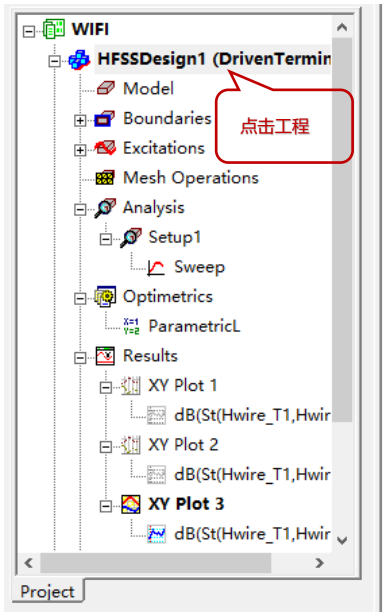


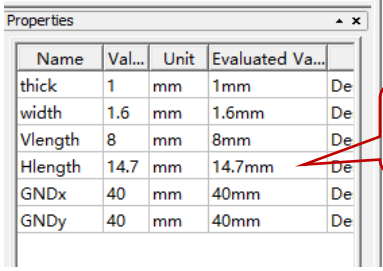
我仿真了三次就有 3 个分析报告，可以查看历史分析记录。

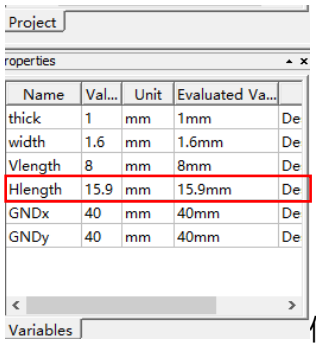
我们现在知道了，在 15.9mm 水平天线的长度下，2.45G 回波损耗最优



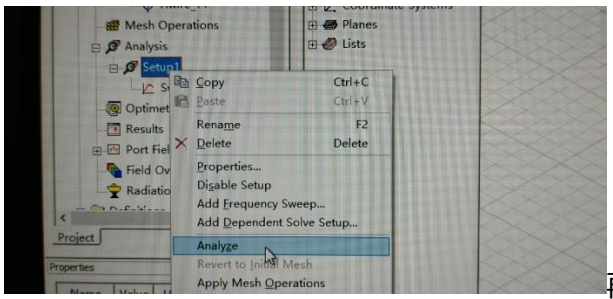
我如何将 15.9mm 这个参数回填进模型进行单独仿真呢?



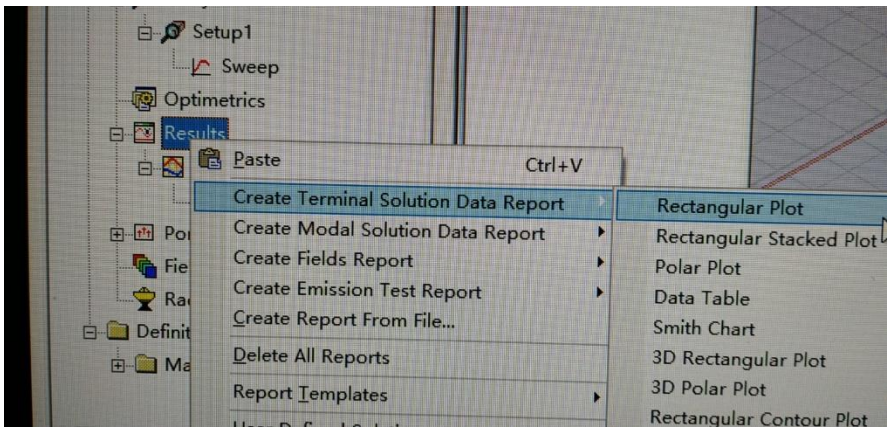




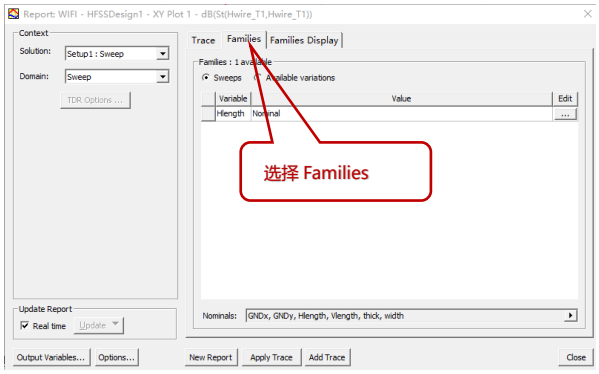
修改成功，保存开始仿真

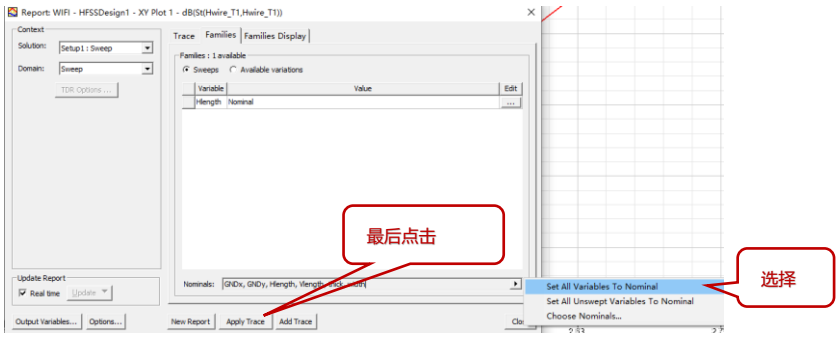


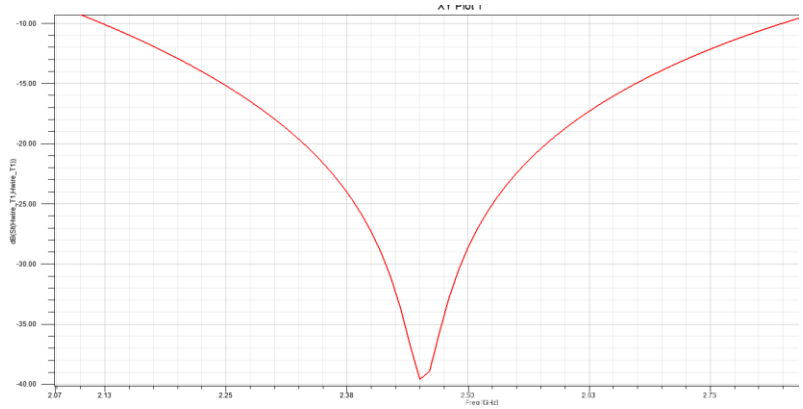
再次运行单个曲线仿真



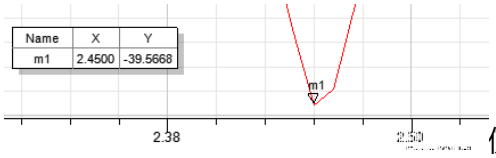
生成报告，需要设置一些参数





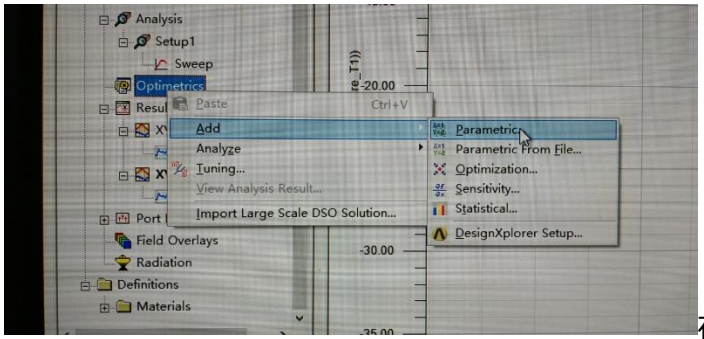


生成 2.4G 单个回波损耗曲线。

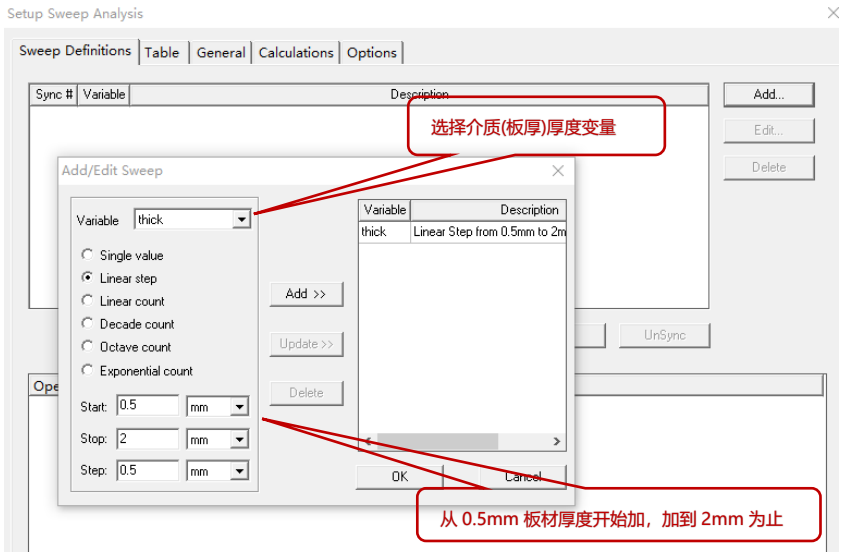


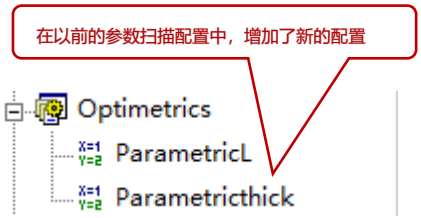
使用 mark 可以看出来，2.45G 时回波损耗有-39.5dB

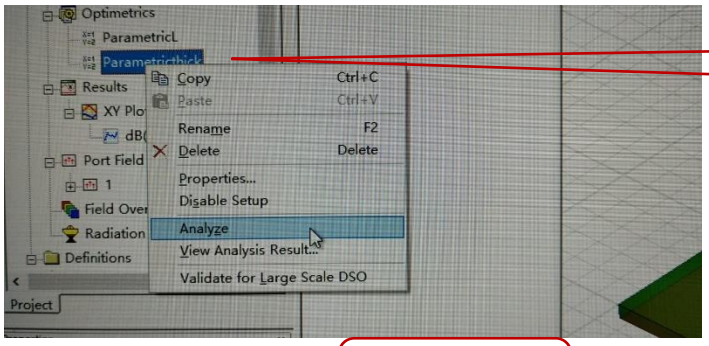
PCB 板厚对天线的影响，扫描仿真

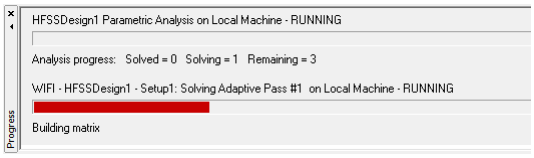


在原来的基础上又增加 PCB 板厚参数扫描

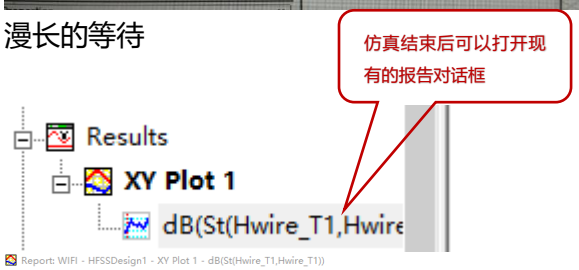


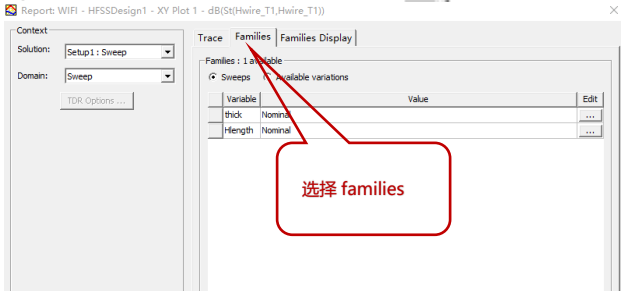


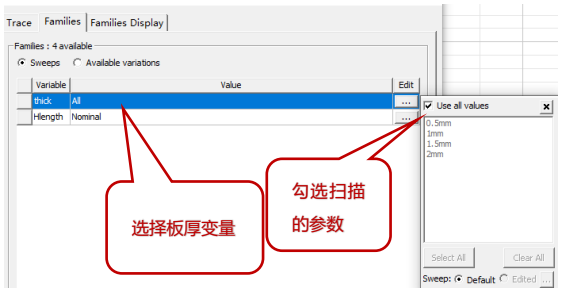


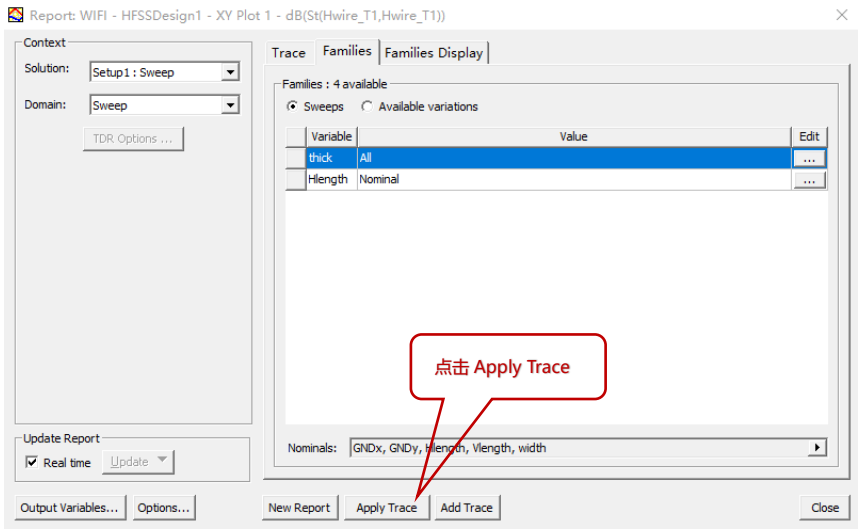


漫长的等待

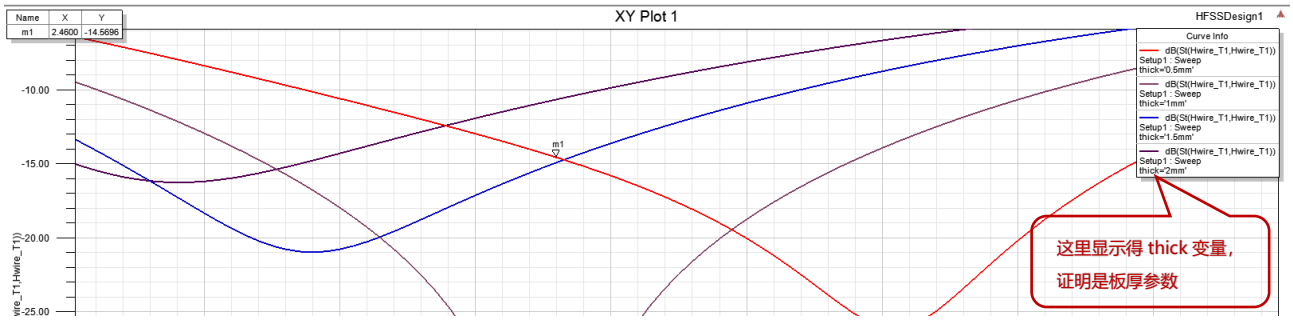






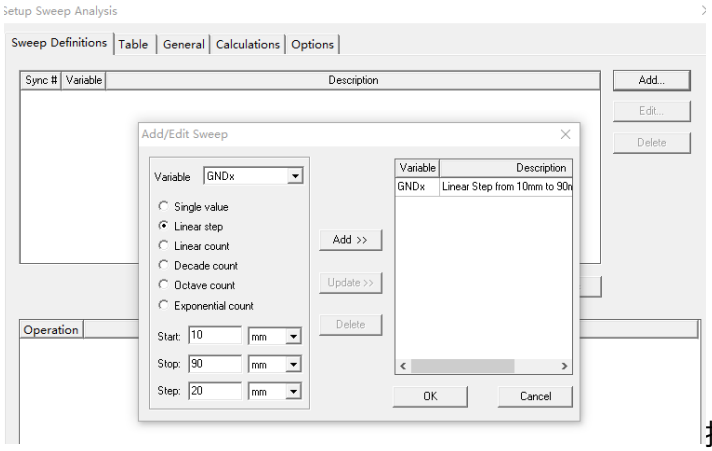


生成回波损耗与板厚的关系



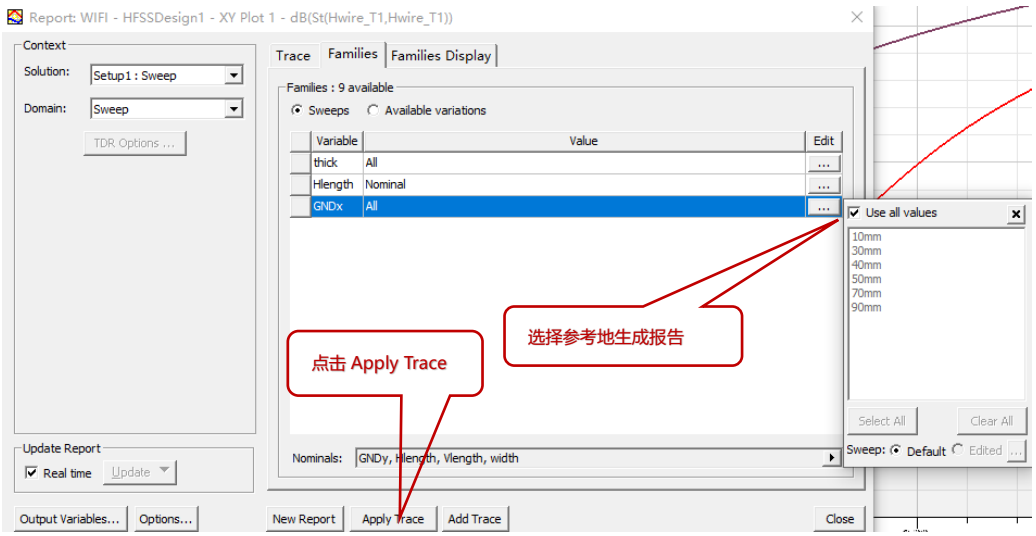
从仿真结果可以看出，即使天线尺寸不变，PCB 板厚度发生变化，天线谐振频率也会发生巨大变化。随着 PCB 板厚度增加，天线的谐振频率段会降低。

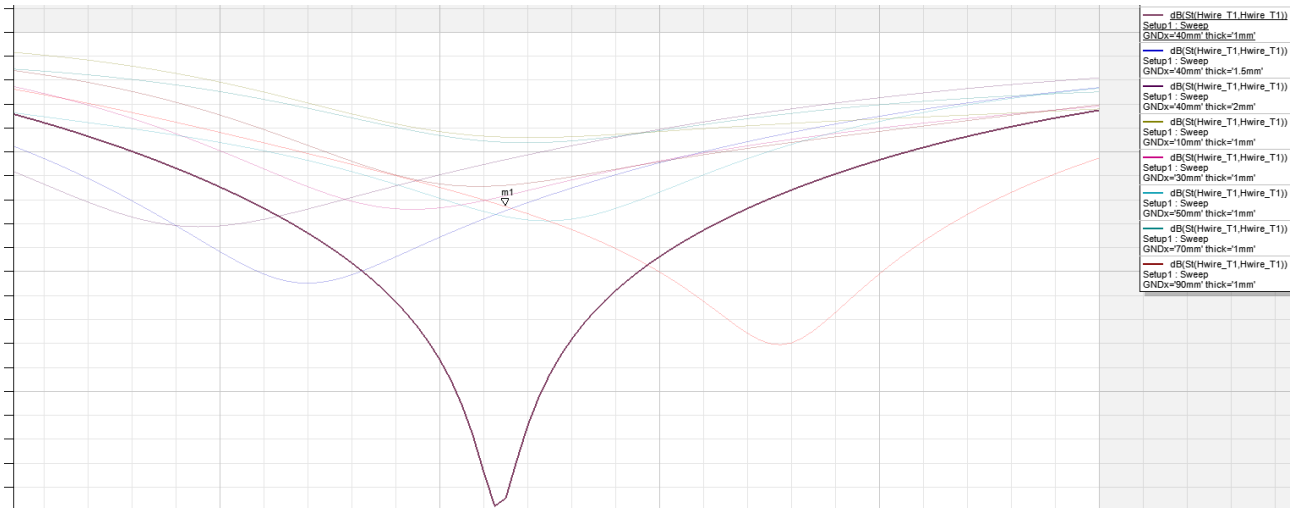
参考地大小对天线性能的影响



扫描修改 GND x 方向的大小

开始进行扫描分析





根据仿真结果，参考地大小的变化也对天线有一定的影响，参考地大小变化对天线谐振频率影响不大。但是对天线输入阻抗影响很大。

蛇形倒 L PCB 天线设计

蛇形倒 L 天线和上一章讲的普通倒 L 天线都是单极子天线结构，所以蛇形倒 L 天线和普通倒 L 天线性能是一样的。只是蛇形倒 L 天线节约 PCB 尺寸。

我们知道一个 PCB 天线，只要参考地和 PCB 厚度尺寸都固定了，那么天线的工作频率只和天线长度有关

PCB 板设计要求

介质材料: FR4, Er = 4.4

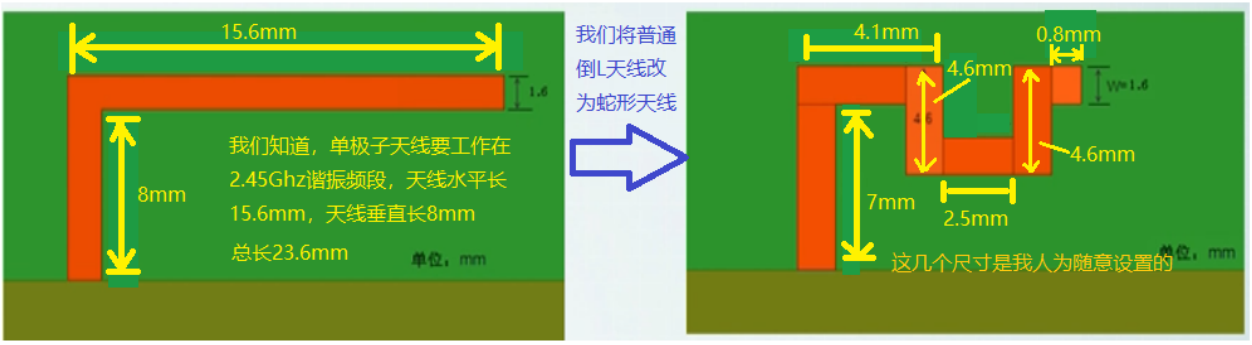
板厚: 1mm

天线走线宽度: 1.6mm

PCB 板尺寸: 65mm x 40mm

参考地大小: 40mm x 40mm

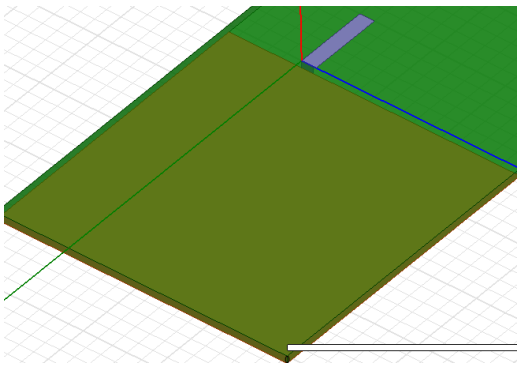
工作频率还是使用 2.45Ghz wifi 频段

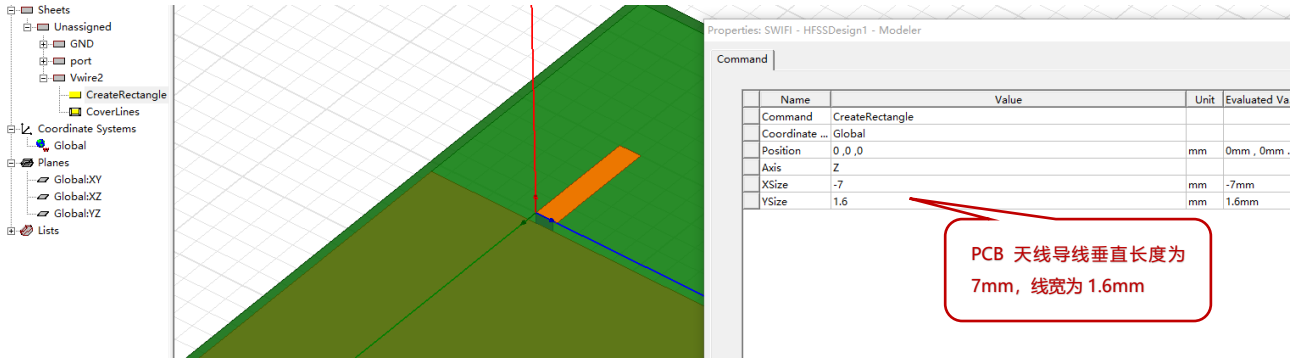


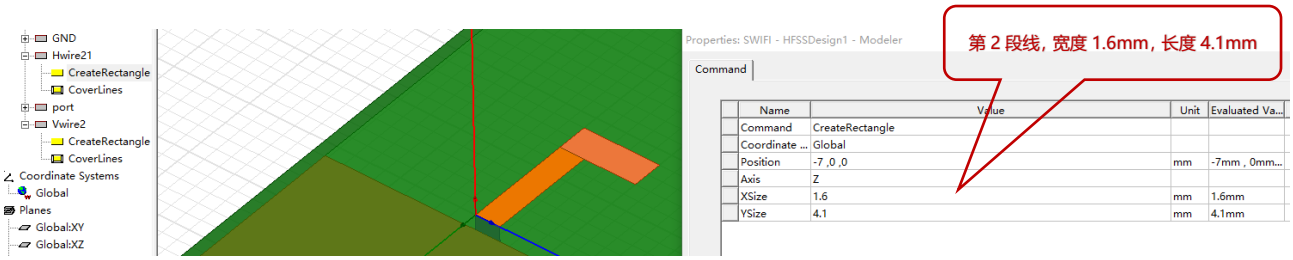
所以我们先估算出普通倒L天线的长度

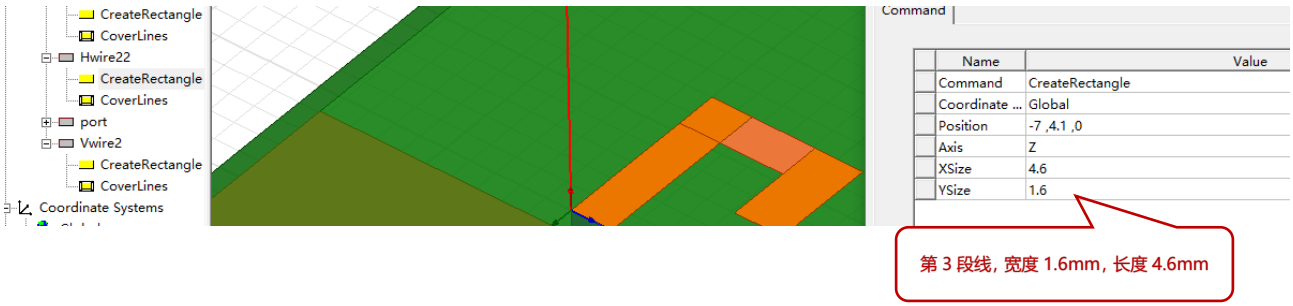
这个蛇形天线总长度还是23.6mm

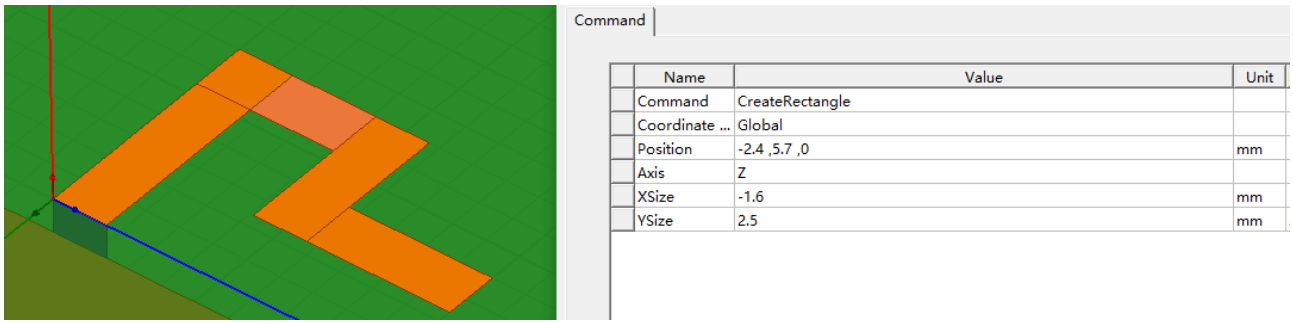
我们将上一章节的工程拷贝过来修改



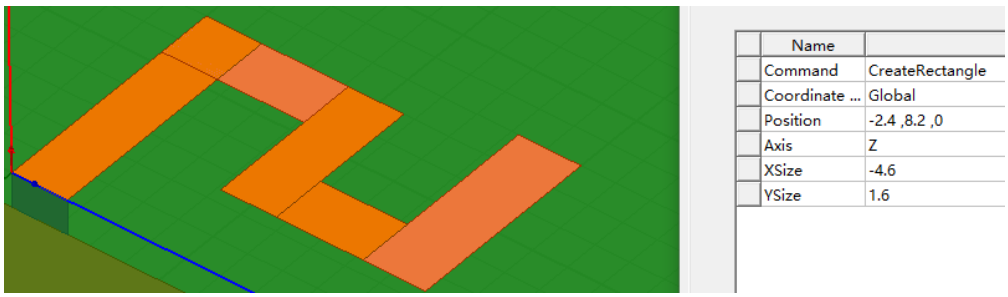


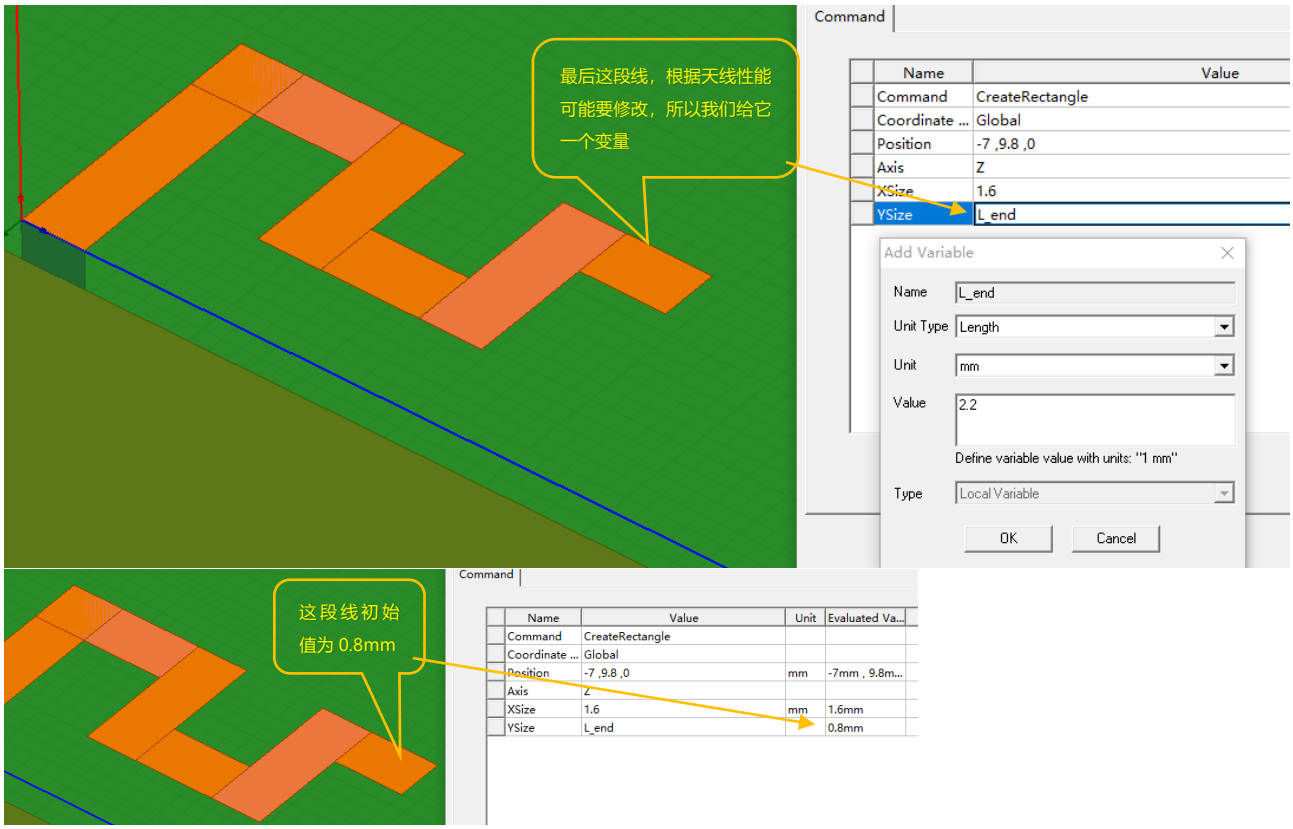


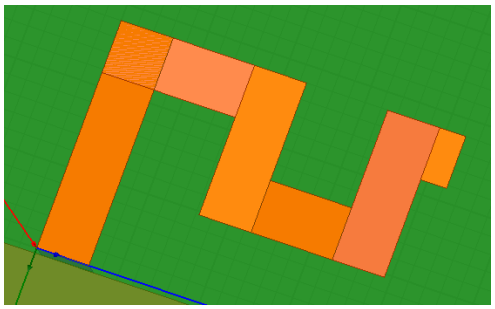




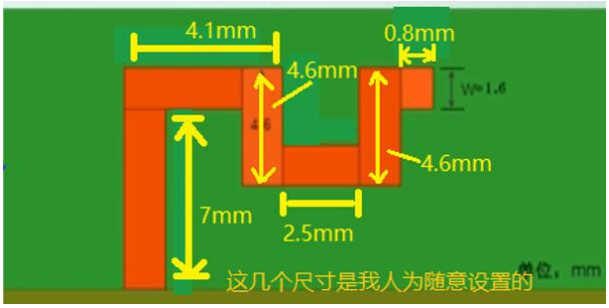
第 4 段线，宽度 1.6mm，长度 2.5mm



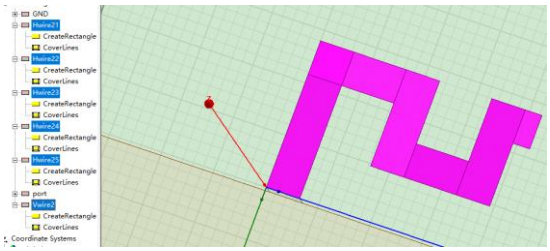




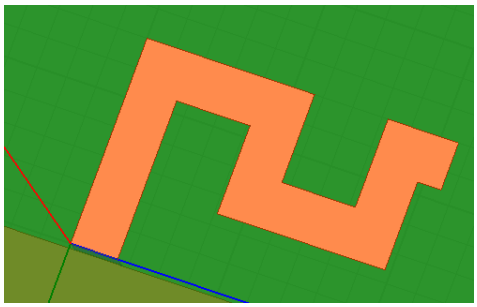
HFSS 设计的天线



我设想的蛇形天线尺寸。



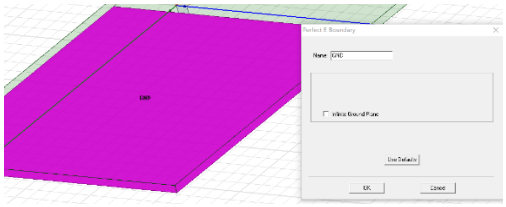
将线段连接起来



线段连接起来后

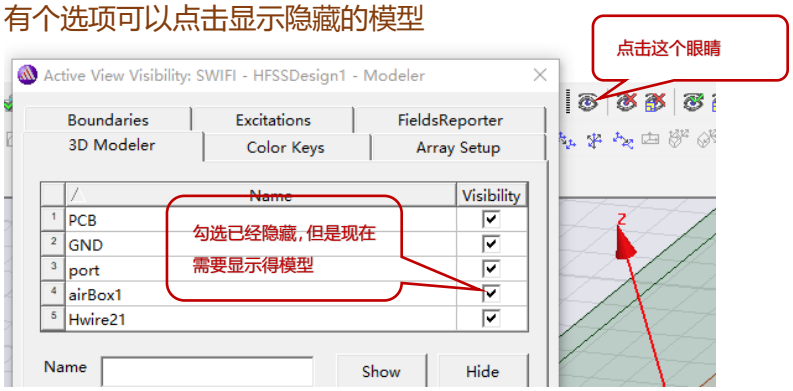
下面设置激励条件，边界条件，求解设置

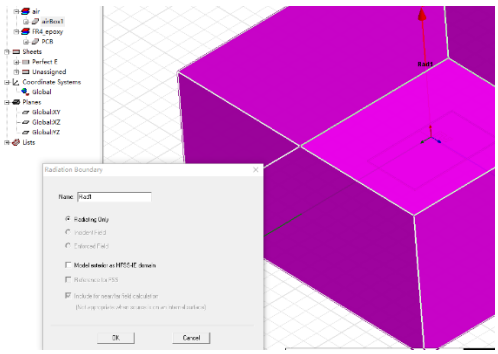
设置导线电气属性

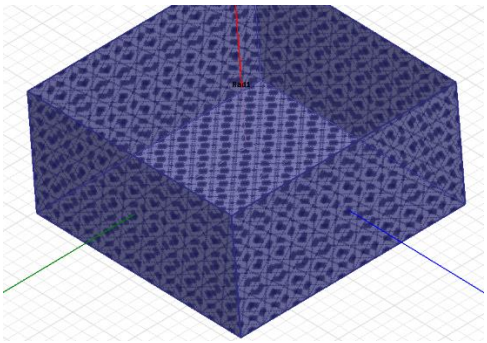


设置 GND 电气属性

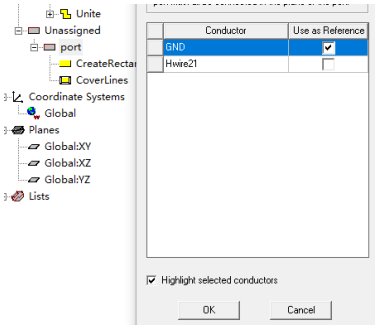
有个选项可以点击显示隐藏的模型



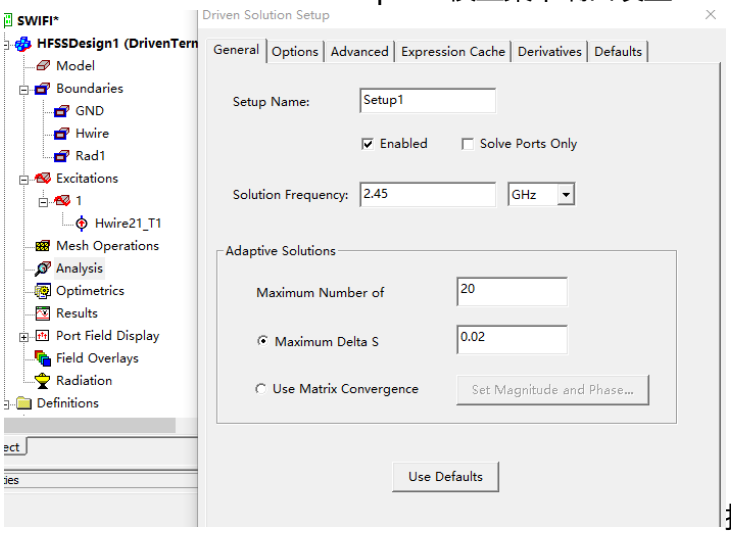




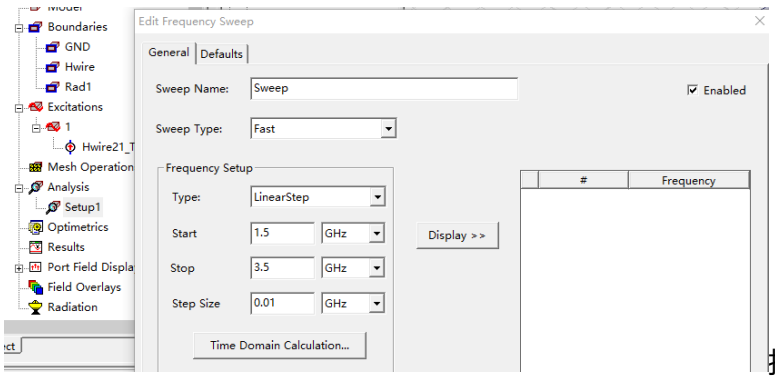
空气盒模型设置边界条件



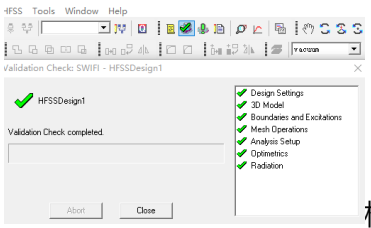
port 模型集中端口设置



扫描频率设置，和前面章节差不多

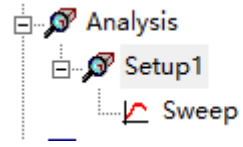


扫描频率范围设置，和前面差不多

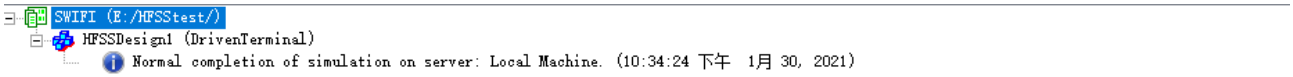


检验模型参数都正确

下面开始仿真



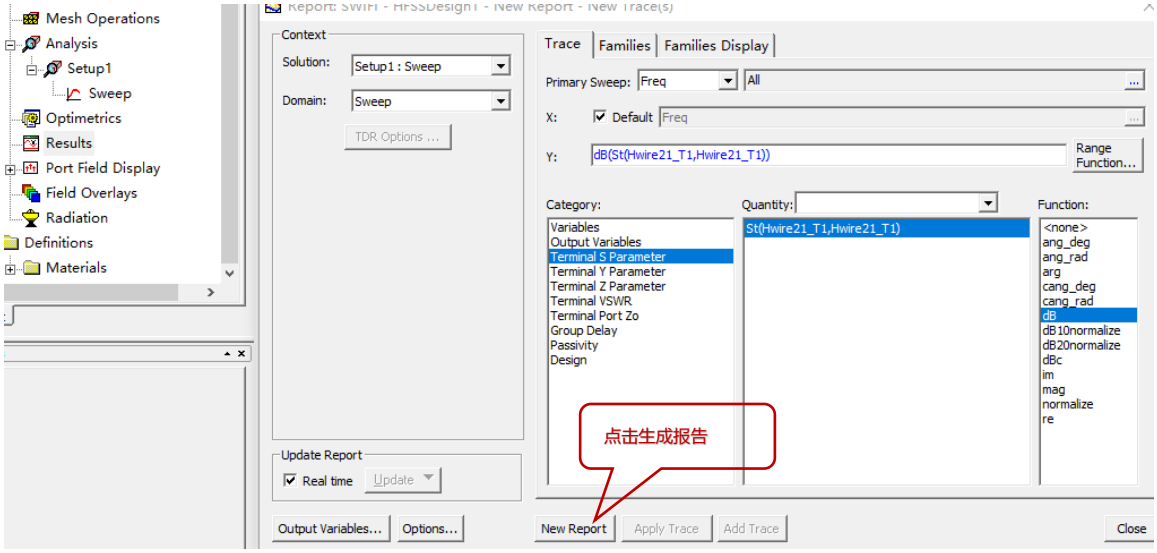
右键 Setup1->Analyze，开始仿真

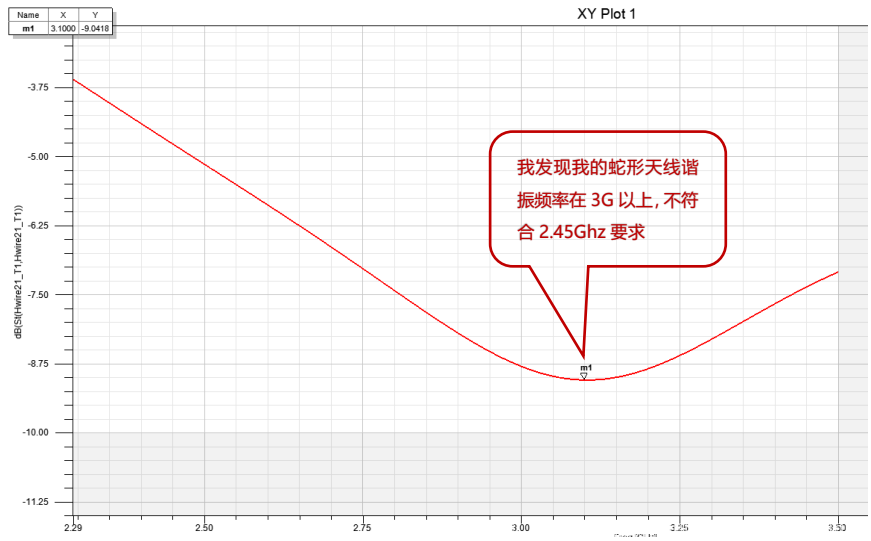


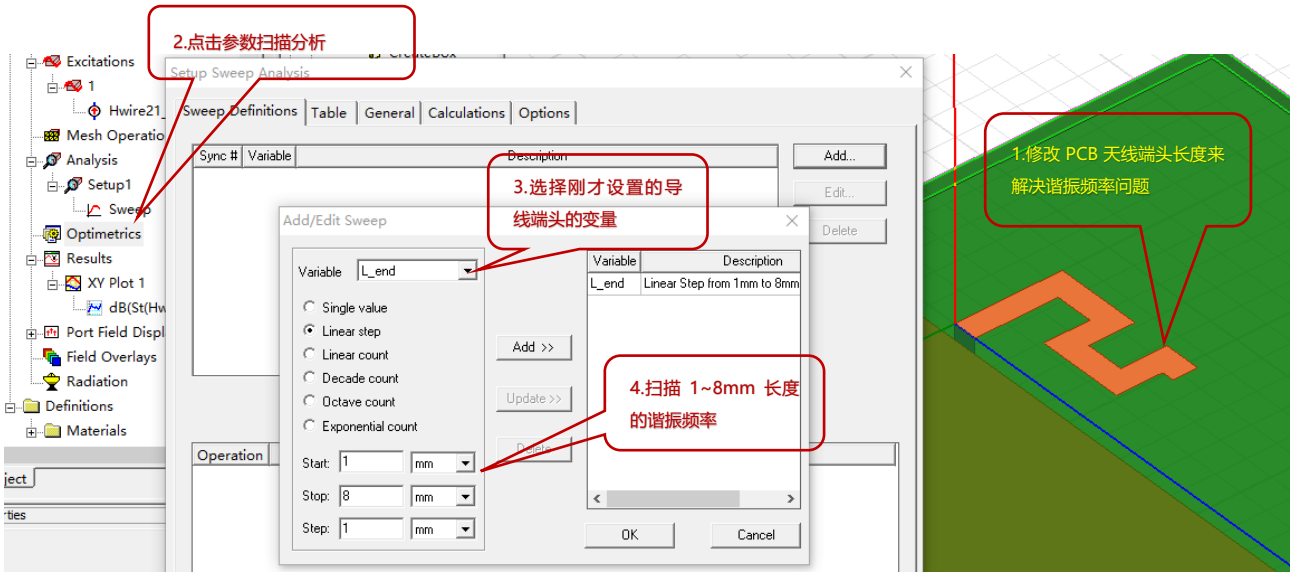
显示仿真分析正常结束

Results->Create Terminal Solution Data Report 打开报告

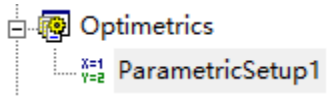
再选择 Rectangular Plot



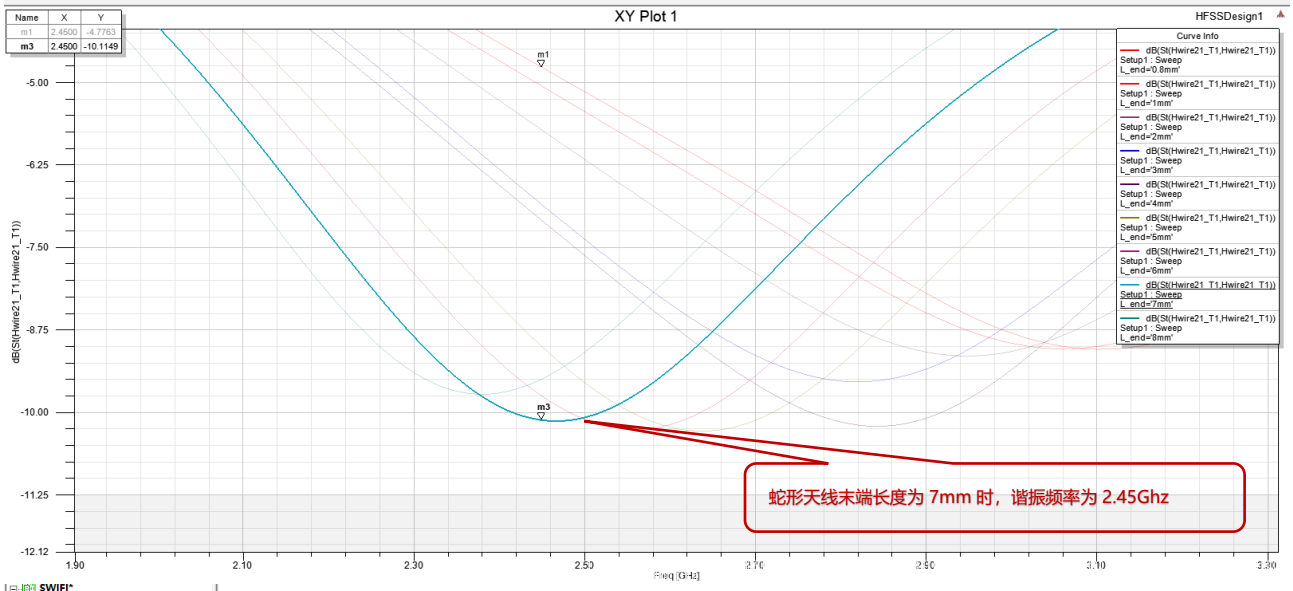


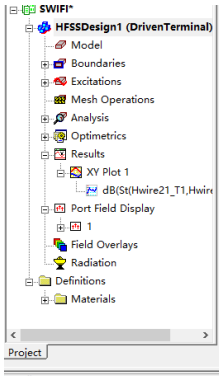


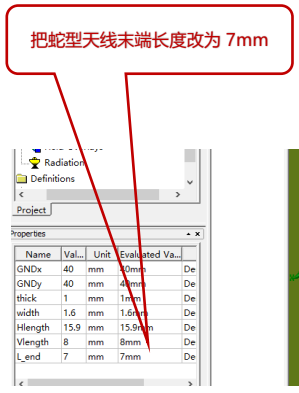
设置完毕

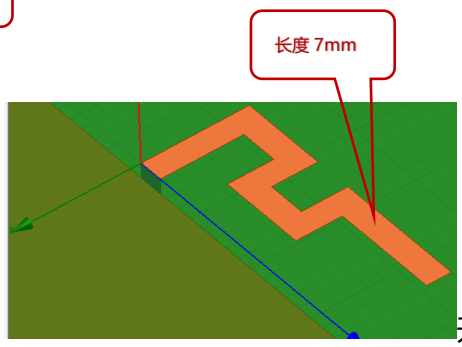


在 Optimetrics 下仿真

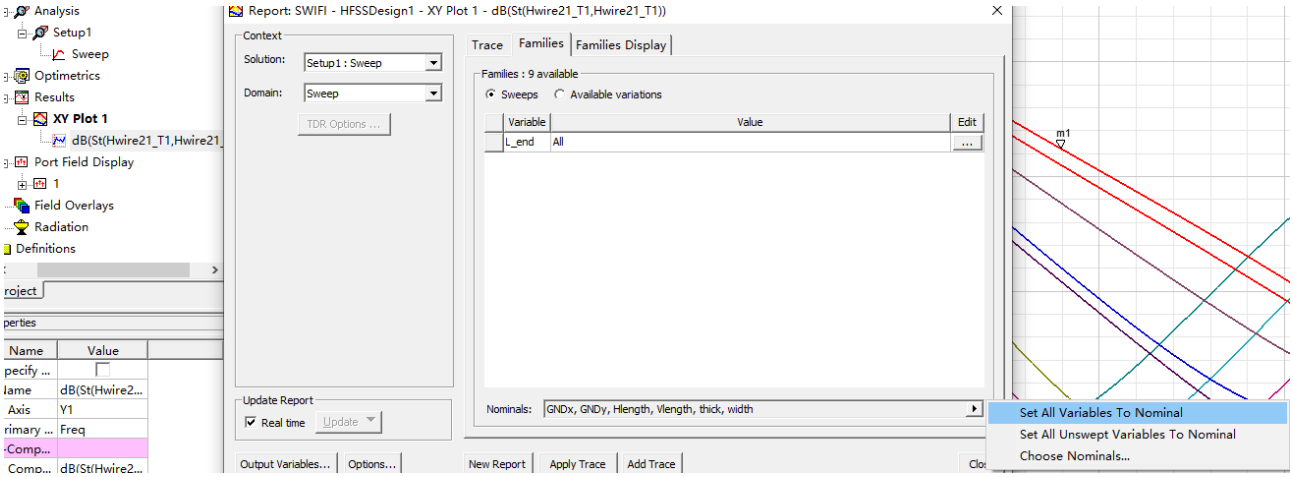




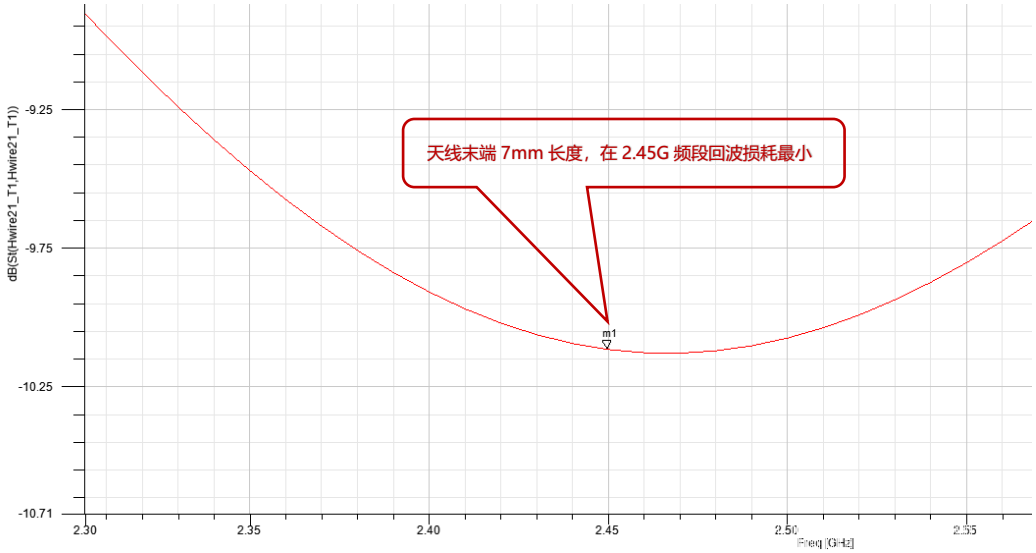


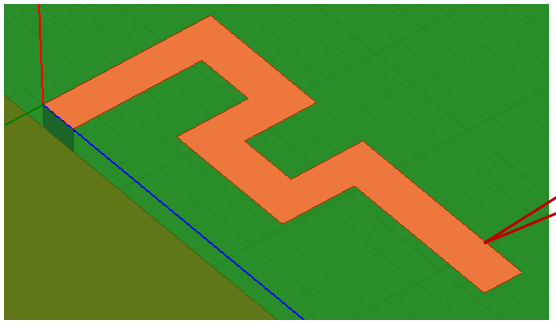


开始仿真

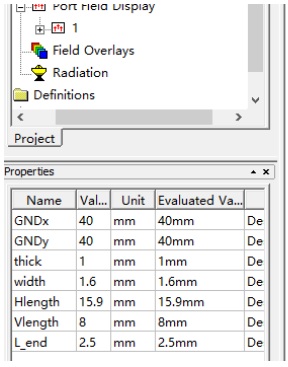


在仿真报告 families 中，将 L_end 变量设置为初始值，然后点击 Apply Trace

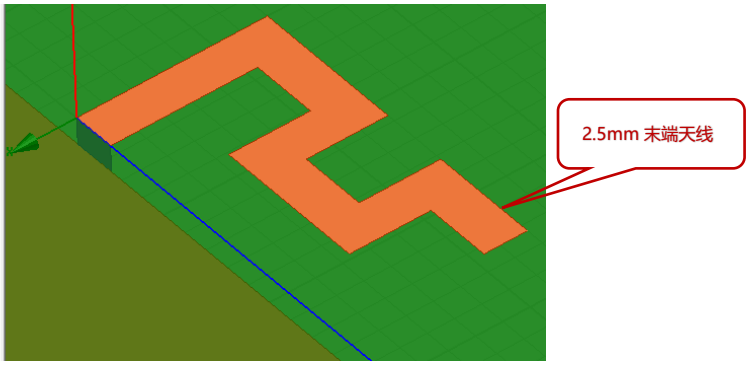


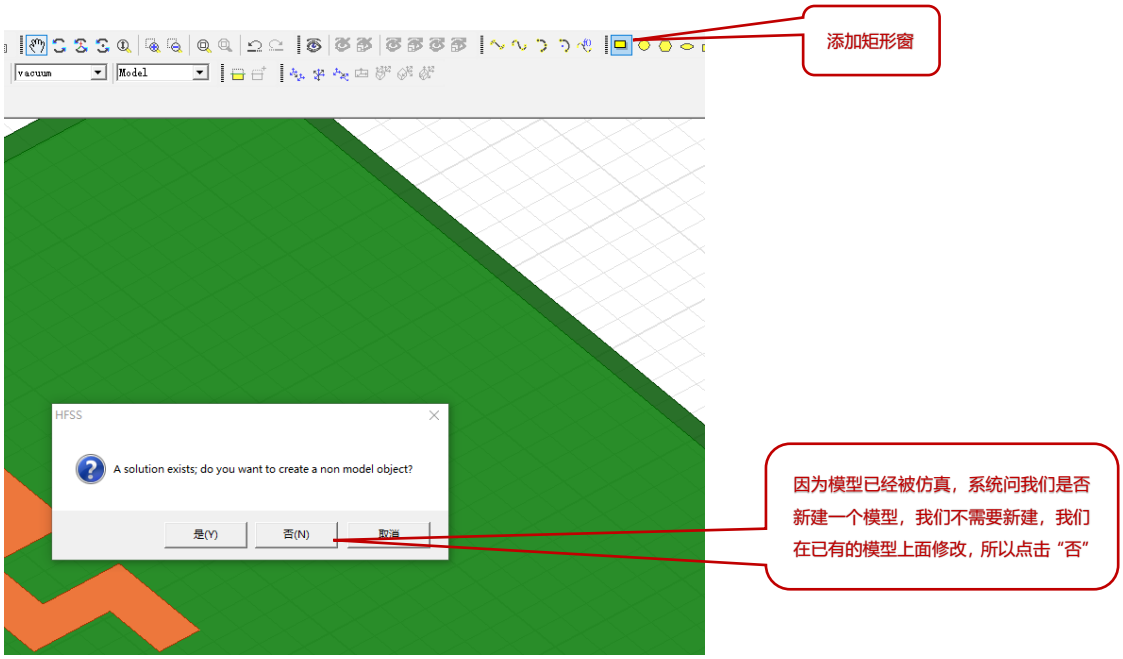


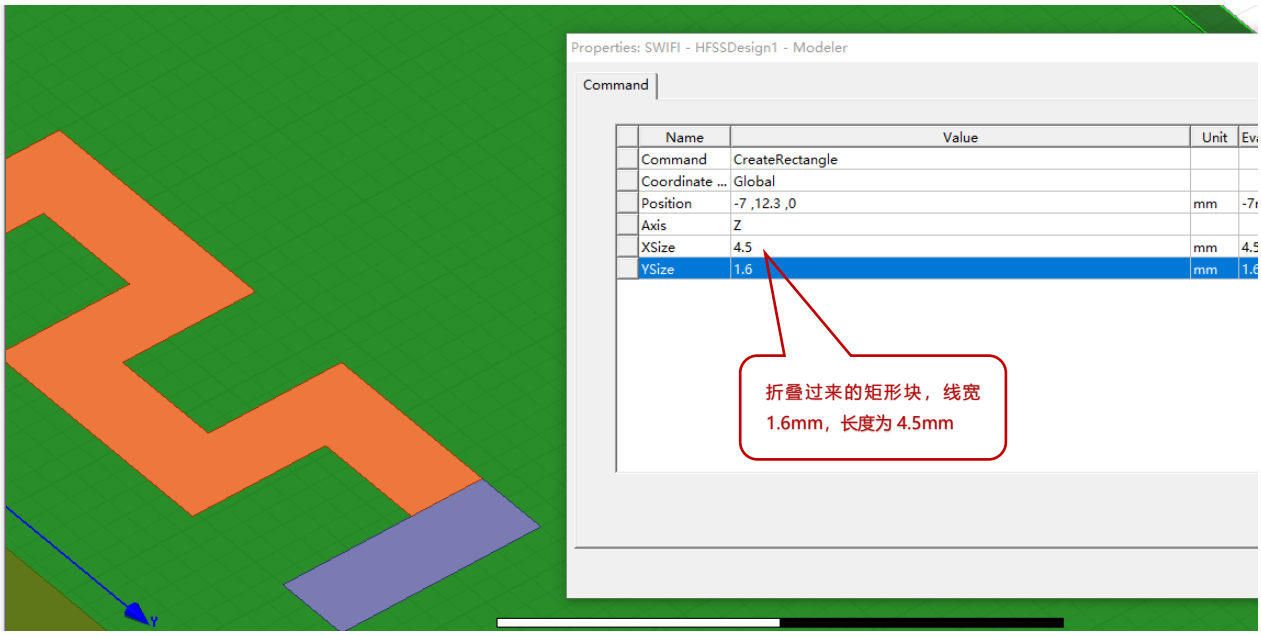
末端长度我们改成 2.5mm。那么 7mm-2.5mm，折叠过来的天线长度就为 4.5mm



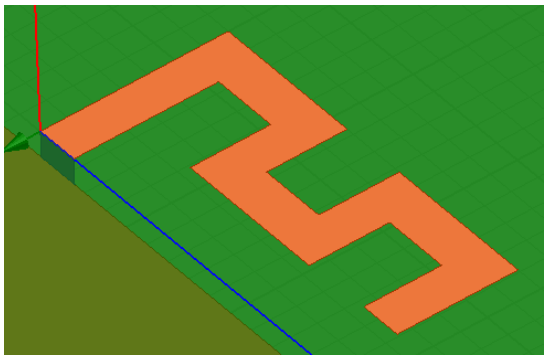
Name	Val...	Unit	Evaluated Va...	
GNDx	40	mm	40mm	De
GNDy	40	mm	40mm	De
thick	1	mm	1mm	De
width	1.6	mm	1.6mm	De
Hlength	15.9	mm	15.9mm	De
Vlength	8	mm	8mm	De
L_end	2.5	mm	2.5mm	De

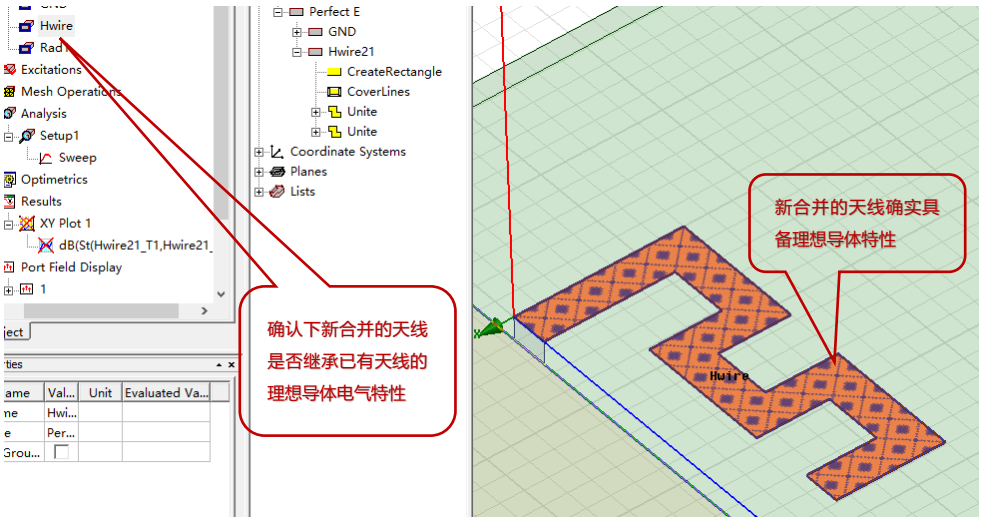



添加矩形窗
HFSS
A solution exists; do you want to create a non model object?
是(Y)
否(N)
取消
因为模型已经被仿真，系统问我们是否新建一个模型，我们不需要新建，我们在已有的模型上面修改，所以点击“否”

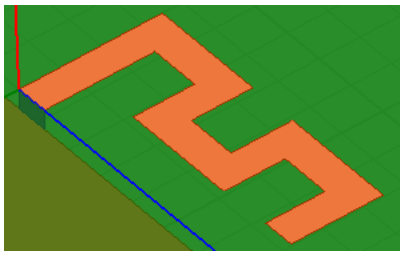

Properties: SWIFI - HFSSDesign1 - Modeler
Command
Name
Value
Unit
Command
CreateRectangle
Coordinate ...
Global
Position
-7 ,12.3 ,0
mm
Axis
Z
XSize
4.5
mm
YSize
1.6
mm
折叠过来的矩形块，线宽1.6mm，长度为 4.5mm

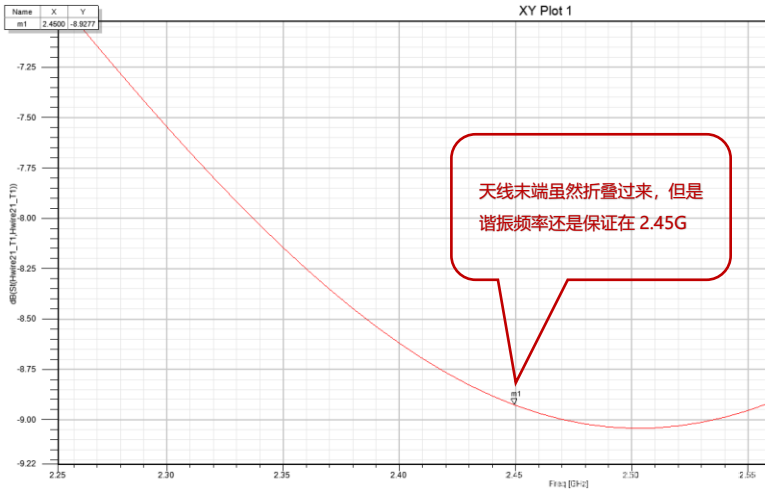
记住要合并天线图形



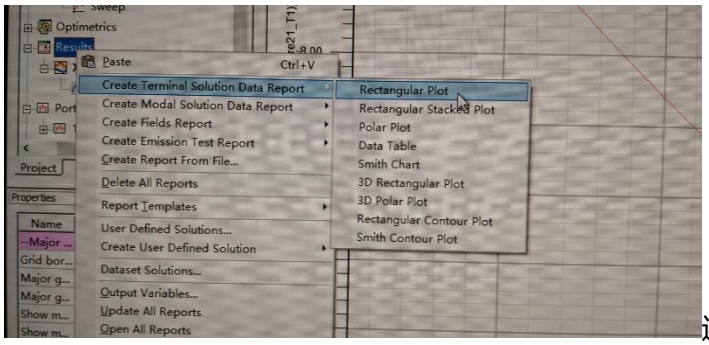


在 Analysis->setup1 运行仿真分析

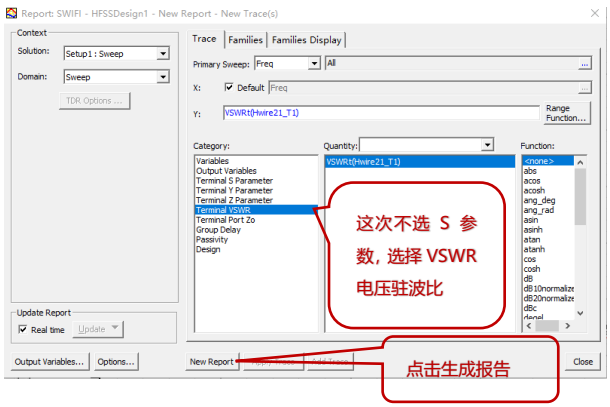


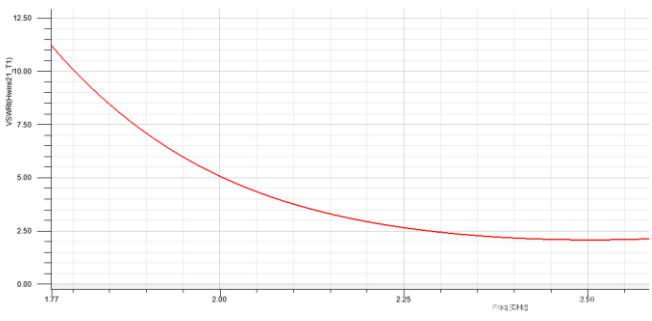


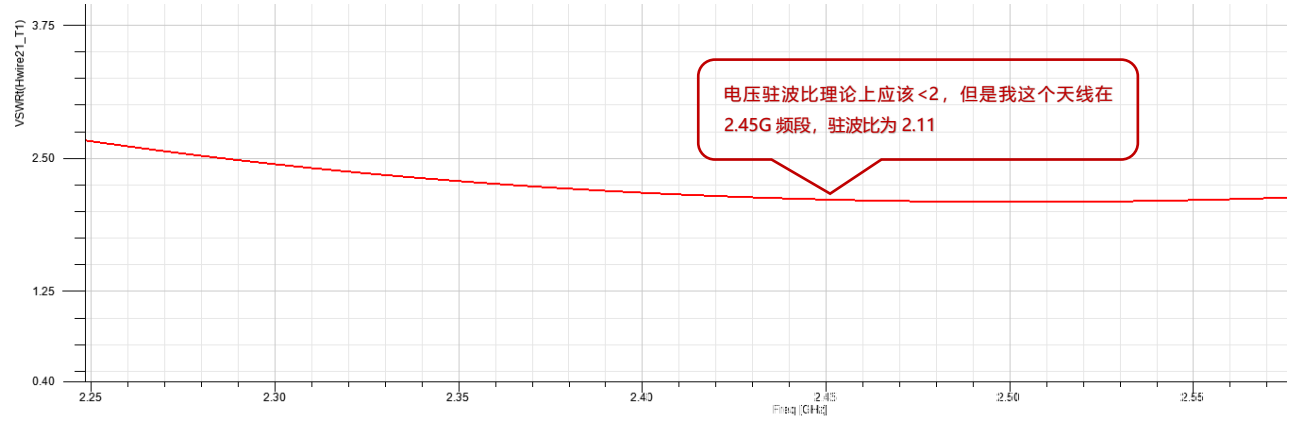
天线电压驻波比



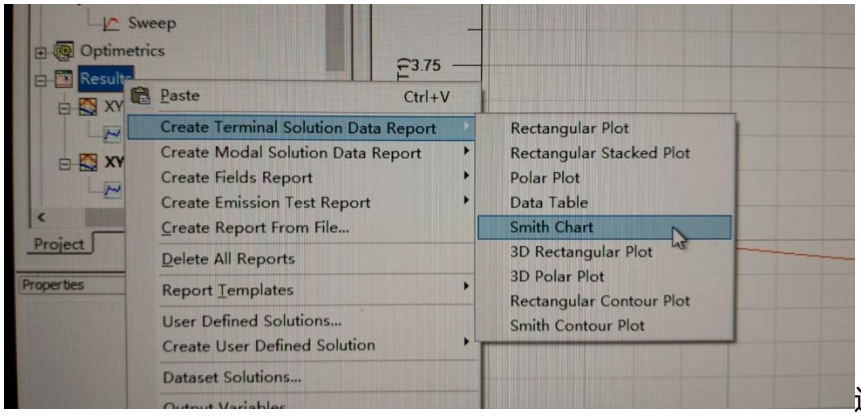
还是选择报告查询驻波比



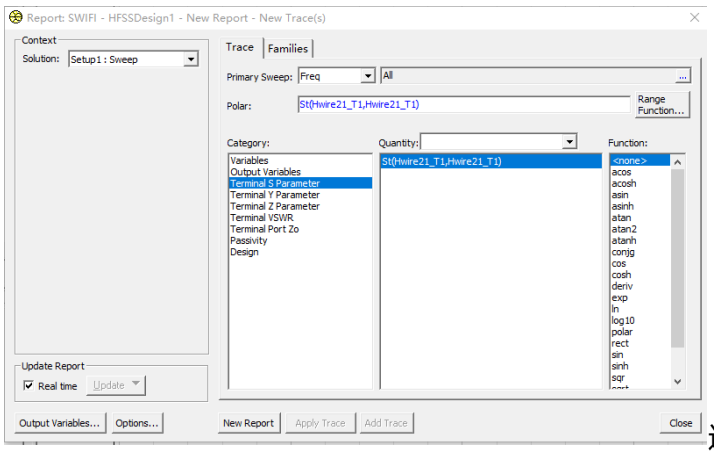




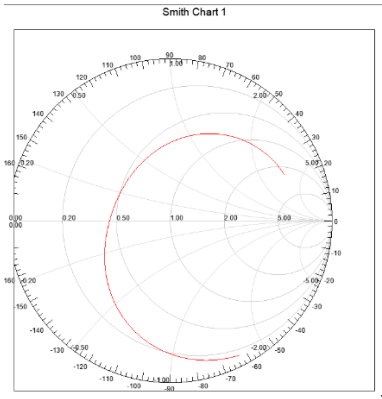
史密斯圆图查看天线阻抗



选择 Smith Chart 查看天线阻抗

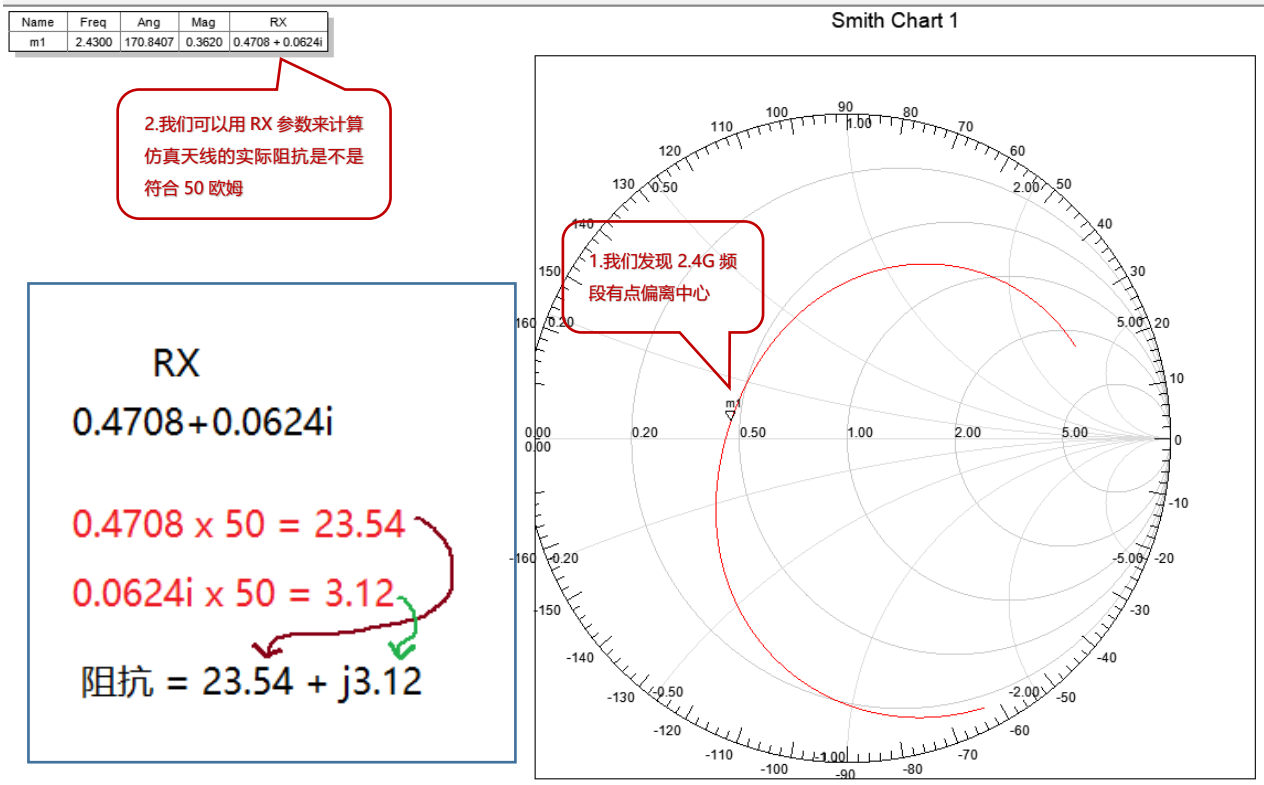


还是选择 S 参数，点击 New Report



这就是 smith 圆图

我们用 mark 点寻找 smith 圆图在 2.4G 的位置

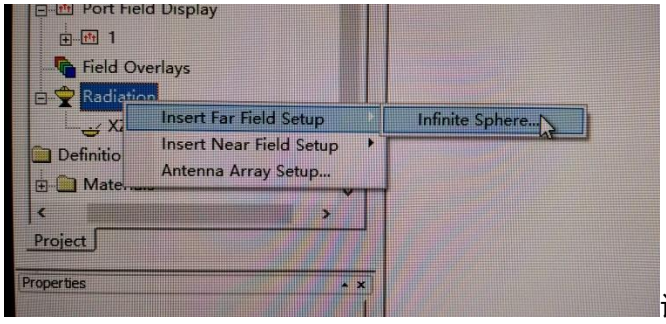


所以在 2.45Ghz 时，天线输入阻抗为 23.54+j3.12

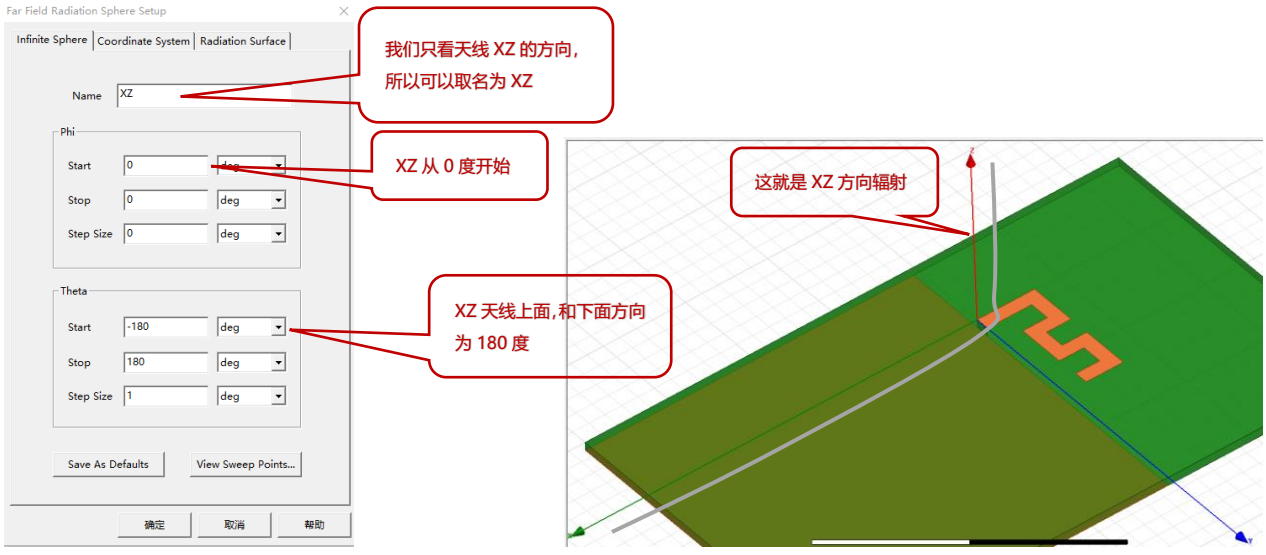
最重要是查看天线辐射方向图

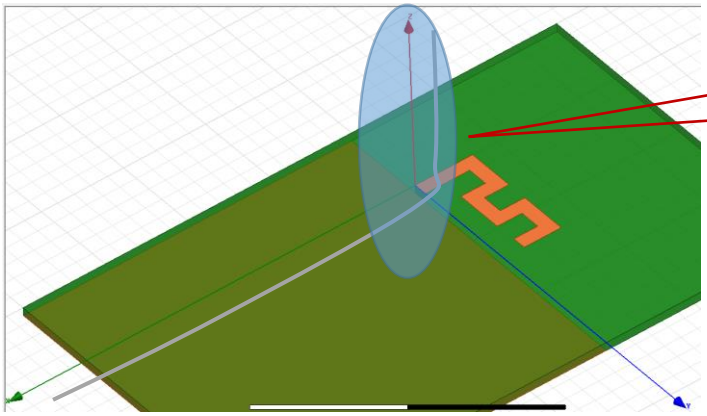
如何查看二维天线方向图

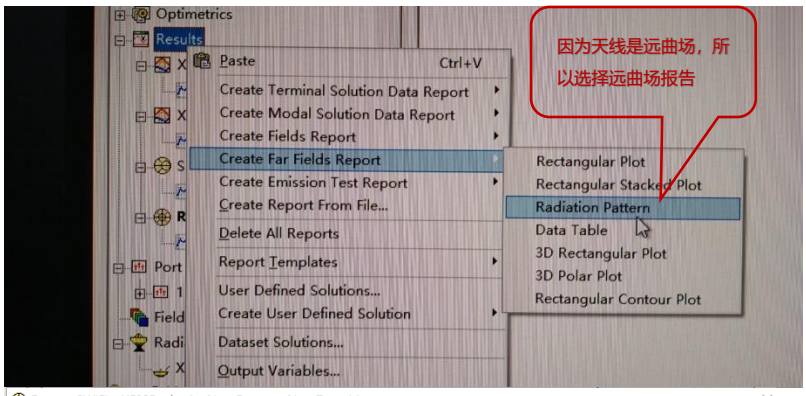
选择 XZ 方向的天线方向图，那么就要把 XZ 方向定义为辐射表面

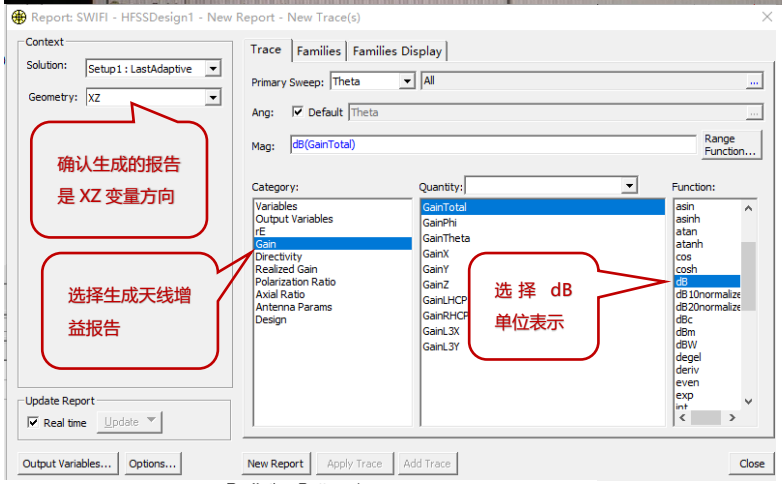


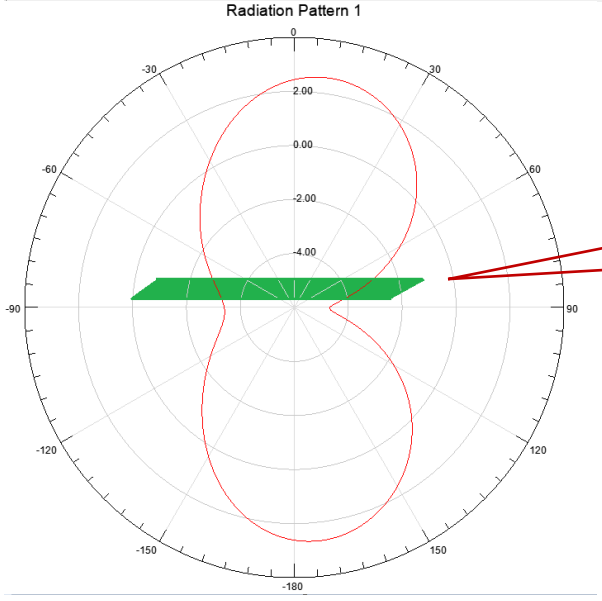
设置天线 XZ 方向参数



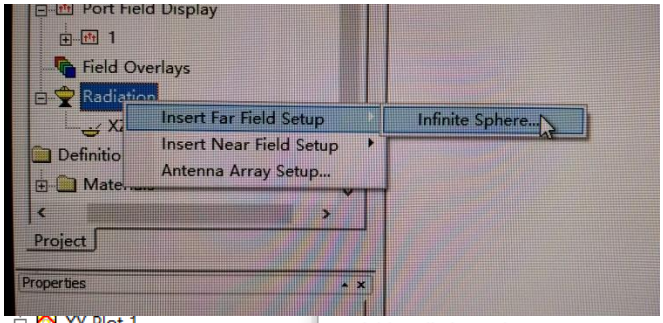

XZ 也就是天线辐射穿过电路板，天线铜皮上面和下面的辐射

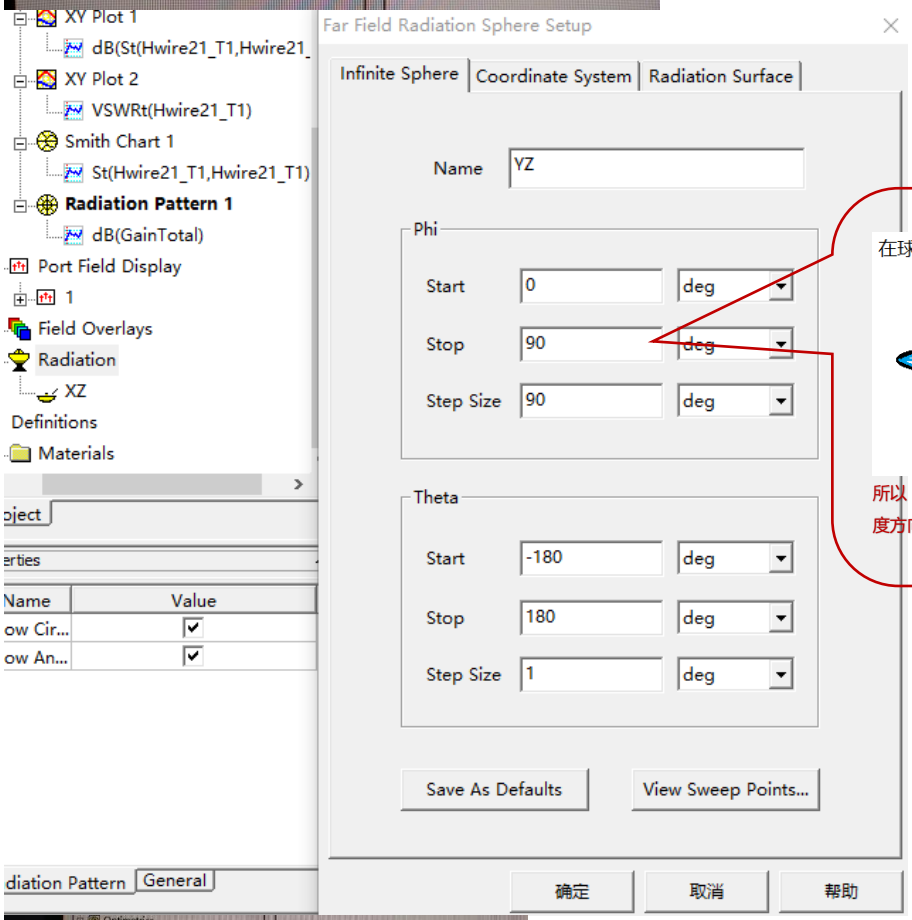

因为天线是远曲场，所以选择远曲场报告
Optimetrics
Results
Paste Ctrl+V
Create Terminal Solution Data Report
Create Modal Solution Data Report
Create Fields Report
Create Far Fields Report
Create Emission Test Report
Create Report From File...
Delete All Reports
Report Templates
User Defined Solutions...
Create User Defined Solution
Dataset Solutions...
Output Variables...
Rectangular Plot
Rectangular Stacked Plot
Radiation Pattern
Data Table
3D Rectangular Plot
3D Polar Plot
Rectangular Contour Plot

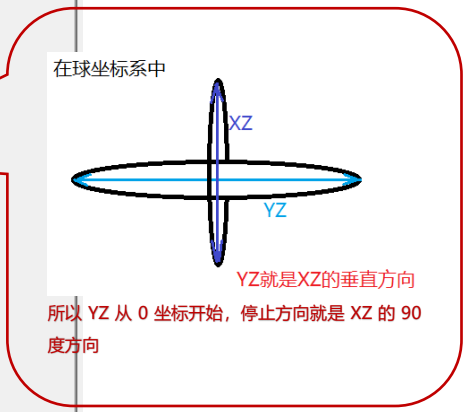

Report: SWIFI - HFSSDesign1 - New Report - New Trace(s)
Context
Solution: Setup1 : LastAdaptive
Geometry: XZ
确认生成的报告是 XZ 变量方向
选择生成天线增益报告
Trace
Families
Families Display
Primary Sweep: Theta
Ang: Default Theta
Mag: dB(GainTotal)
Range Function...
Category:
Quantity:
Function:
Variables
Output Variables
rE
Gain
Directivity
Realized Gain
Polarization Ratio
Axial Ratio
Antenna Params
Design
GainTotal
GainPhi
GainTheta
GainX
GainY
GainZ
GainLHCP
GainRHCP
GainL3X
GainL3Y
选 择 dB 单位表示
asin
asinh
atan
atanh
cos
cosh
dB
dB10normalize
dB20normalize
dBc
dBm
dBW
degel
deriv
even
exp
Update Report
Real time
Update
Output Variables...
Options...
New Report
Apply Trace
Add Trace
Close

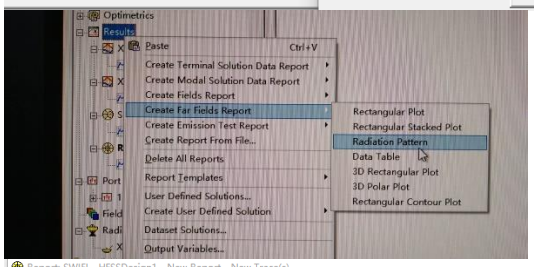

Radiation Pattern 1
0
30
60
90
120
150
-180
-150
-120
-90
-60
-30
2.00
0.00
-2.00
-4.00
XZ 方向，电路板上下方向的辐射增益强度

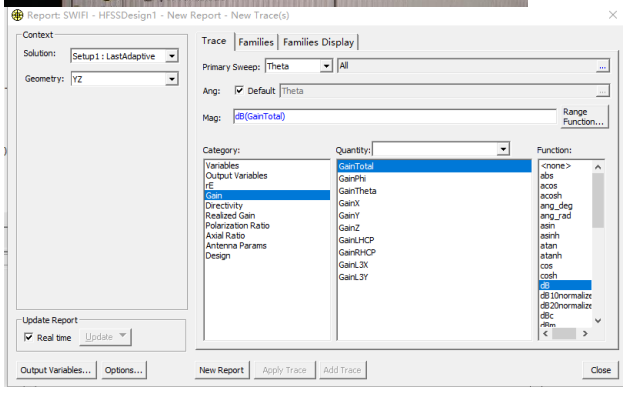
下面查看 YZ 方向的辐射图

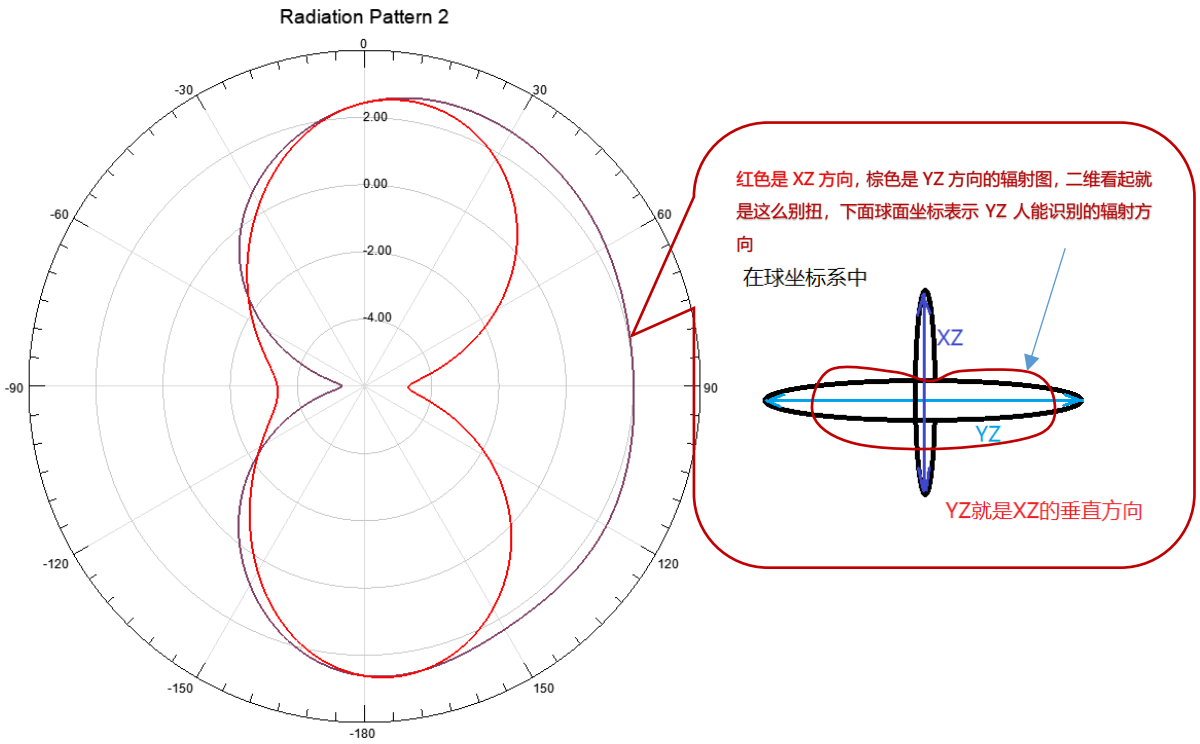






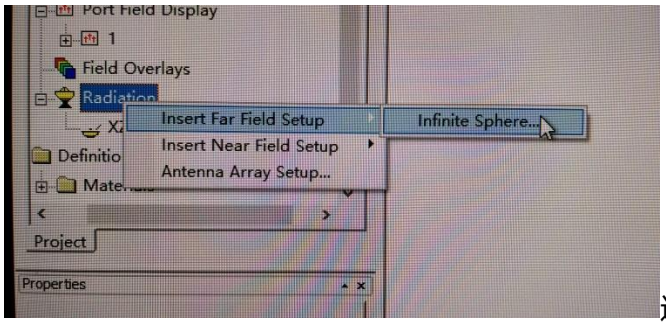




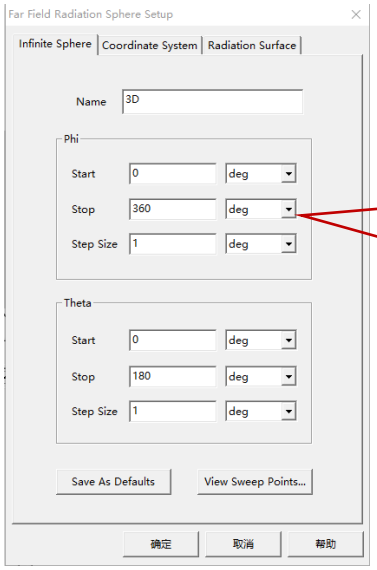


所有为了解决二维坐标系不直观现象，下面进行三维坐标的辐射图仿真

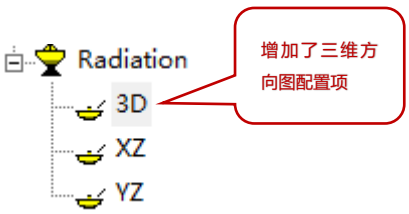
三维辐射面仿真



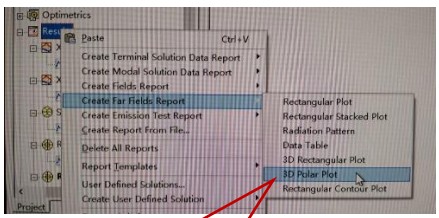
还是打开方向图参数设置



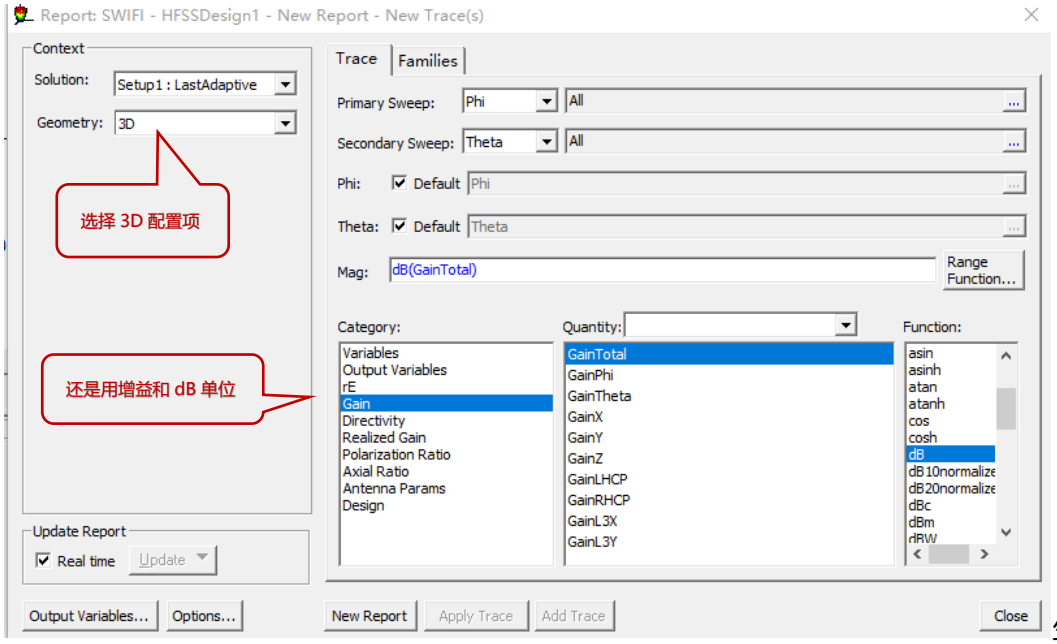
三维方向图就是0~360度扫描



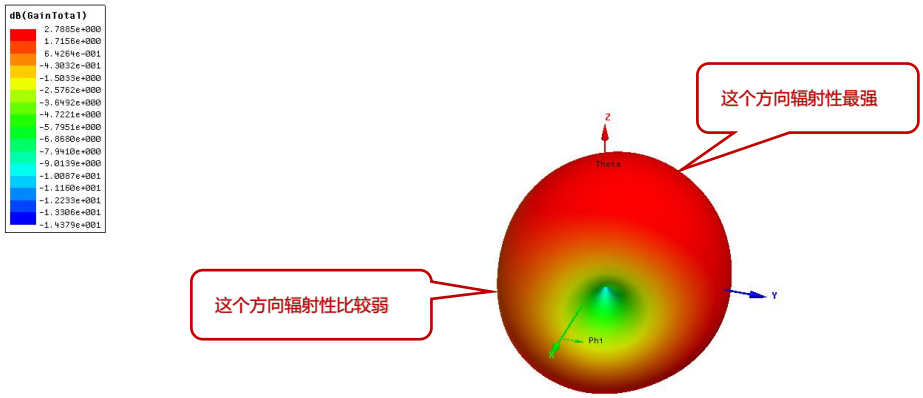
增加了三维方向图配置项



选择 3D 报告生成

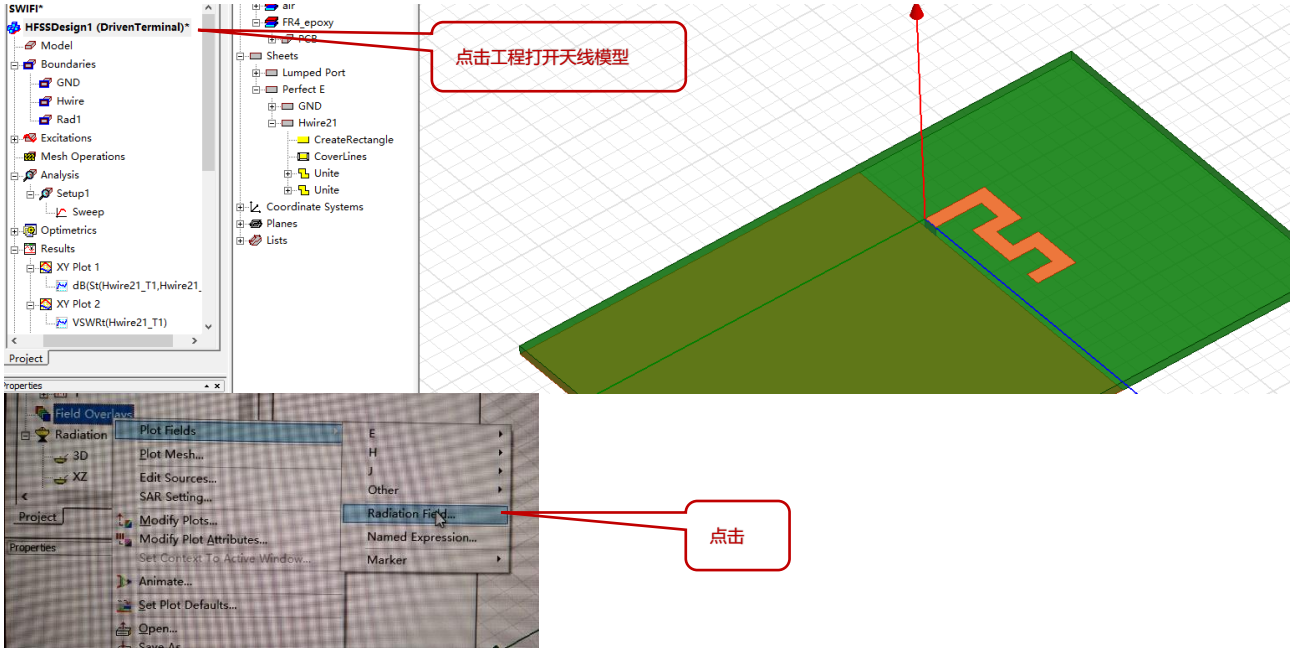


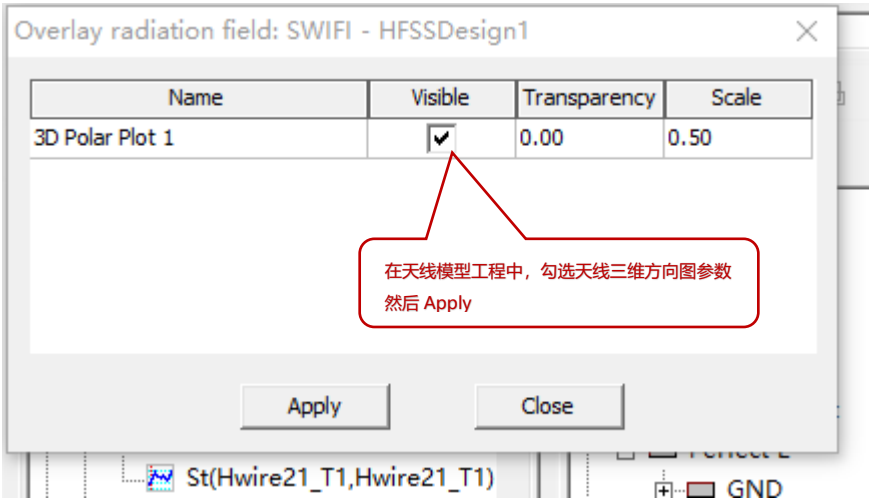
生成报告

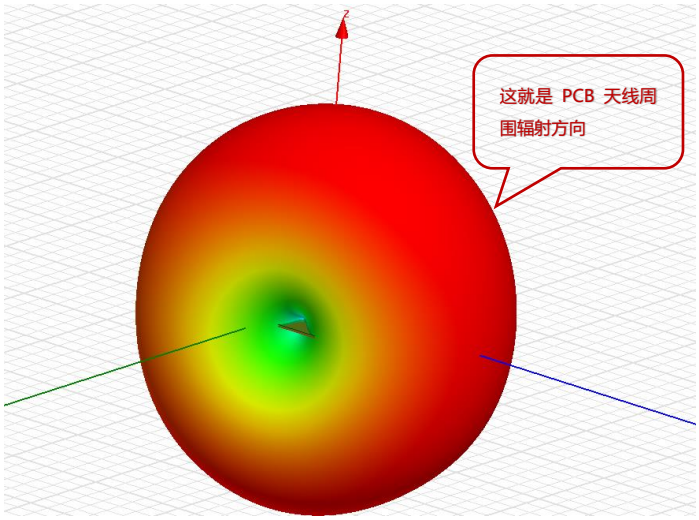


如果我们要将天线三维方向图和前面 PCB 天线模型做对比，怎么办呢？

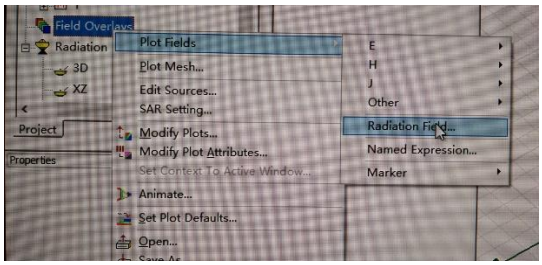
将天线三维方向图和 PCB 天线模型叠加到一起

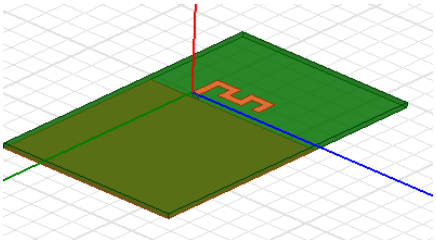




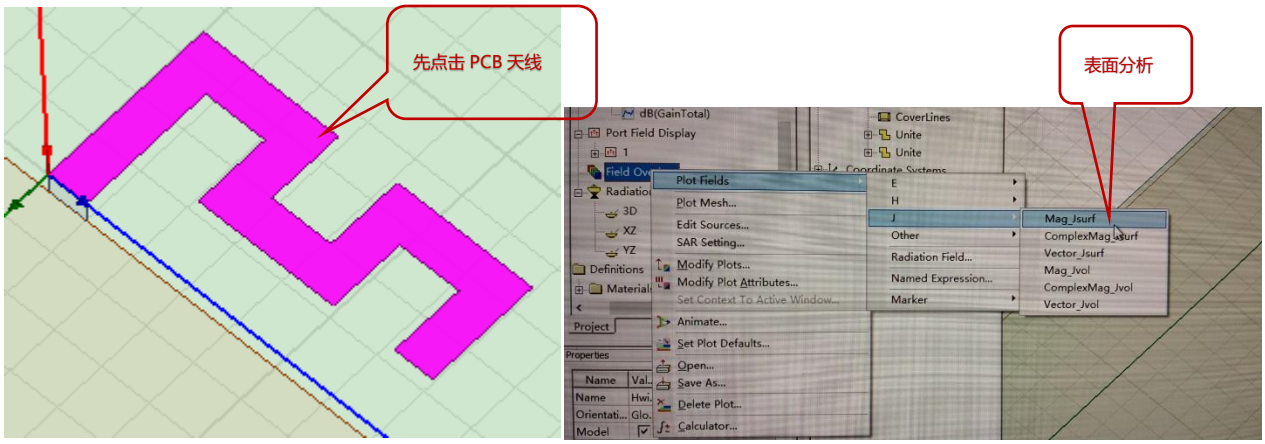


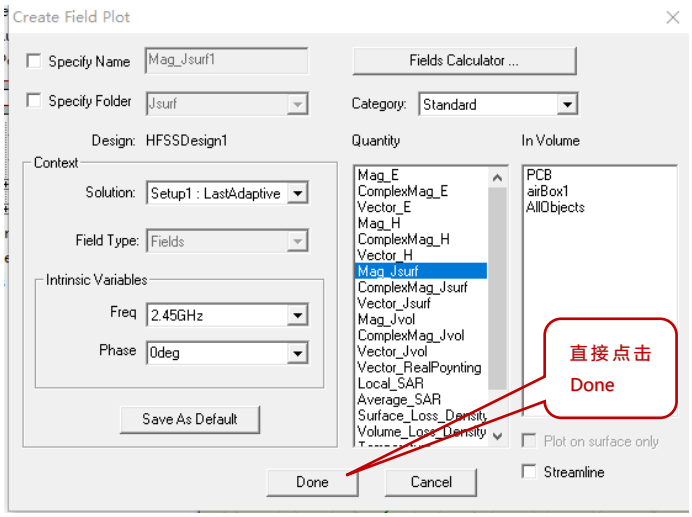
如果要取消天线模式的方向图

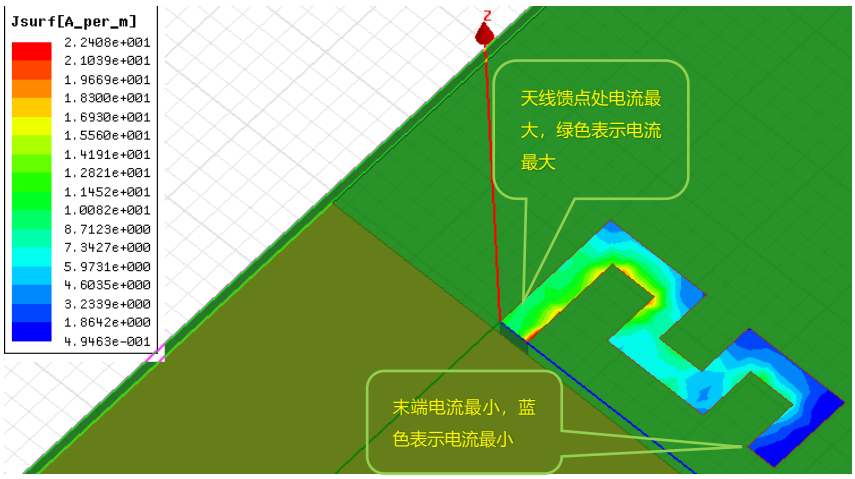




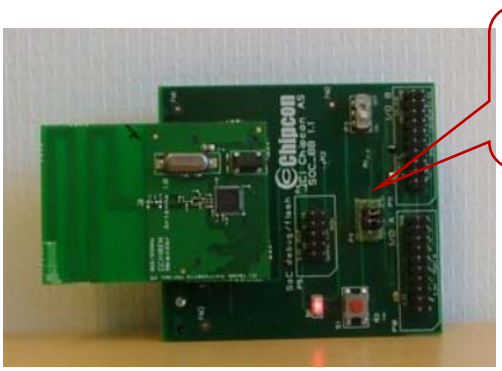
PCB 天线表面电流分布



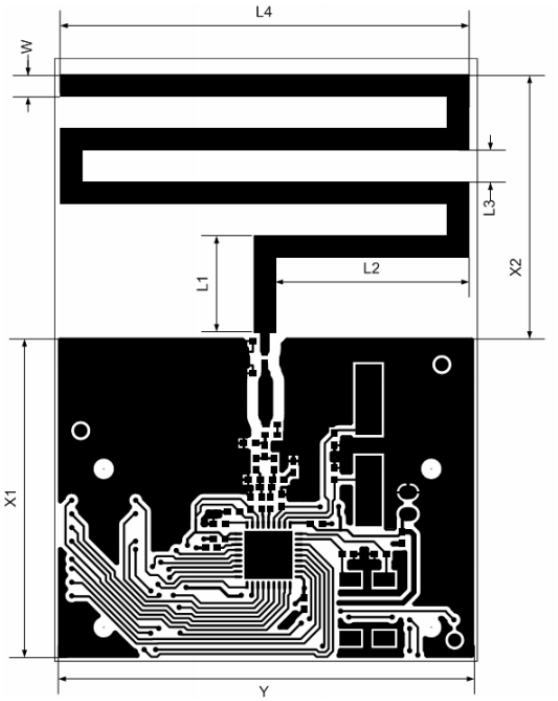




TI 单极子天线 868M,915M,955M 仿真



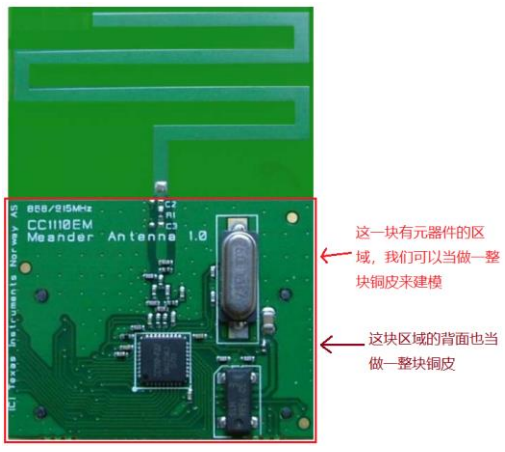
以下是 TI 官方给出的天线尺寸设计

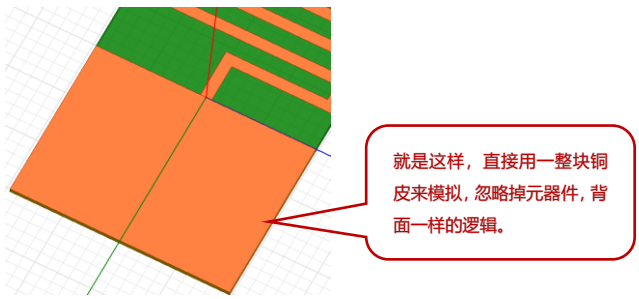


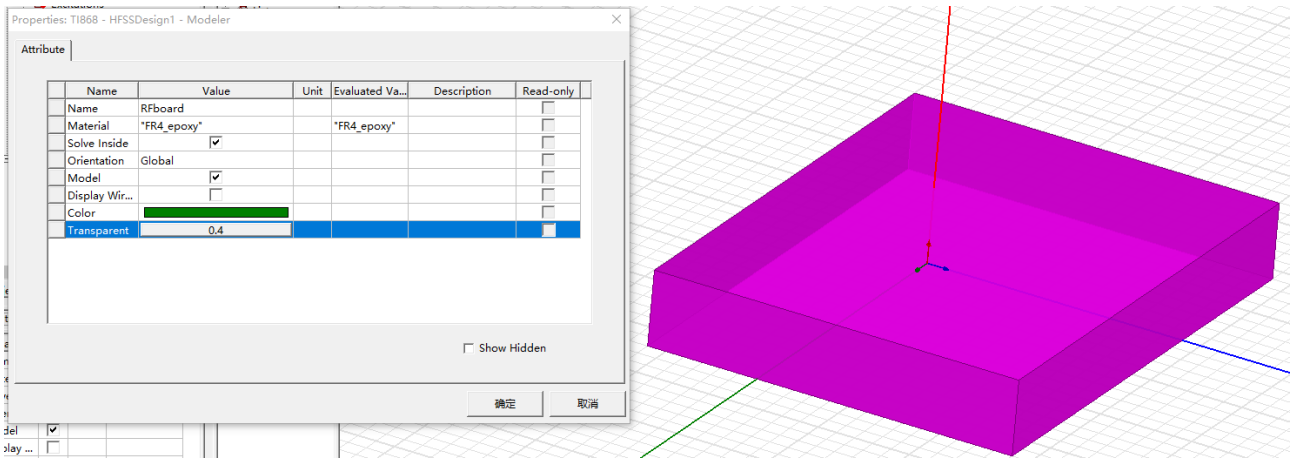
L1	9.0 mm	**Y**	39.0 mm
L2	18.0 mm	**X1**	30.0 mm
L3	3.0 mm	**X2**	24.0 mm
L4	38.0 mm	**W**	2.0 mm

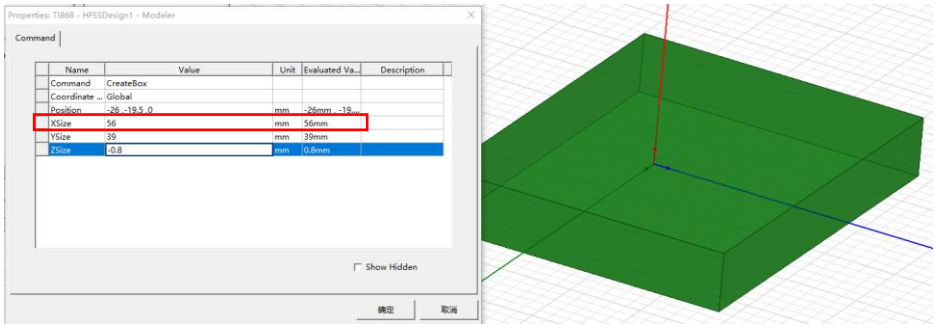
该天线 PCB 板厚度 0.8mm，PCB 板介质材料是 FR4

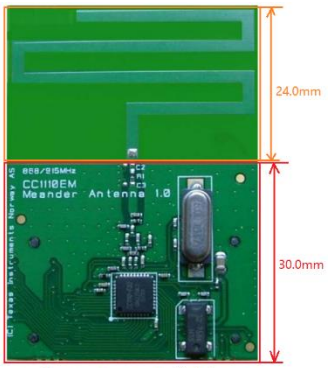
设计思路





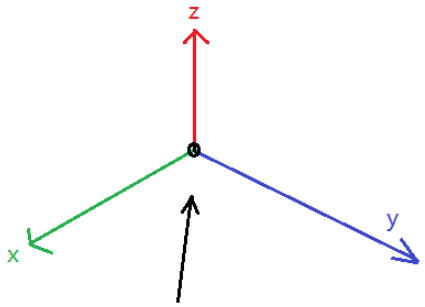






PCB长 30.0+24.0 = 54mm 我们留余量取56mm

怎么理解原点位置?

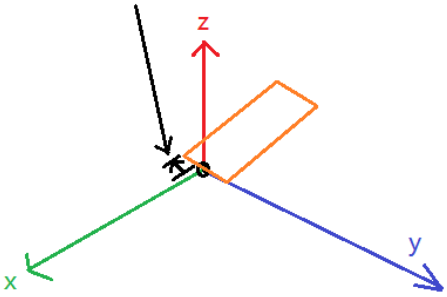


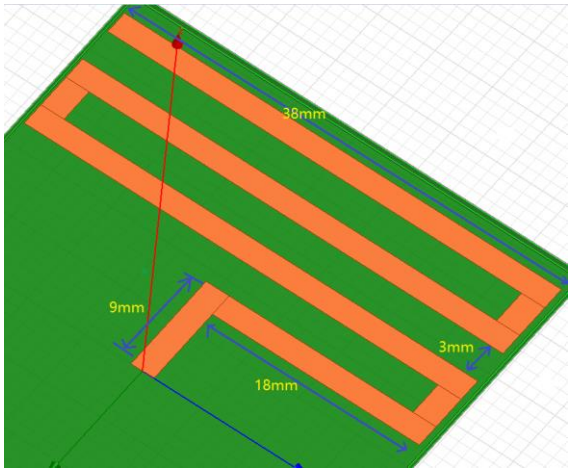
(x = 0,
y =0,
z = 0)

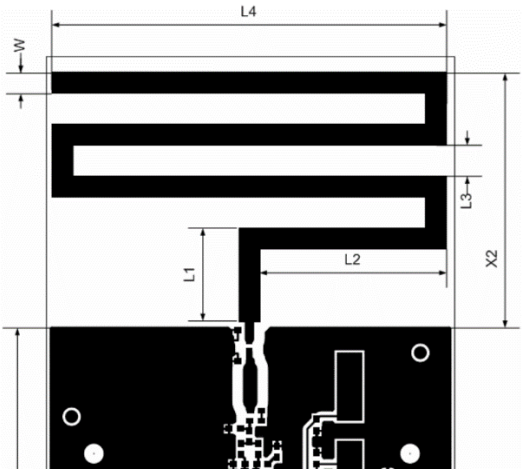
这一点就是原点，不管你软件界面怎么晃动，原点都在这里，

Position(0，-1, 0)

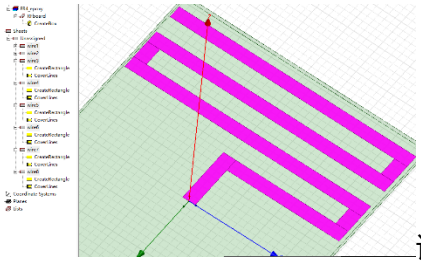
这个-1就是从原点开始，向y 负方向移动，因为我的天线宽度正好是2mm，所以y移动-1mm刚好天线一半





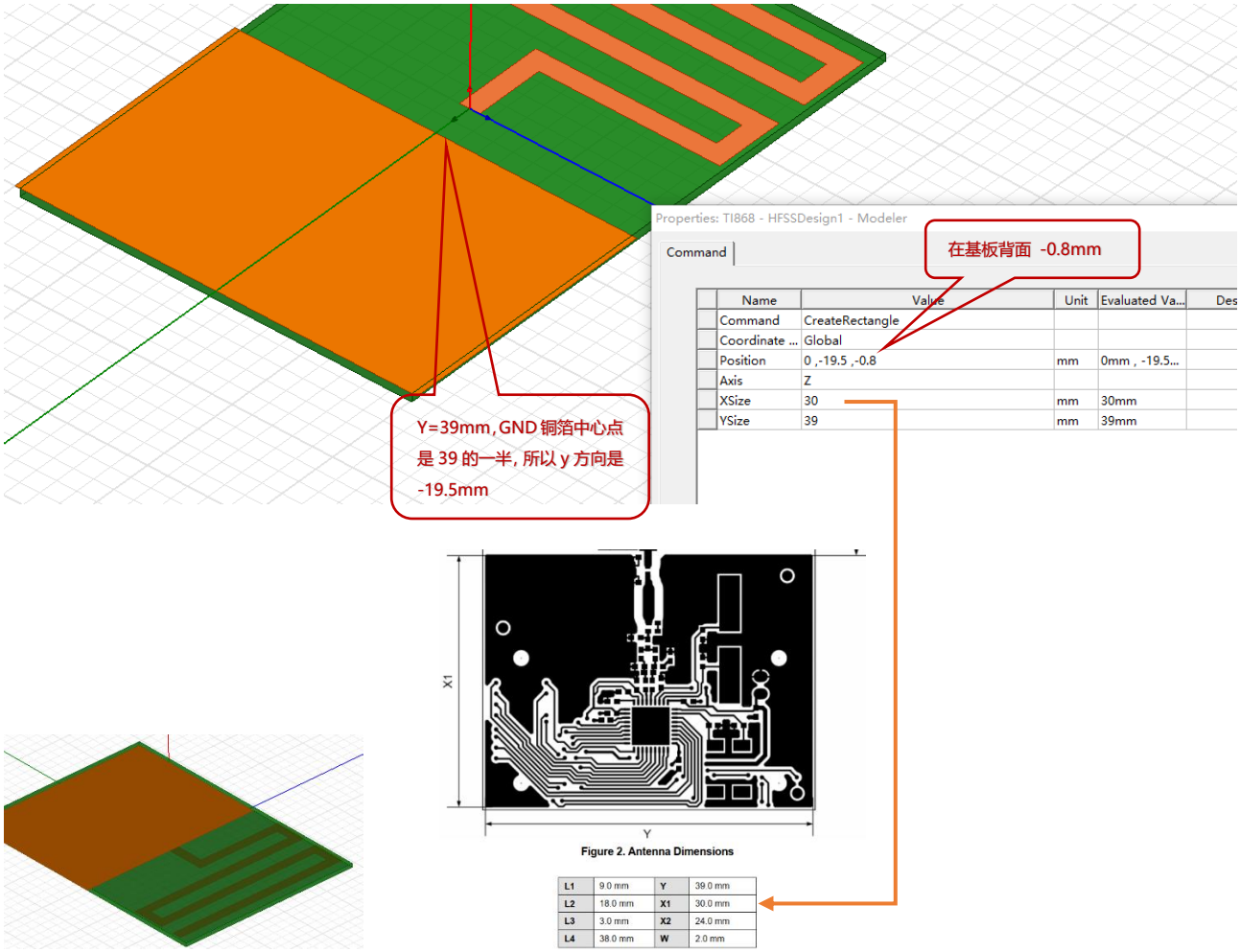


按照前几章讲的软件使用方法，参照 TI 的 PCB 天线尺寸，完成天线模型

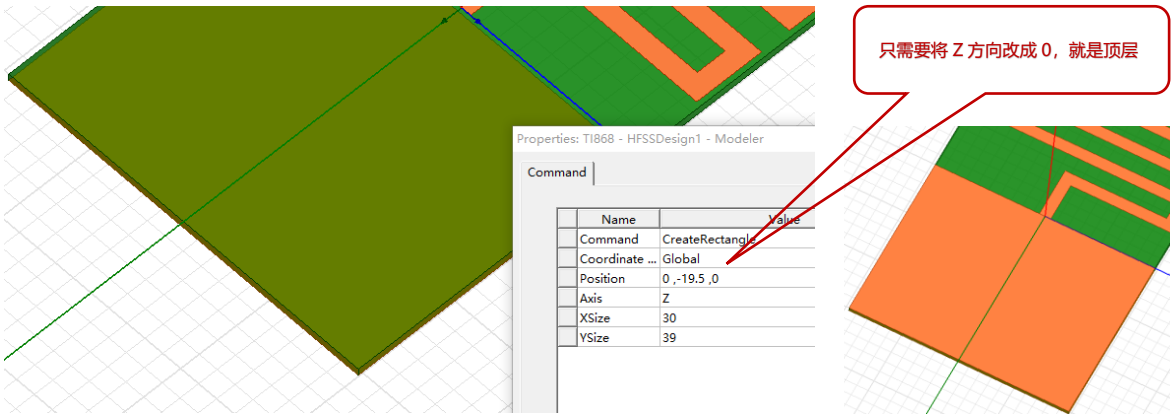


记得合并导线

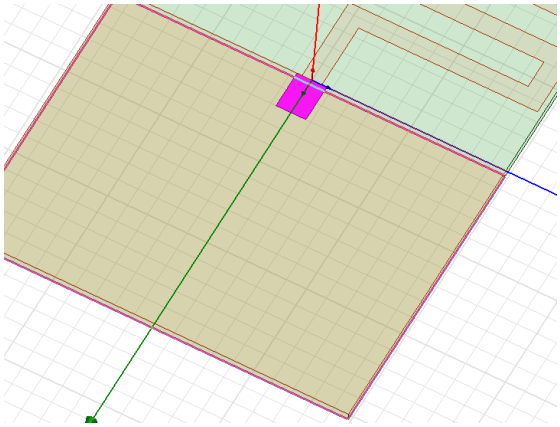
PCB 顶层和底层 GND 铜箔设计

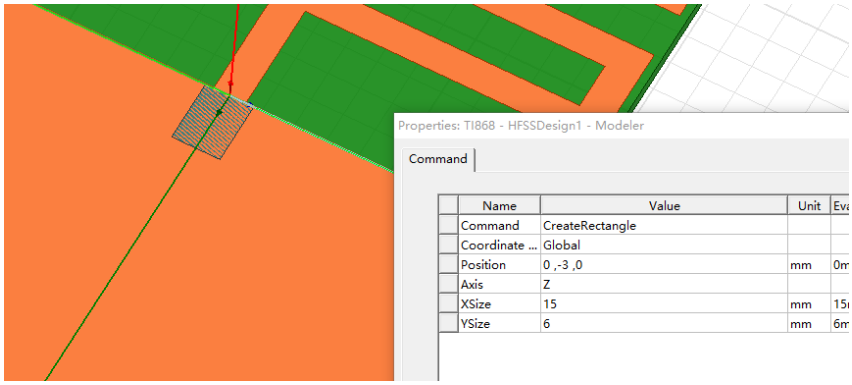


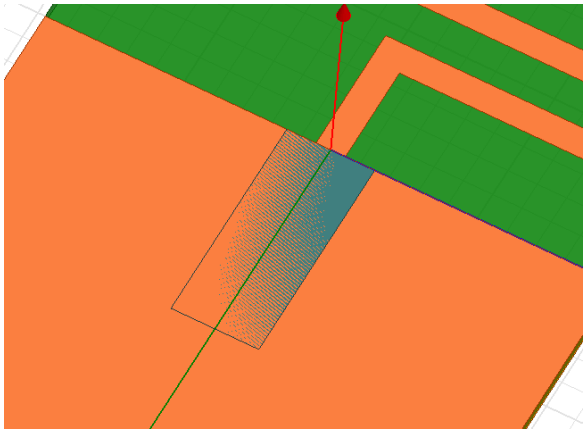
在 PCB 顶层创建 GND

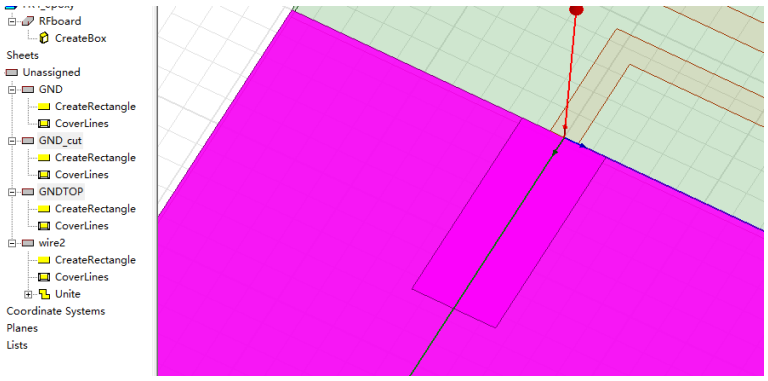


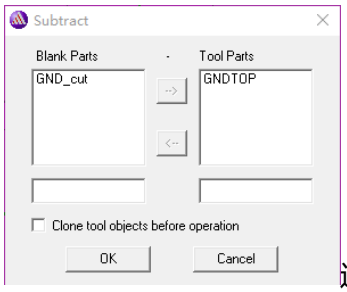
模拟天线到芯片的路径中需要一个阻抗为 50 欧姆的馈线





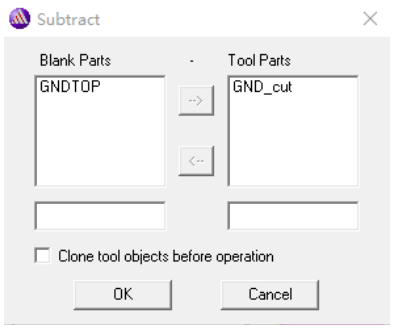


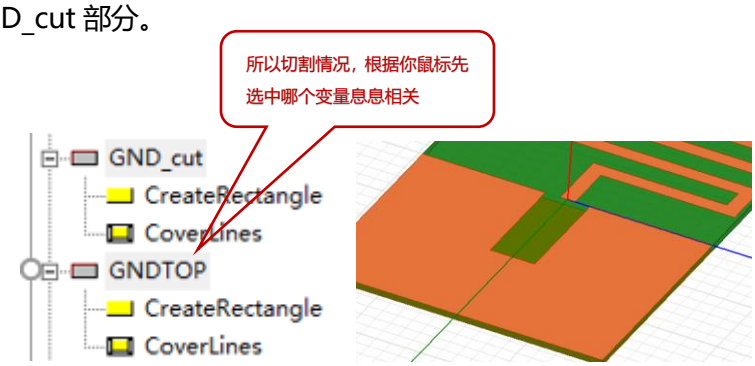




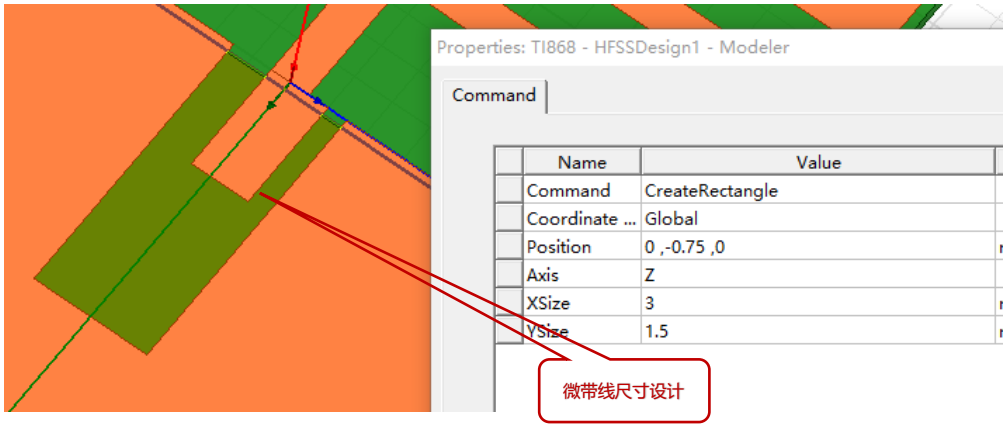
这样是不行的，一定要 GNDTOP 在左，GND_cut 变量在右

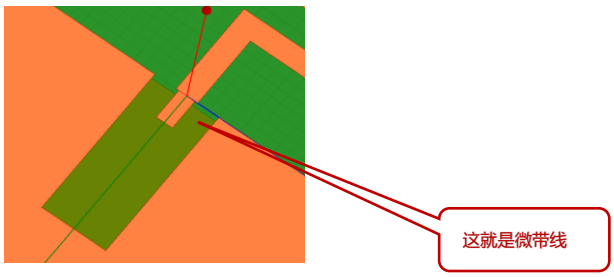
意思就是保留 GNDTOP，减去 GND_cut 部分。



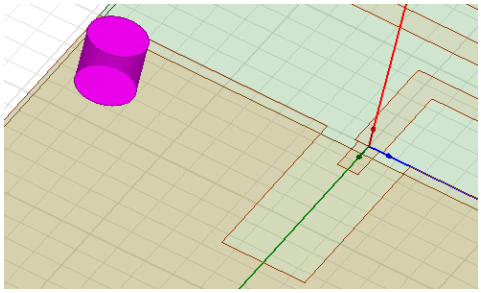


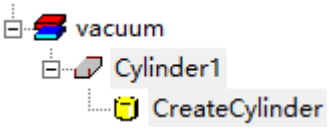
向挖空区域引入微带线



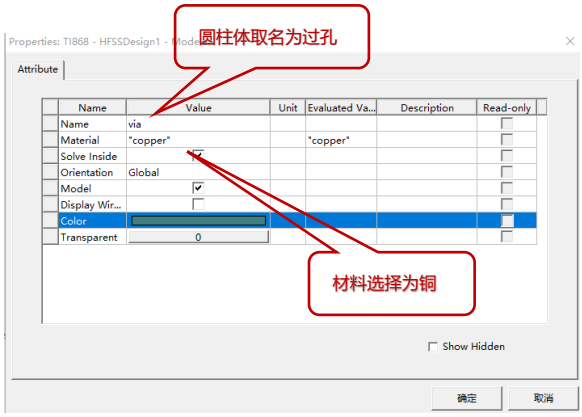


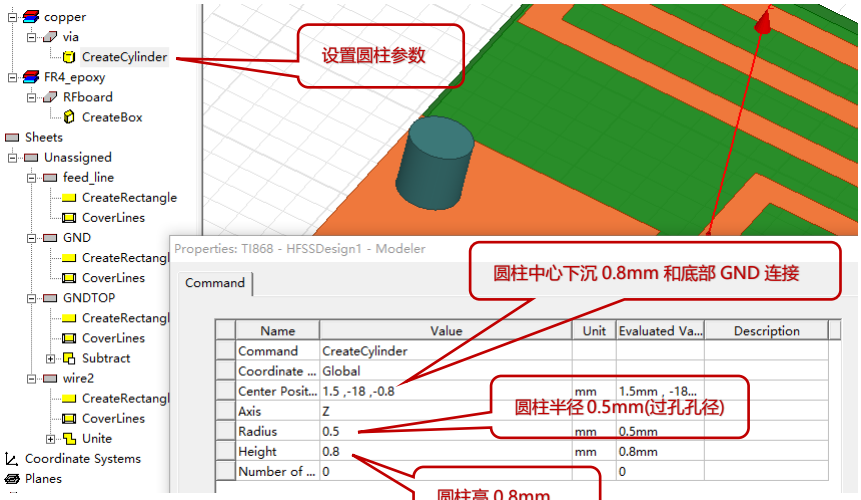
模拟过孔将顶层和底层的 GND 铜箔连接起来

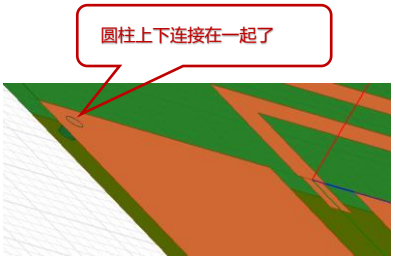




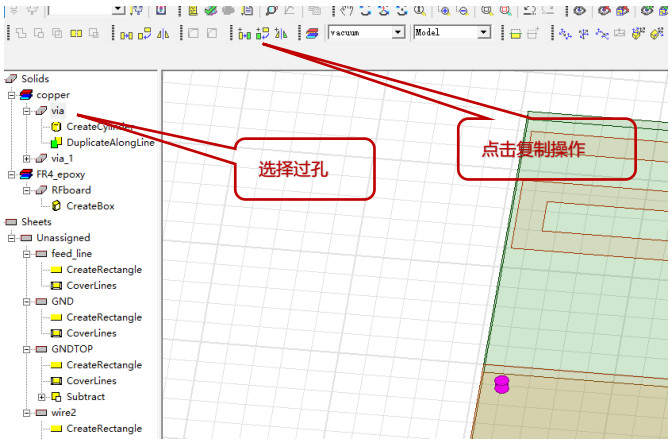
修改圆柱体名称和参数

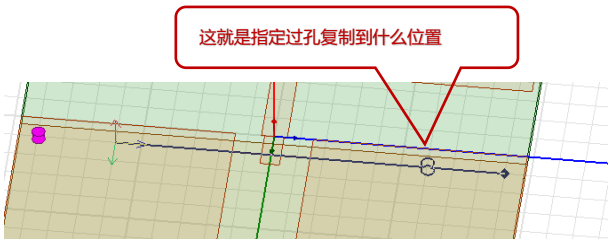




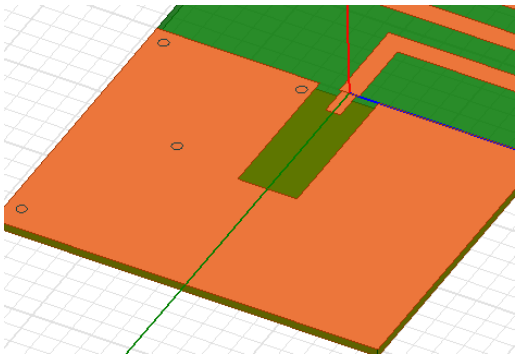


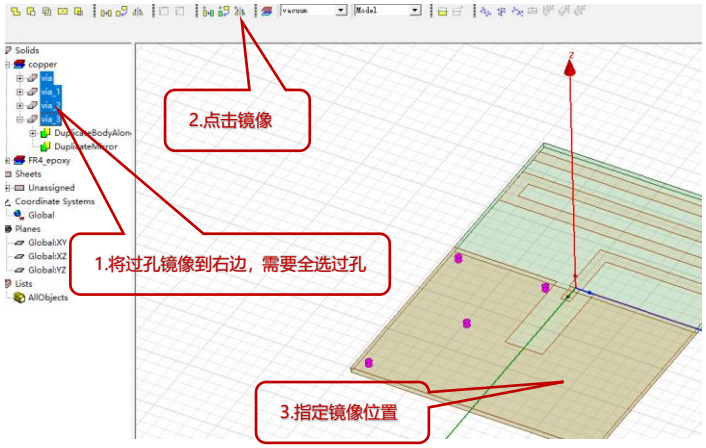
复制多个过孔

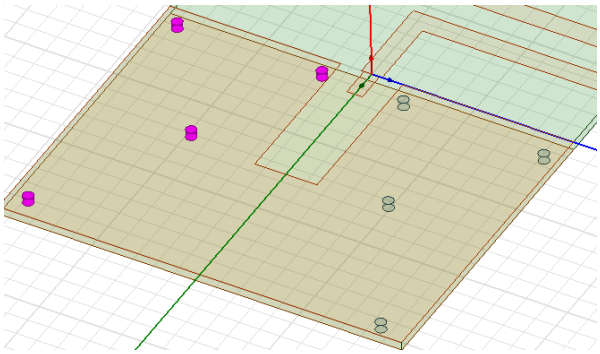


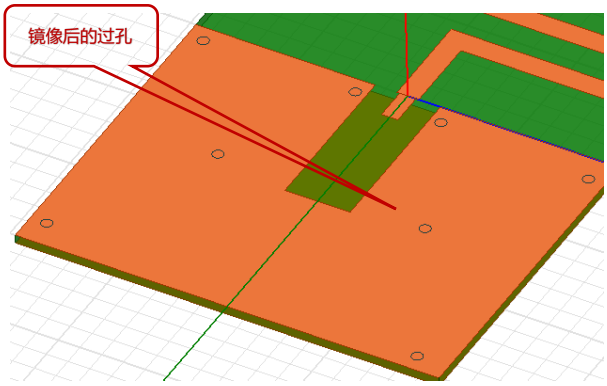


在 PCB 上打一些过孔

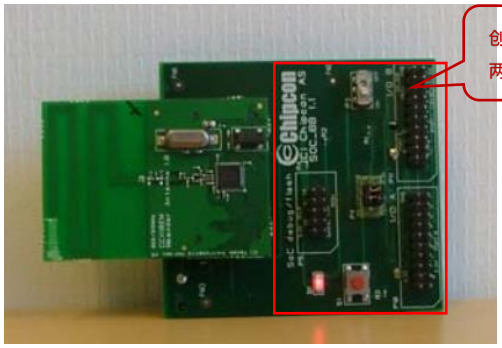


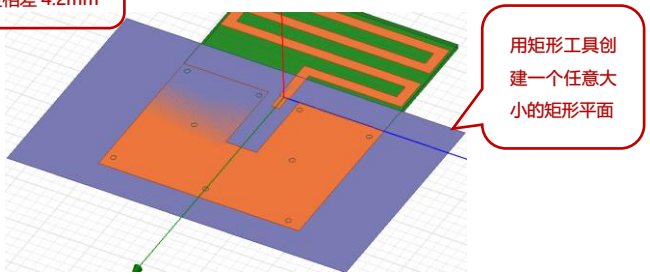


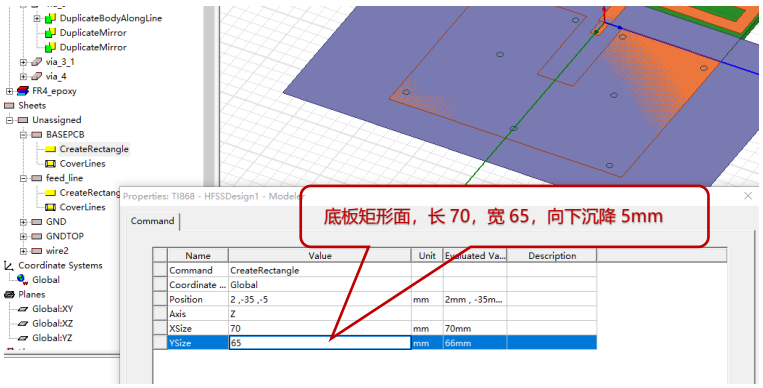


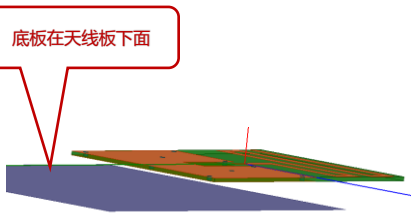


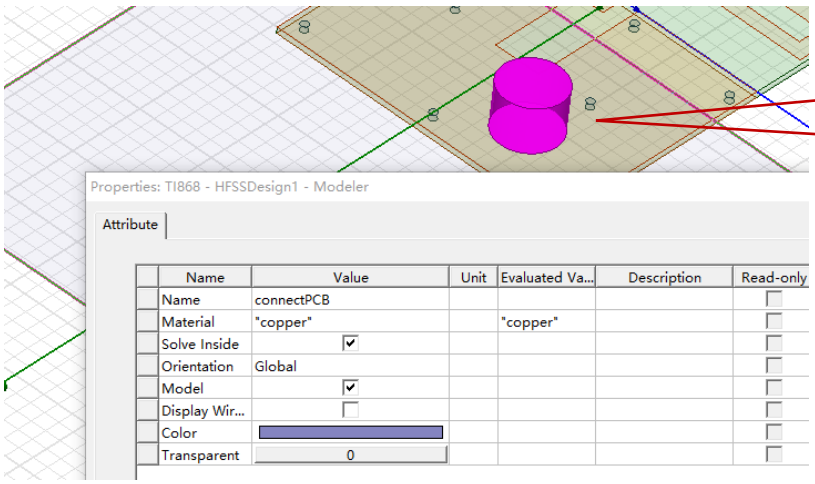
创建底部 PCB 模型



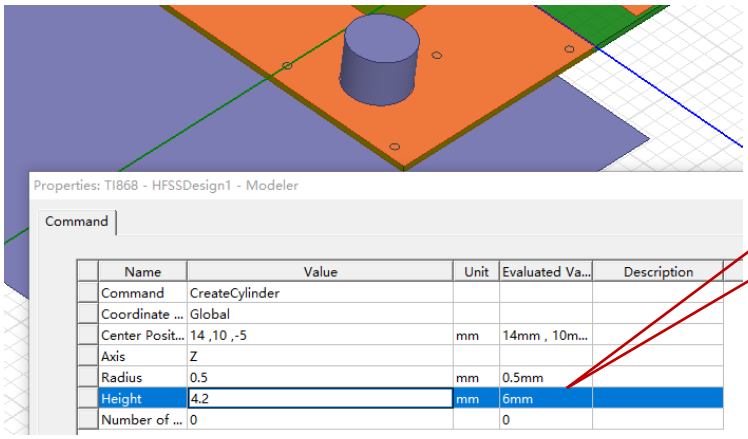




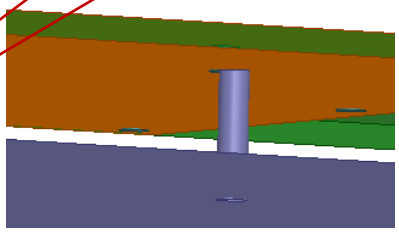




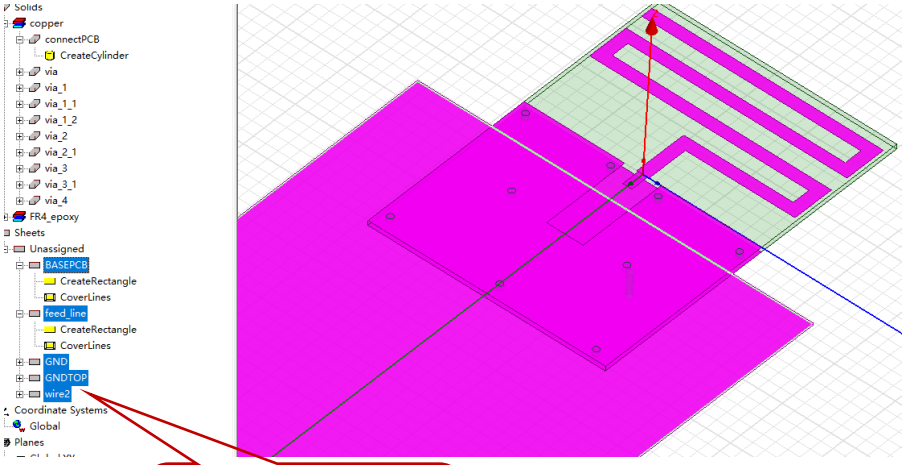
新建一个圆柱体，将底板和天线板连接起来，模拟接插件



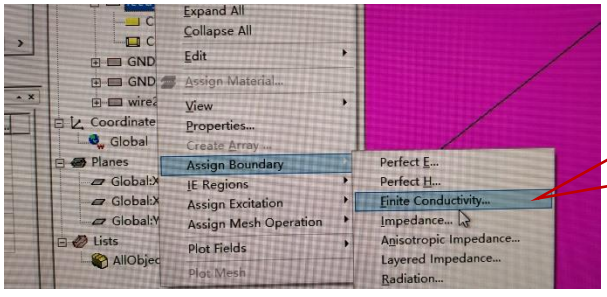
圆柱体先向下沉降 5mm，然后再用 4.2mm 圆柱体高度将上面天线 PCB 连接起来



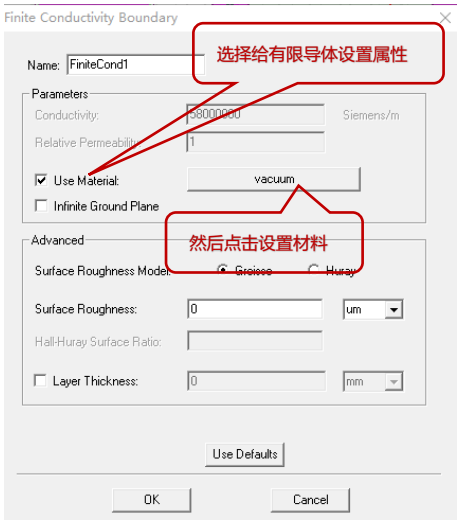
有限电导体设置

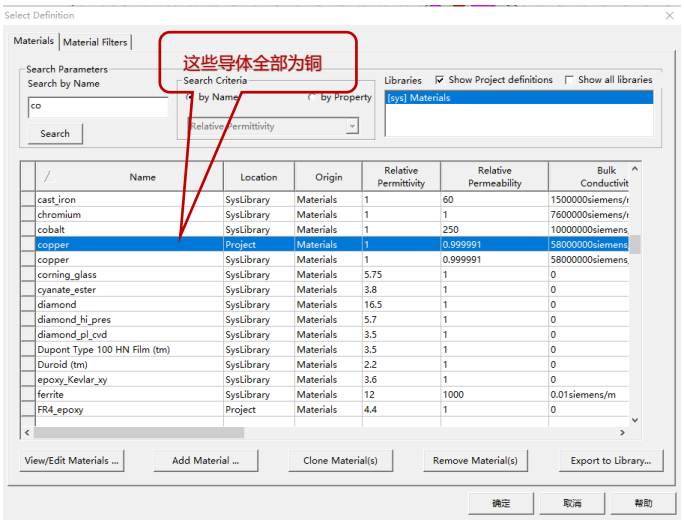


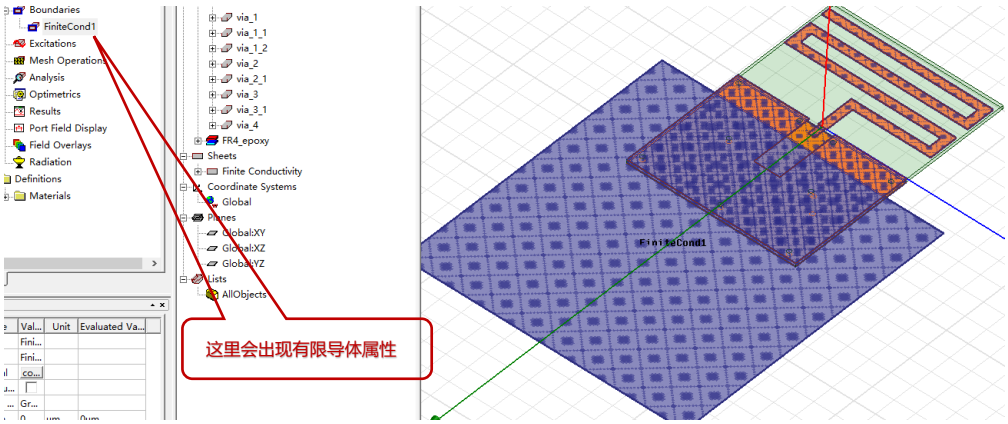
前面章节做的都是理想电导体，现在我们选择这几个 PCB 敷铜的区域，做有限电导体特性



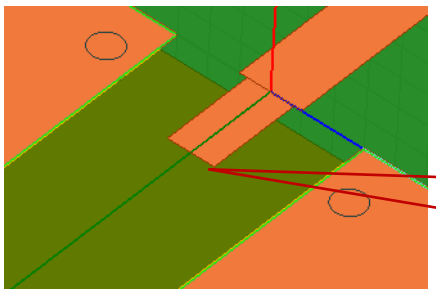
将 BASEPCB,feed_line ，GND，GNDTOP，wire2 都做成有限电导体，右键点击



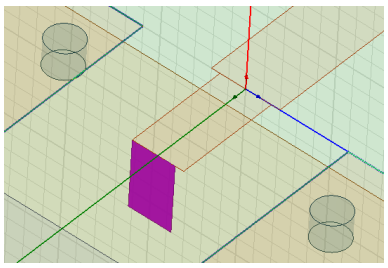


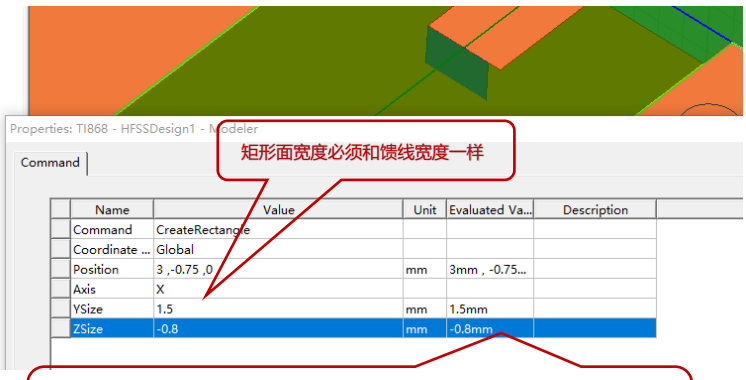


增加激励信号源，模拟射频信号



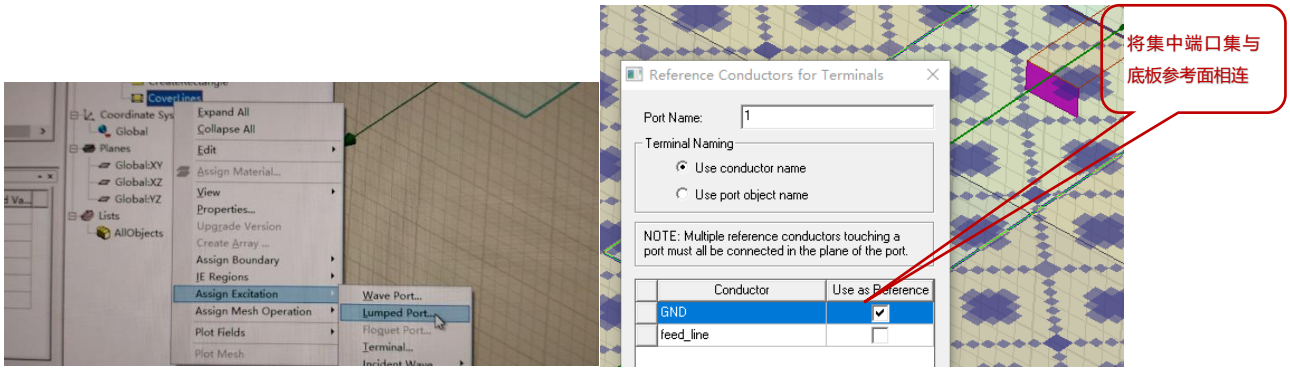
激励信号源是在馈线处增加，需要增加一块矩形，连接天线 PCB 和底板





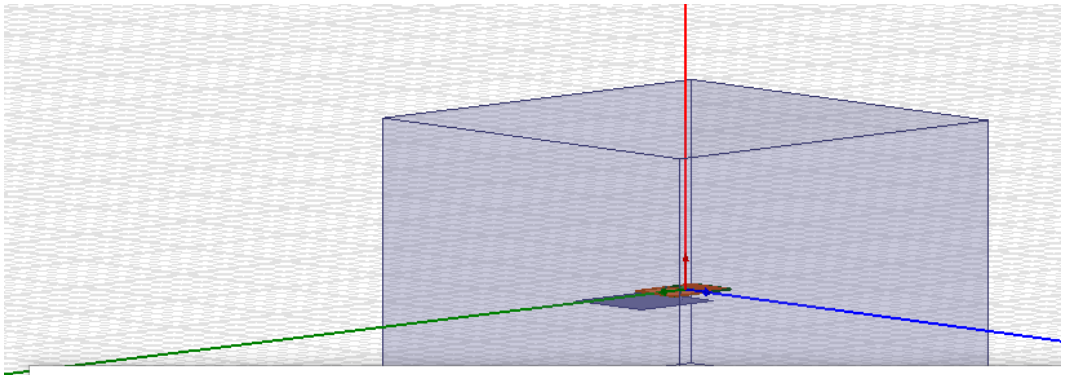
矩形面需要将顶层馈线和底板连接起来，顶层馈线和底板相差 0.8mm，相当于被天线 PCB 基板厚度隔离了 0.8mm，所以靠矩形激励源连接。

设置激励源为集中端口集



增加空气盒

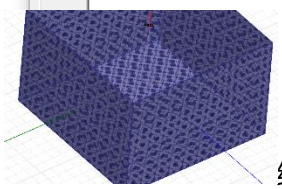
频段 868M 对应 1/4 波长为 86mm，空气盒与 PCB 的间距必须大于 86mm



Properties: TI868 - HFSSDesign1 - Modeler

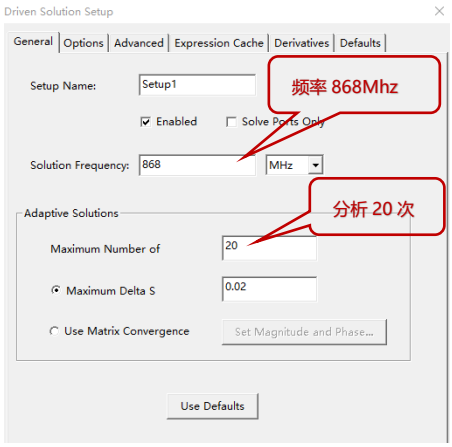
Command

Name	Value	Unit	Evaluated Va...	Description
Command	CreateBox			
Coordinate ...	Global			
Position	-150 ,-150 ,-80	mm	-150mm , -1...	
XSize	300	mm	300mm	
YSize	300	mm	300mm	
ZSize	200	mm	200mm	

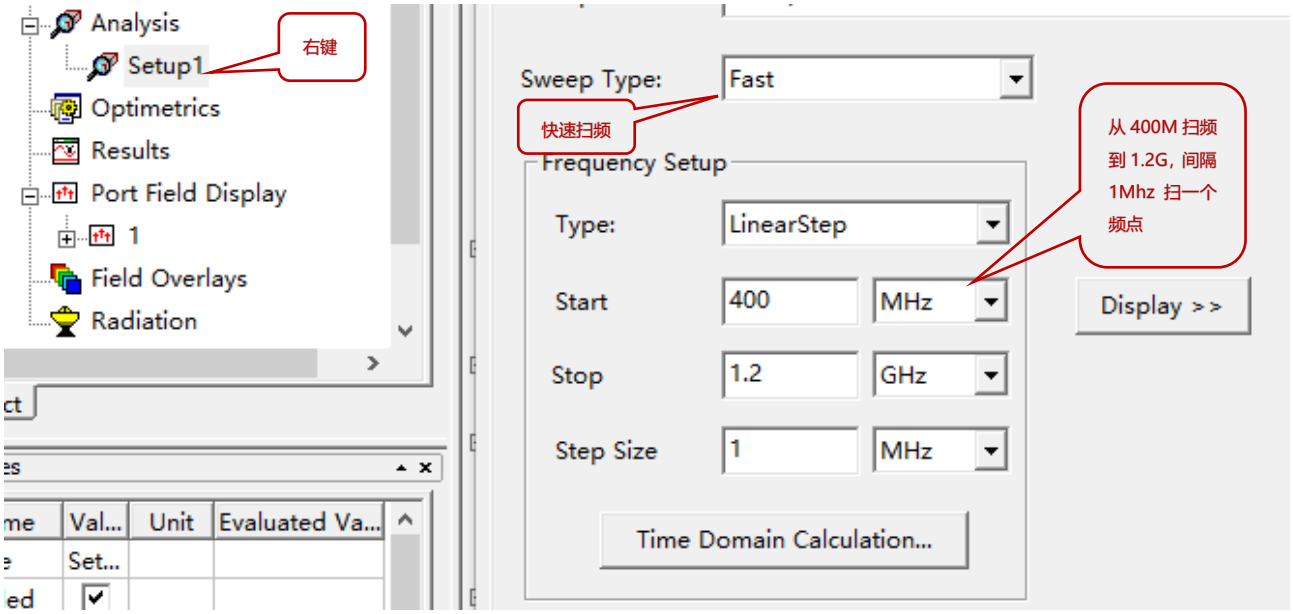


给空气盒增加辐射边界条件

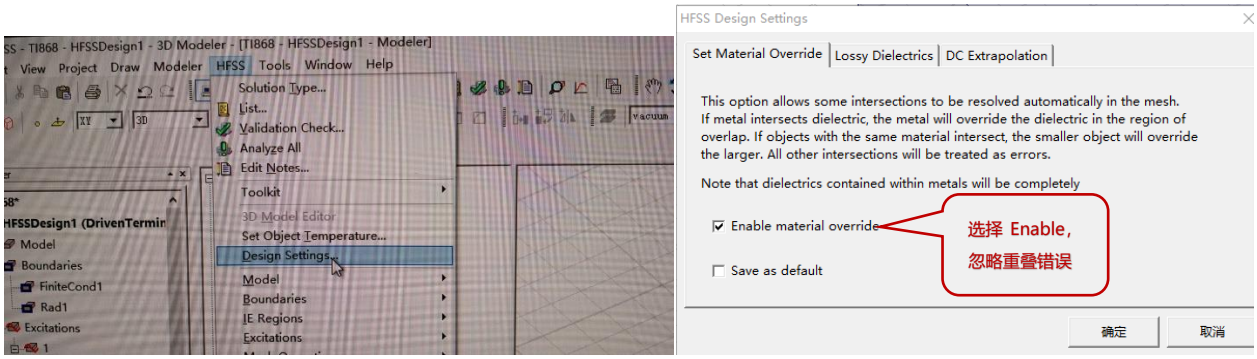
打开 Analysis 设置求解项



设置扫频设置项

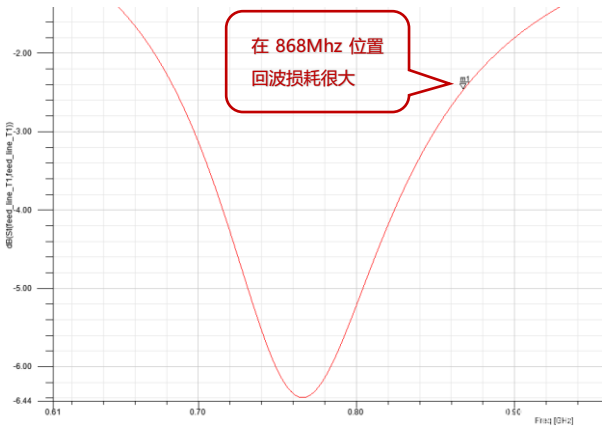


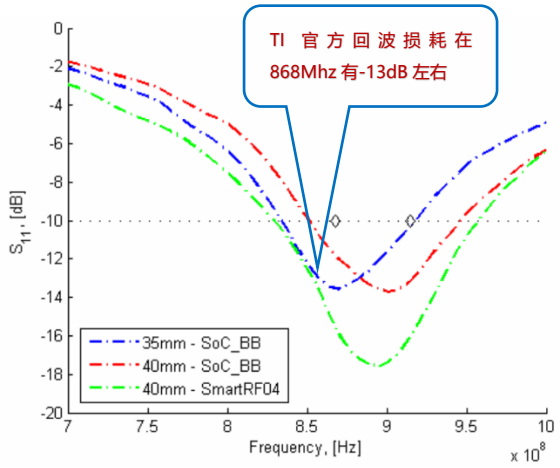
在建模中，过孔和天线表层的铜皮是重叠的，仿真软件很有可能认为是错误，我们要忽略这个错误



开始运行仿真

回波损耗检测



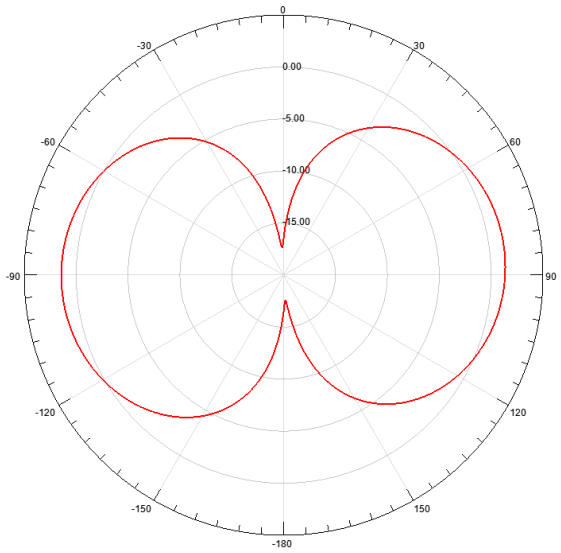


我这个-2dB 回波损耗相差 TI -13dB 太远了，不合格

方向图检测

查看 XY 平面方向

XY 平面方向图



TI 官方的 XY 平面放线图

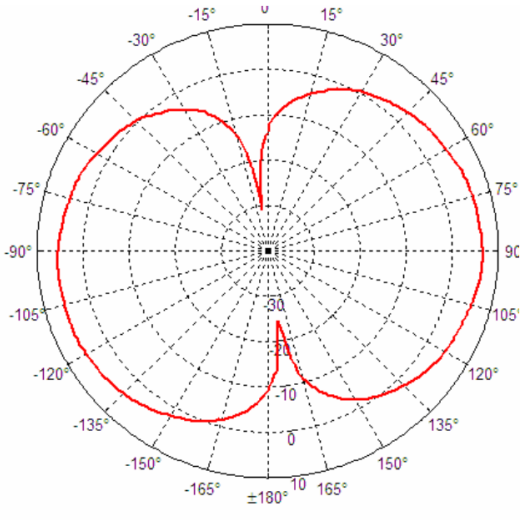
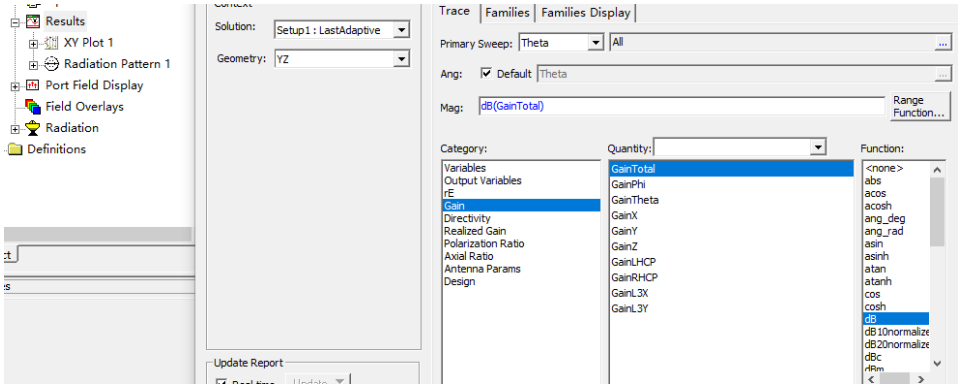
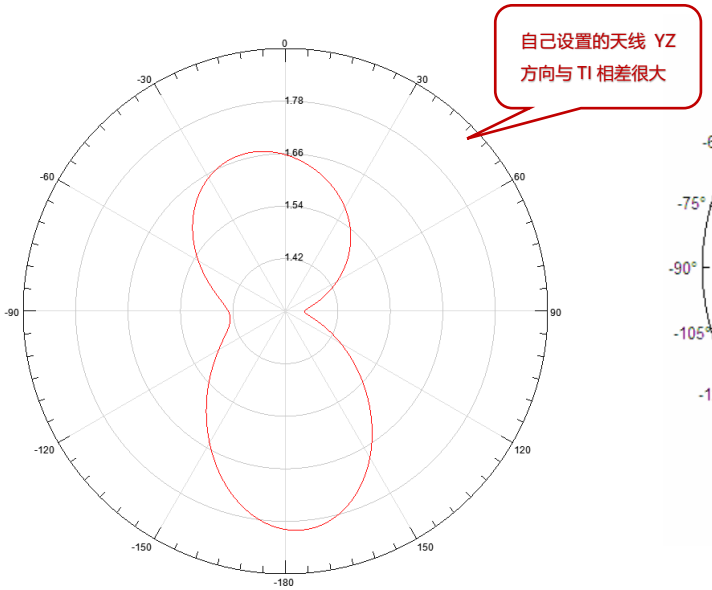


Figure 4. XY Plane, Horizontal Polarization

查看 YZ 方向



YZ 平面方向图



TI 官方 YZ 平面方向图

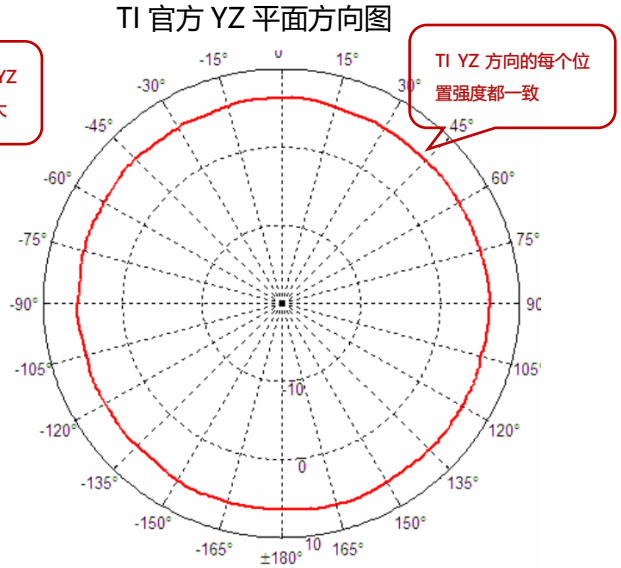


Figure 7. YZ Plane, Vertical Polarization

自己设置的天线，YZ 平面 在每个方向都有不同的衰减，其实这是因为我们设置的 YZ 方向图坐标精度造成的，而不是天线本身的问题。

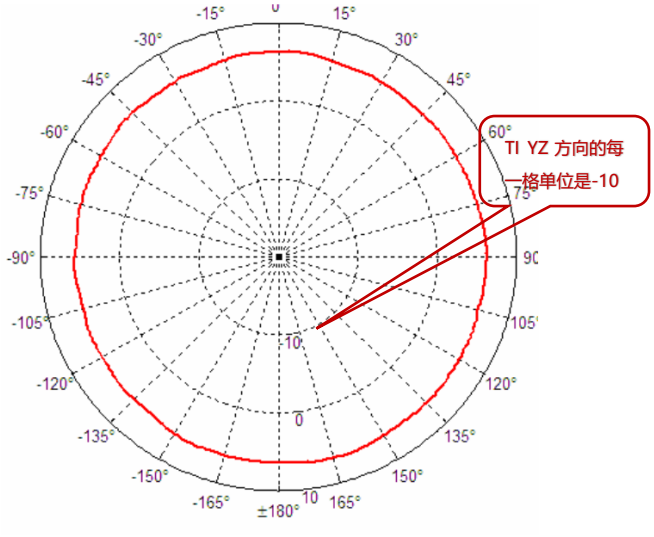
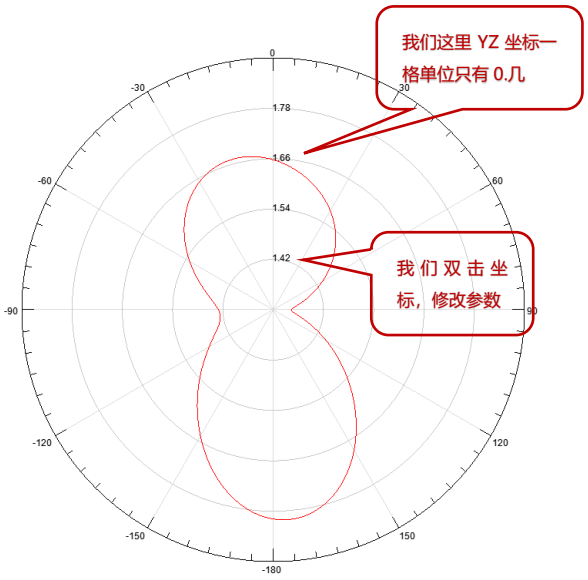
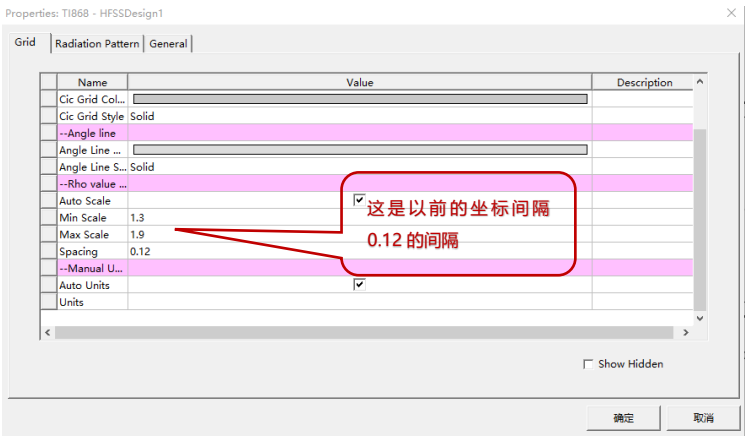
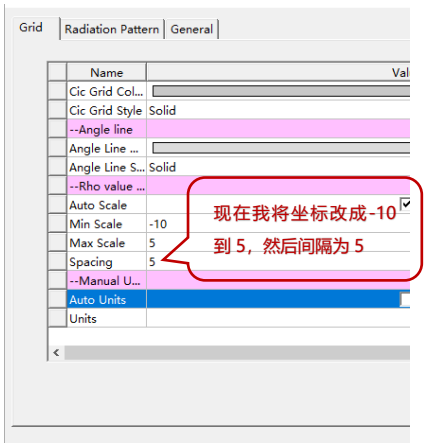
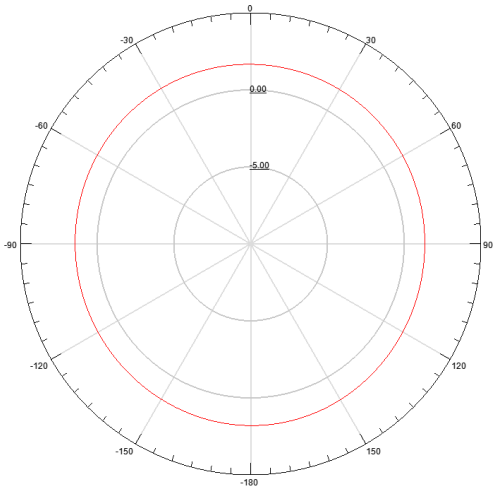


Figure 7. YZ Plane, Vertical Polarization





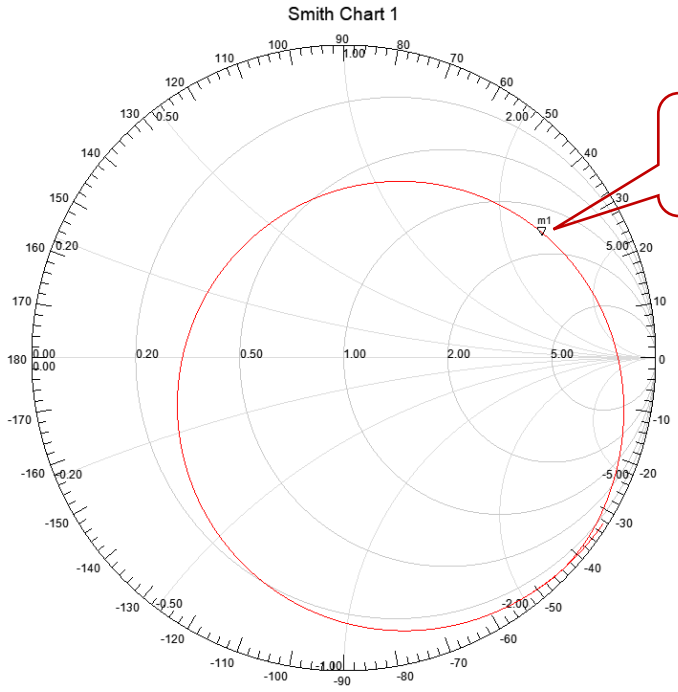


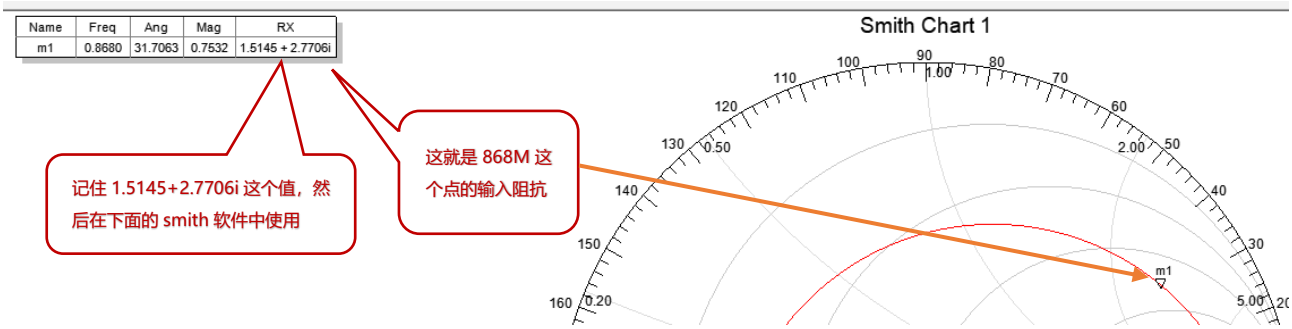


你看 方向图和 TI 官方就差不多了。

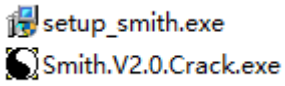
那么 868M PCB 天线只有回波损耗和 TI 官方的相差很多，我们主要优化回波损耗。

优化回波损耗其实就是优化阻抗匹配，下面我们看看 868M PCB 天线的史密斯圆图

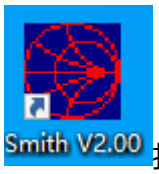




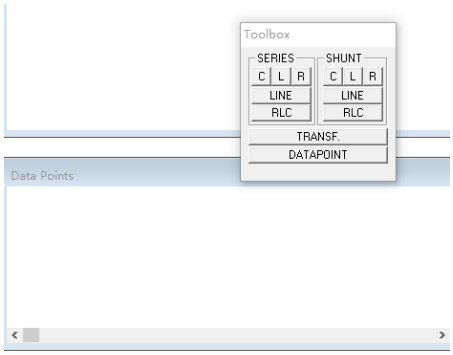
Smith 工具优化阻抗匹配，改善回波损耗



安装 smith 软件



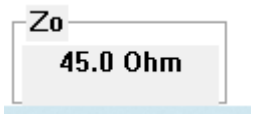
打开 smith 软件

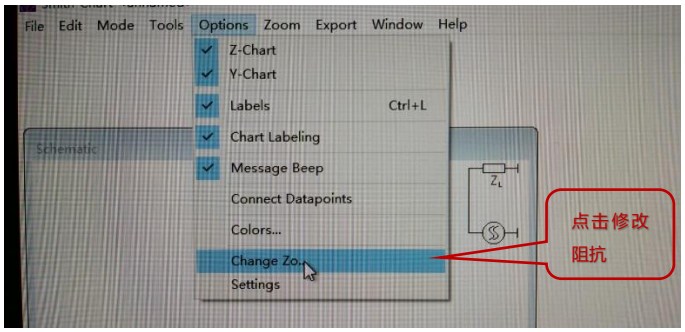


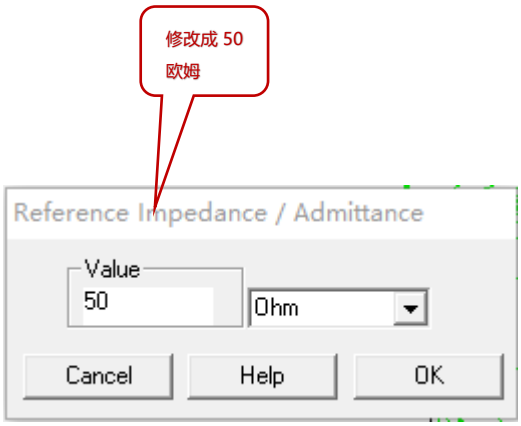
因为smith 软件是通过归一化之后进行计算
所以HFSS得到的特征阻抗值要先 乘以50欧姆

(1.5145 x 50) + (2.7706 x50)i
= 75.725 + 138.53i 这个值就是要填入smith的值

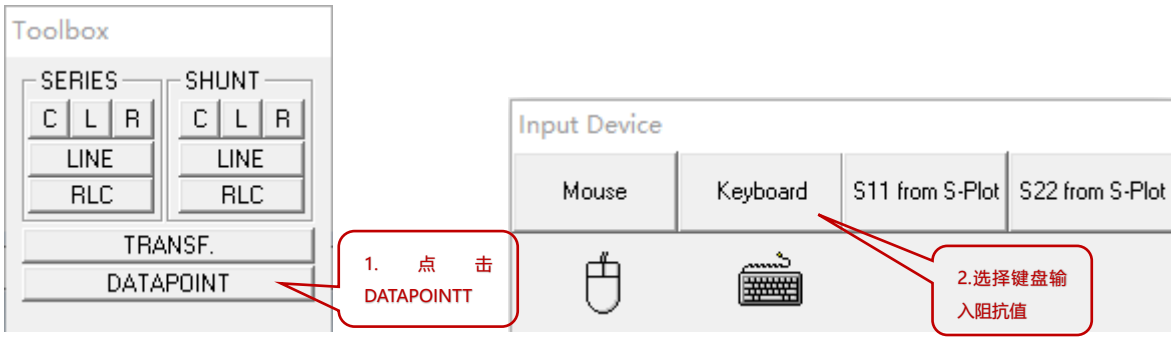
如果 Zo 显示得不是 50 欧姆

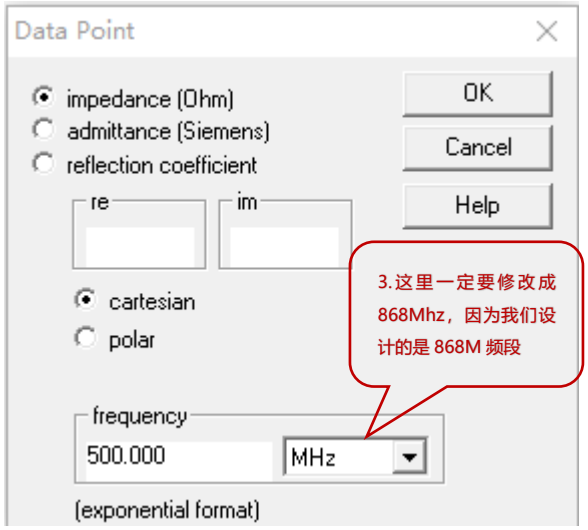


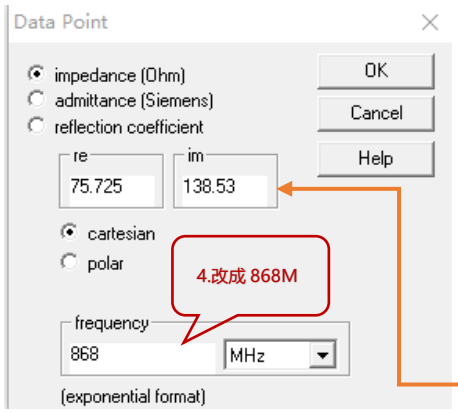




填入阻抗参数



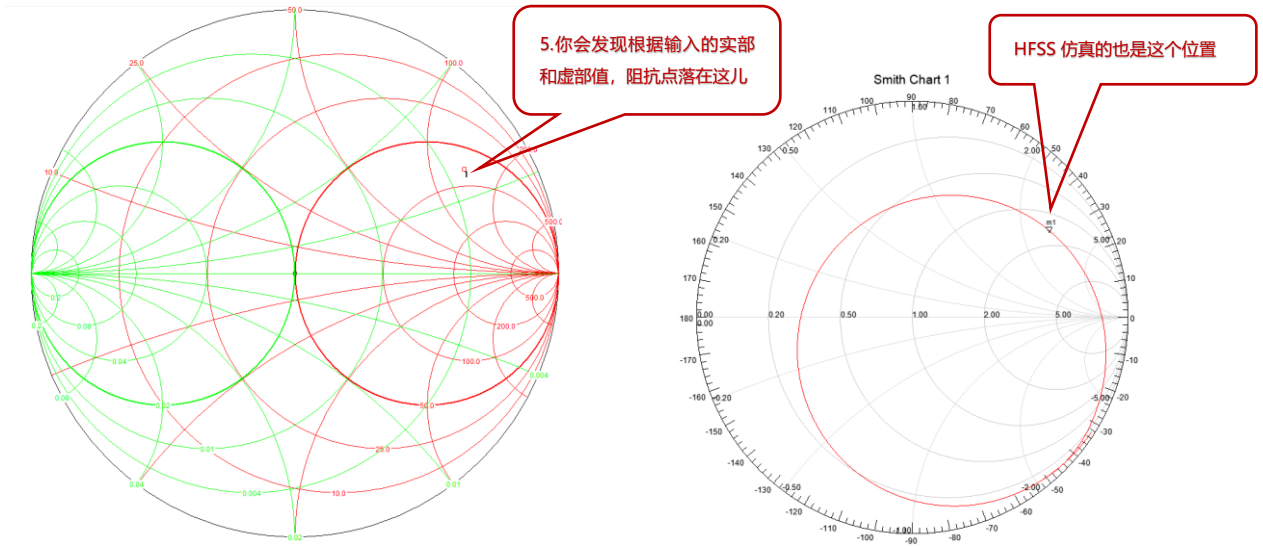




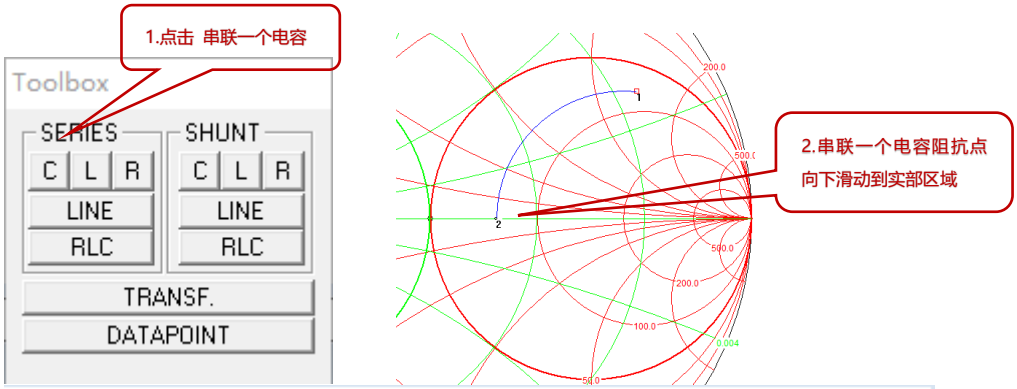
(1.5145 x 50) + (2.7706 x50)i

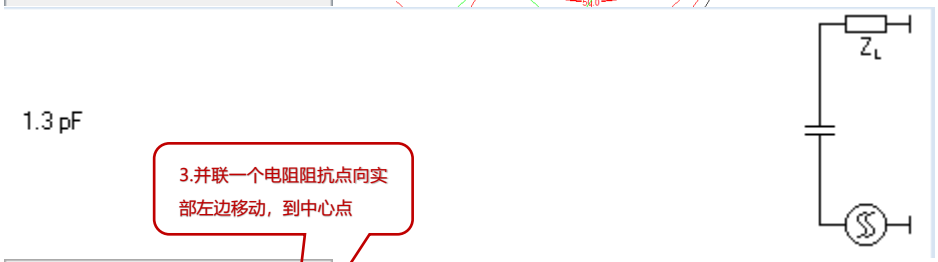
= 75.725 + 138.53i 这个值就是要填入smith的值

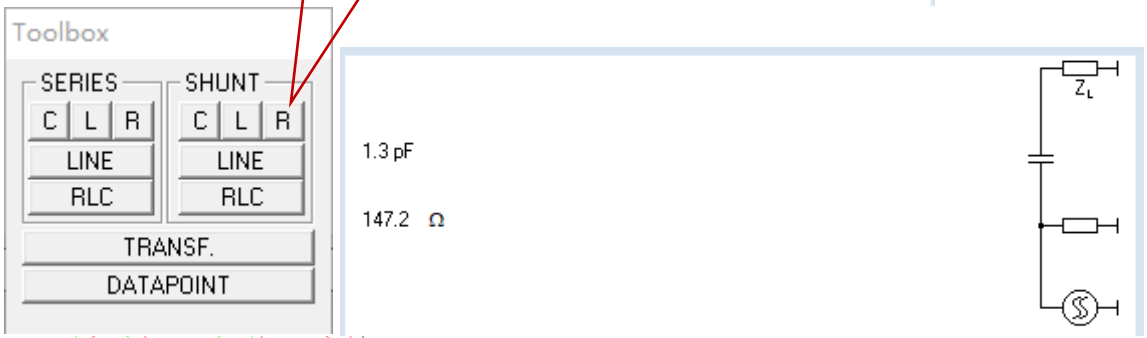
点击确定

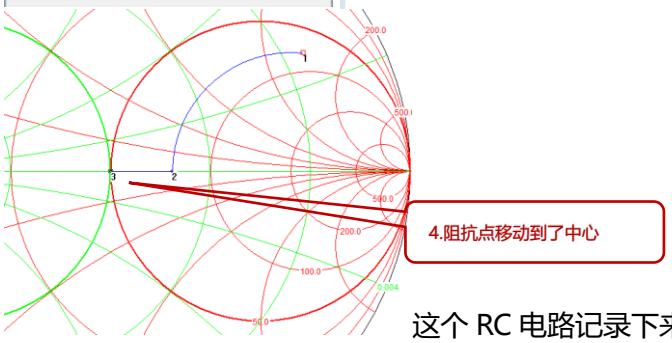


如果将阻抗点移动到中间呢?



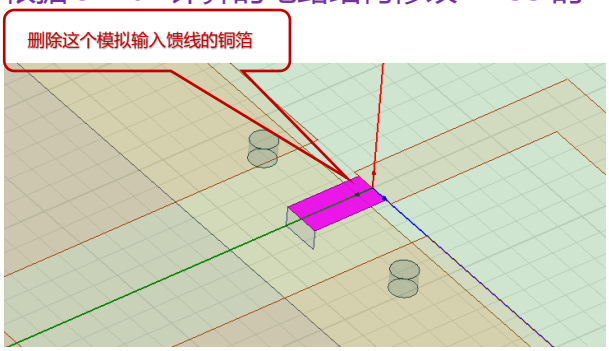


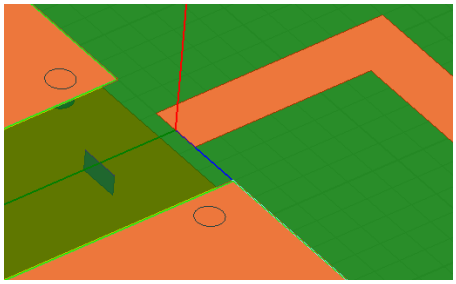




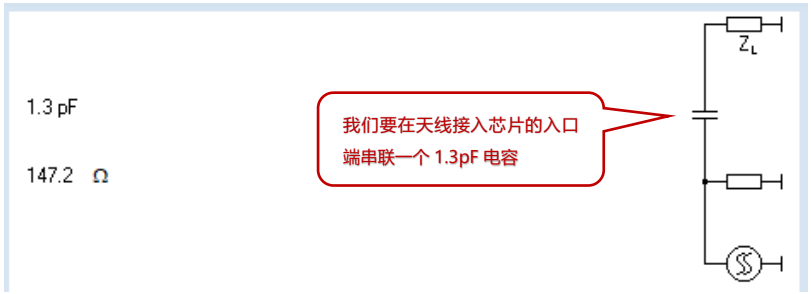
这个 RC 电路记录下来，放到 HFSS 去仿真，或者放在电路板修改。

根据 smith 计算的电路结构修改 HFSS 的 PCB 模型

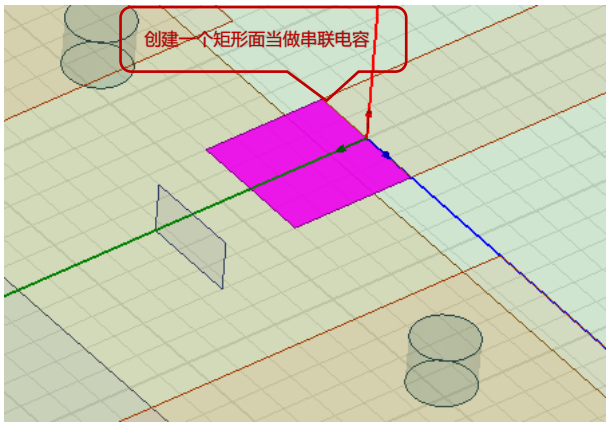


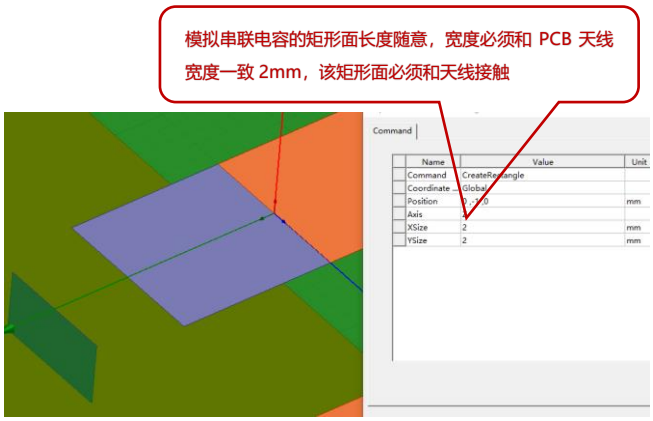


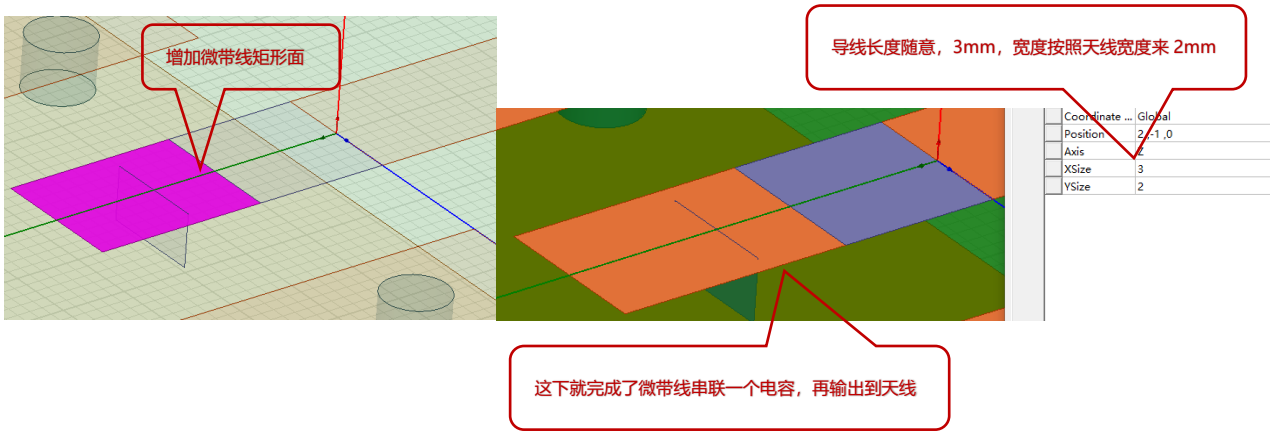
根据电路



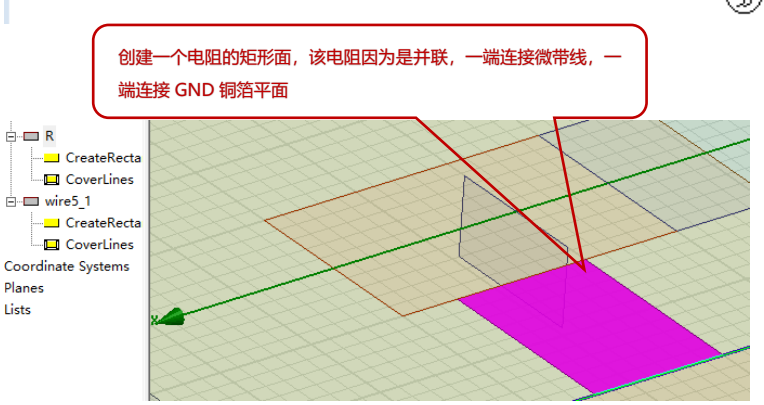
因为 HFSS 软件本身没有电阻电容这些元器件，我们使用矩形模拟铜箔，HFSS 的铜箔材料有寄生参数设置，这些寄生参数就是电阻，电容，电感。很多材料都有电阻，电容，电感这些寄生参数，也称为边界条件。



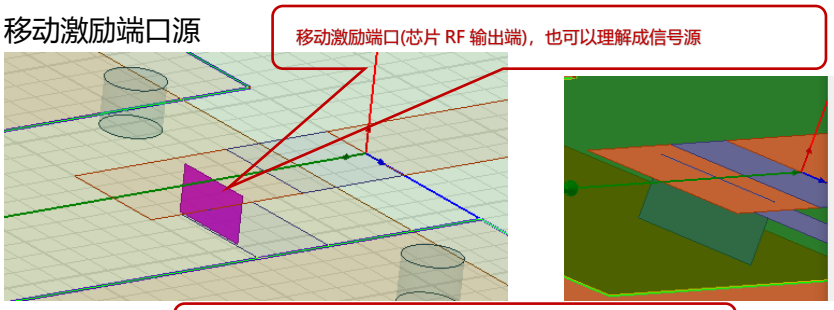


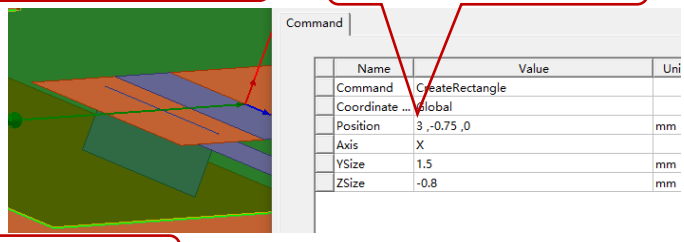


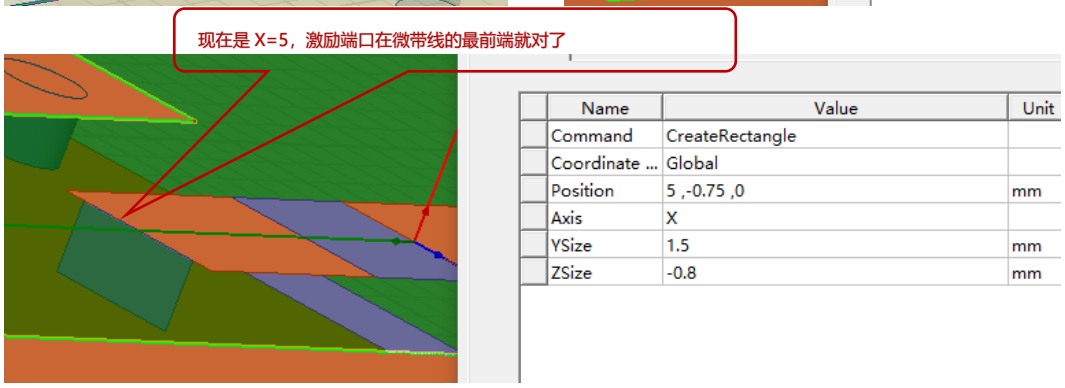
并联电阻

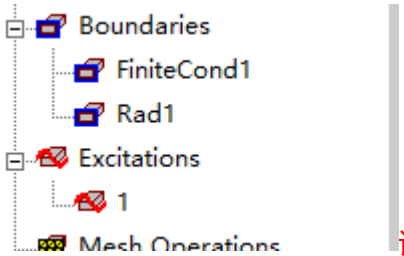


移动激励端口源



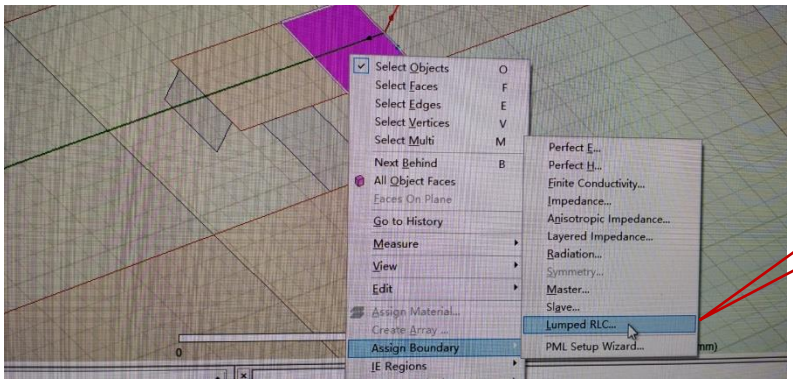


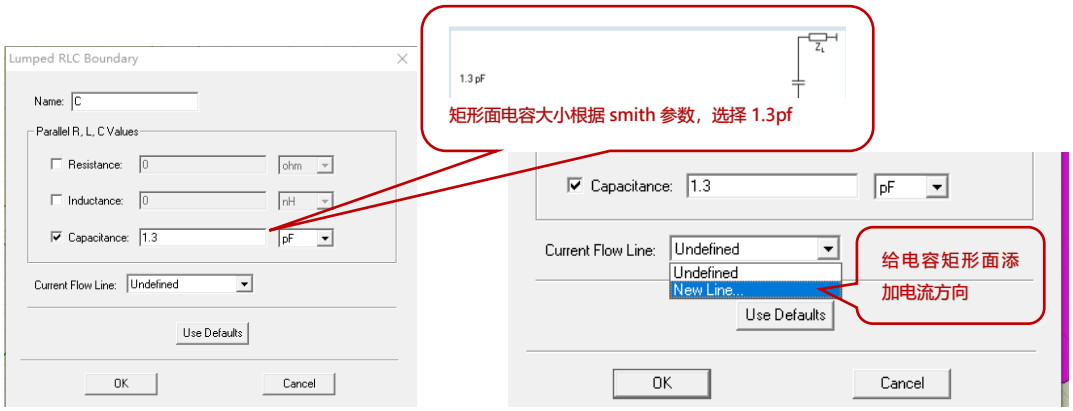


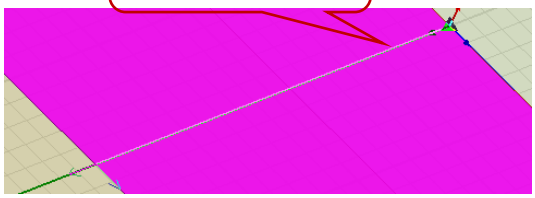


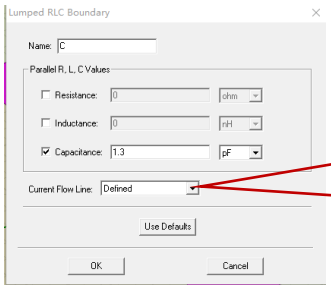
记住，取消删除以前的激励端口设置，然重新设置激励端口

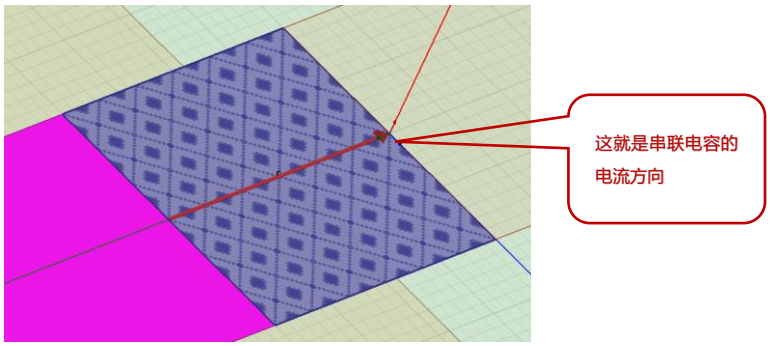
现在给电容矩形面，和电阻矩形面，加入寄生(边界)条件，这两个矩形面才具备电路特性



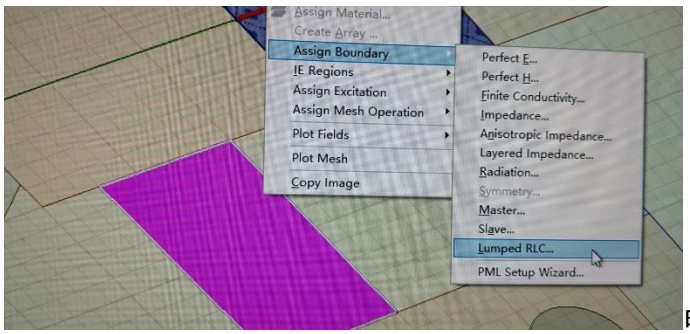




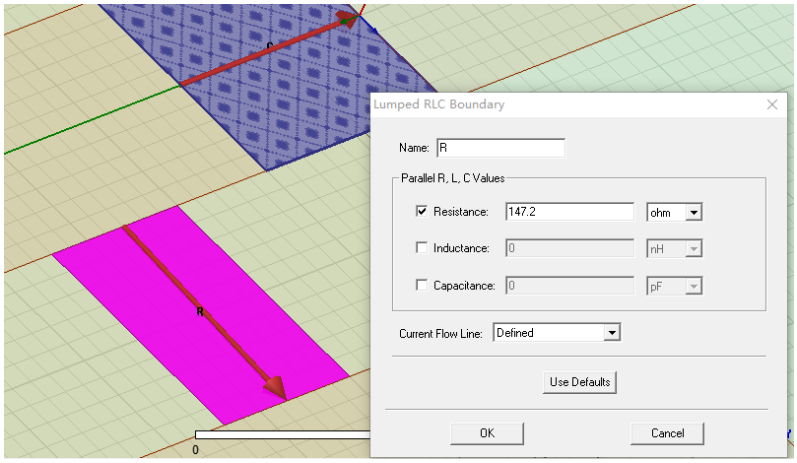




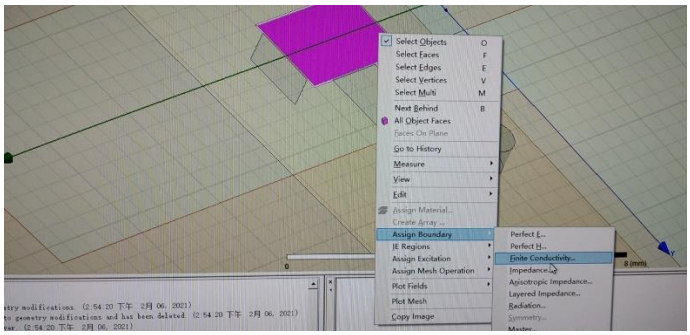
下面设置并联电阻也是同理，只是电流方向，向 GND 平面方向设置。

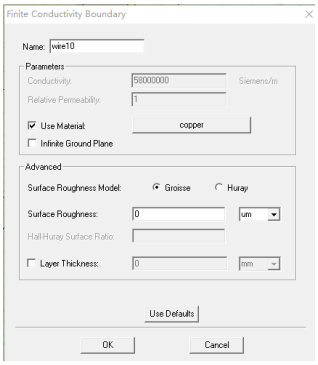


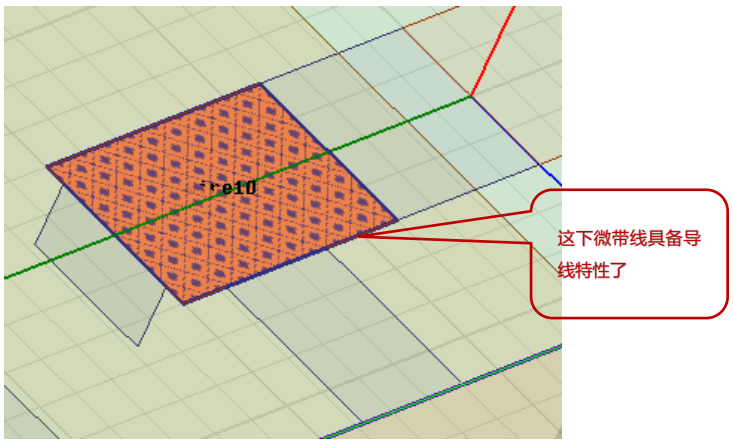
电阻矩形面寄生参数(边界条件)设置



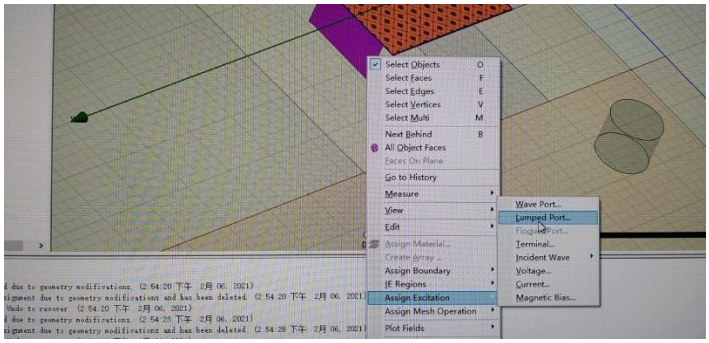
下面设置微带线为有限导体

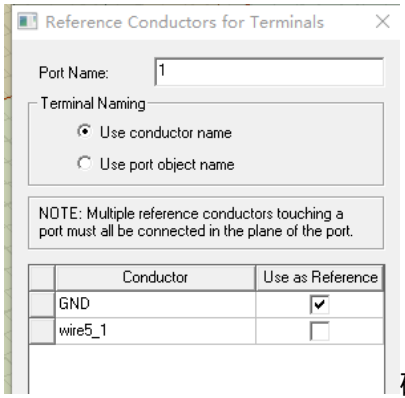




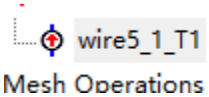


将删除的集中端口集，重新设置上





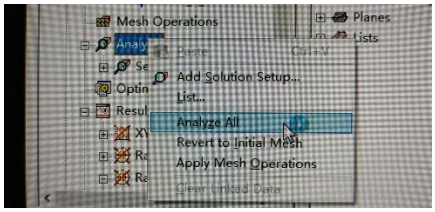
确保集中端口集和 GND 连接



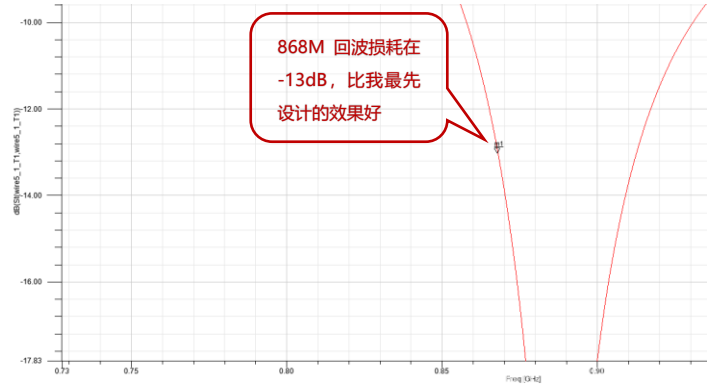
确保集中端口集输出 50 欧姆，也可以理解成激励源(信号源)输出阻抗 50 欧姆。

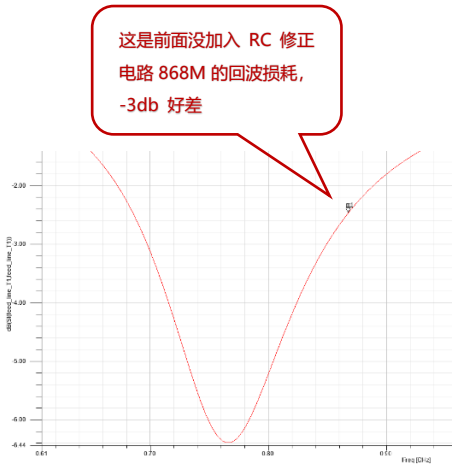
到这里我们所有参数设置完毕。

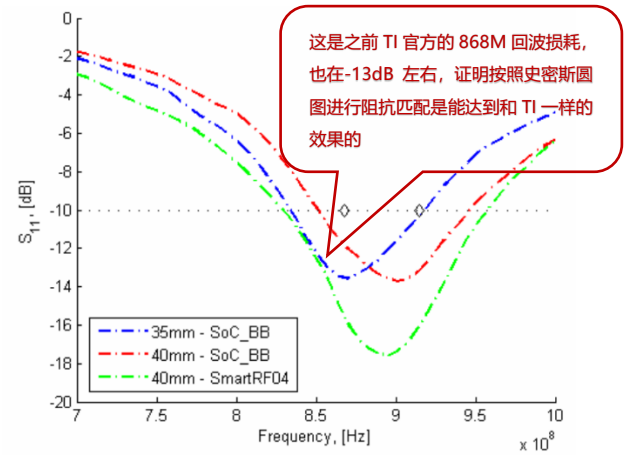
重新运行仿真分析



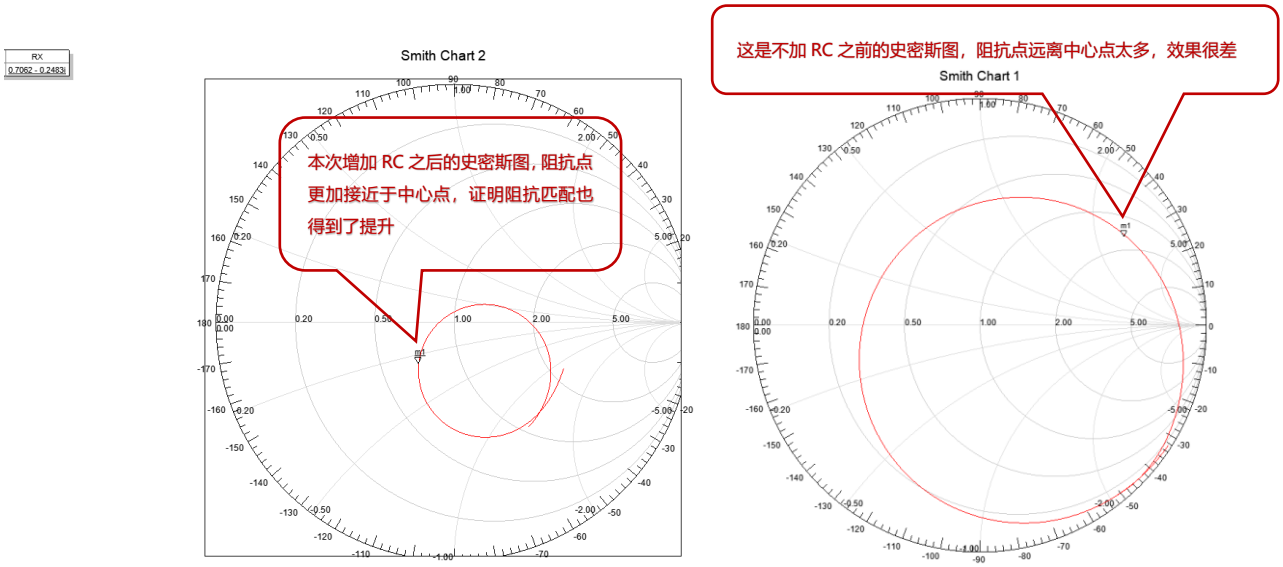
查看仿真结果







回波损耗负得越多，证明性能越高，就有更多的能量传输到微带线和天线上面去。效果更好。



如果阻抗点完完全全在史密斯中心点 1.00，那就堪称绝对完美。实际是不可能这么理想的。

2.4G 和 5.8G WLAN wifi 双频 PCB 天线设计

Dual-band microstrip-fed monopole on RO4003 substrate

双频天线主要参考这篇文献

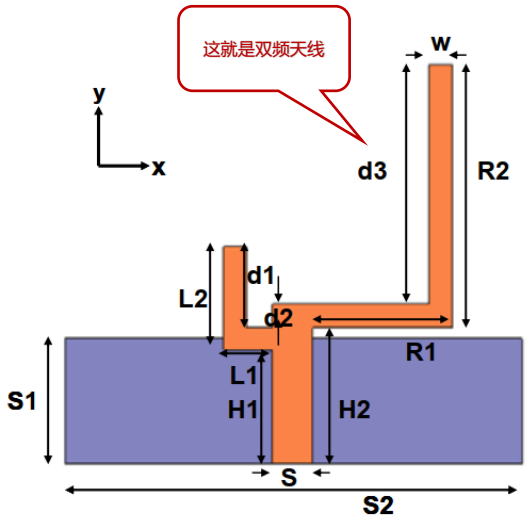


Fig. 1: Antenna Schematic

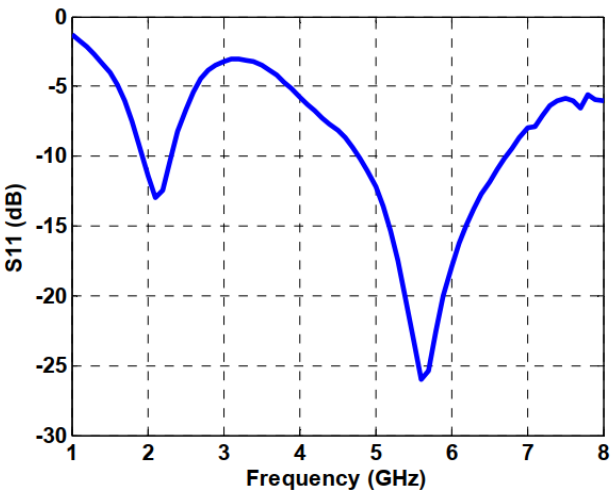
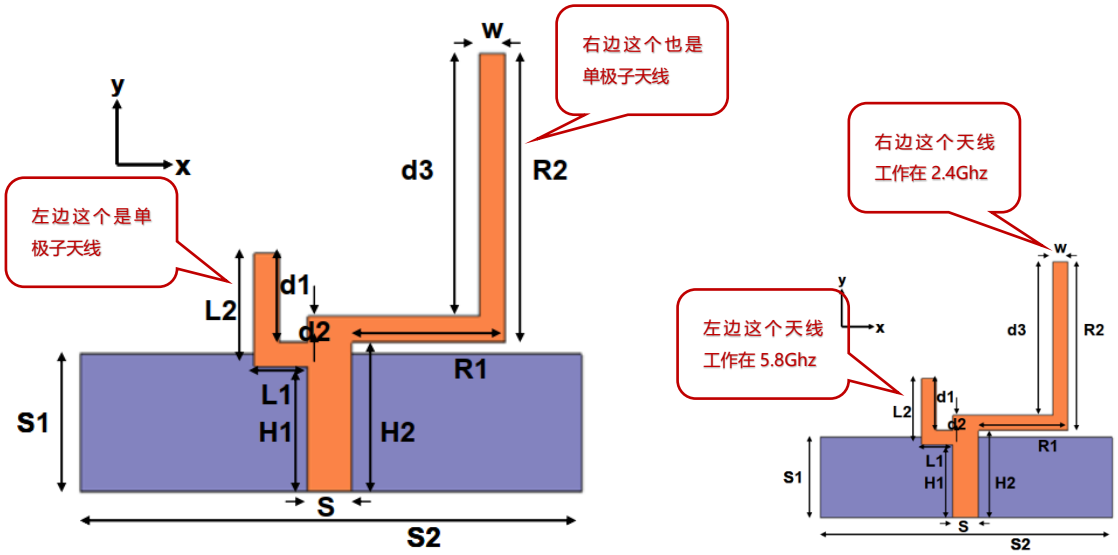


Fig. 2: Return Loss S11

Table I: Dual-monopole antenna dimensions

H1	10.00 mm	S	3.50 mm
H2	12.00 mm	w	2.00 mm
R1	12.25 mm	d1	7.00 mm
R2	23.00 mm	d2	2.00 mm
L1	4.25 mm	d3	21.00 mm
L2	9.00 mm	S1	11.00 mm
		S2	40.00 mm

这个双频天线由两个单极子天线组成，如下:



所以频段越高天线越短，频段越低天线越长。

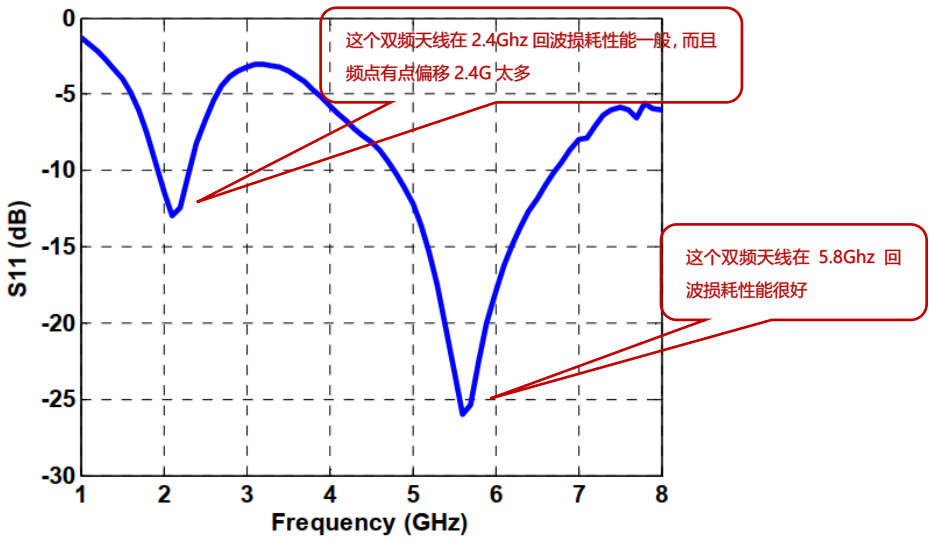


Fig. 2: Return Loss S11

所以这个天线的低频段的天线长度得调整一下

降低，低频段天线长度可以有效改善天线向 2.4G 频段靠拢。

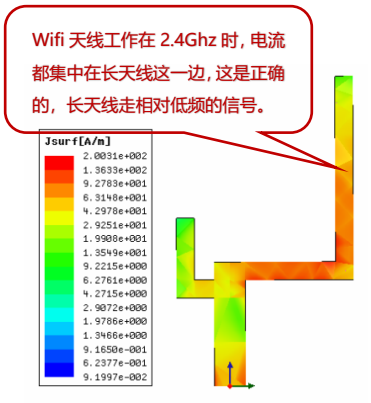


Fig. 3a: Surface current distribution at 2.3 GHz

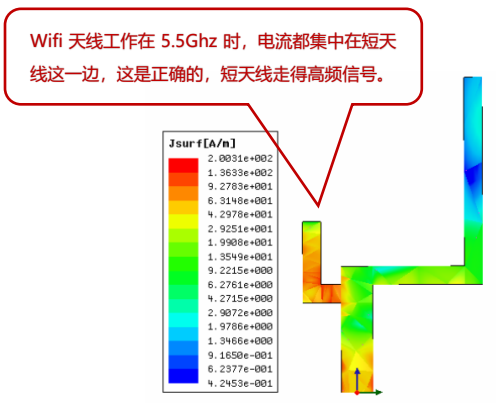


Fig. 3b: Surface current distribution at 5.5 GHz

E plane 是 YZ 方向辐射图，H plane 是 XZ 方向辐射图

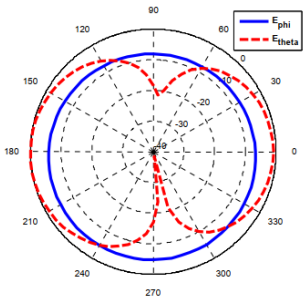


Fig. 4a: E plane at 2.3 GHz

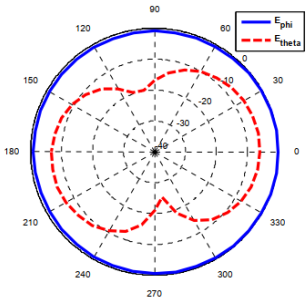


Fig. 4b: H plane at 2.3 GHz

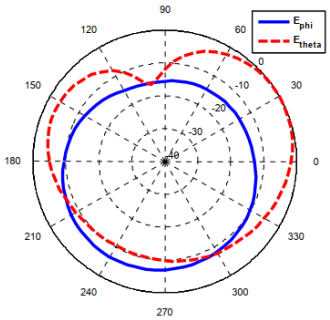


Fig. 5a: E plane at 5.5 GHz

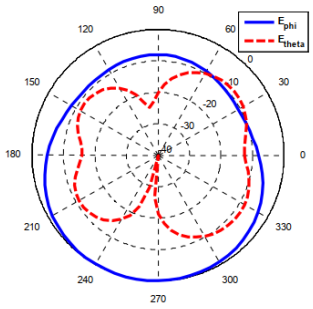


Fig. 5b: H plane at 5.5 GHz

设计指标

双频 PCB 天线: 工作频率 2.4~2.4825Ghz，802.11b 频段，和 5.15~5.825Ghz 802.11a 频段

PCB 板厚: 1.52mm，

PCB 板材: RO4003（罗杰斯）， 介电常数 3.55

PCB 板大小: 40 x40mm

其余尺寸按照手册来

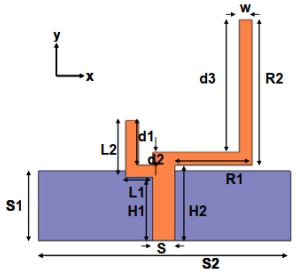
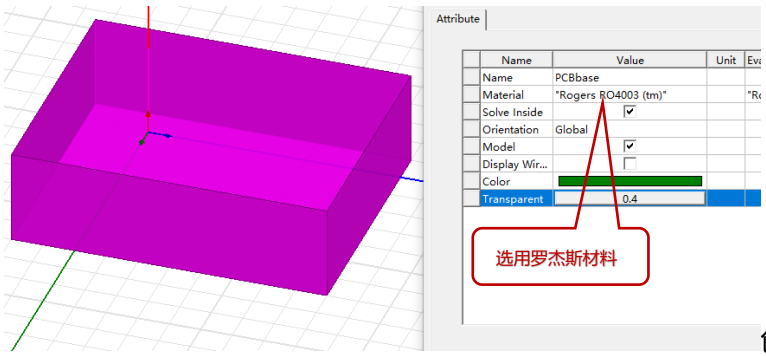


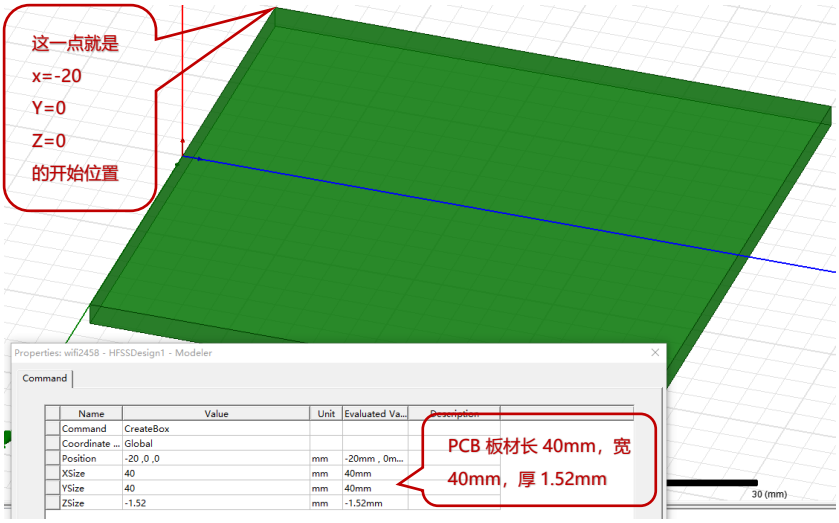
Fig. 1: Antenna Schematic

Table I: Dual-monopole antenna dimensions

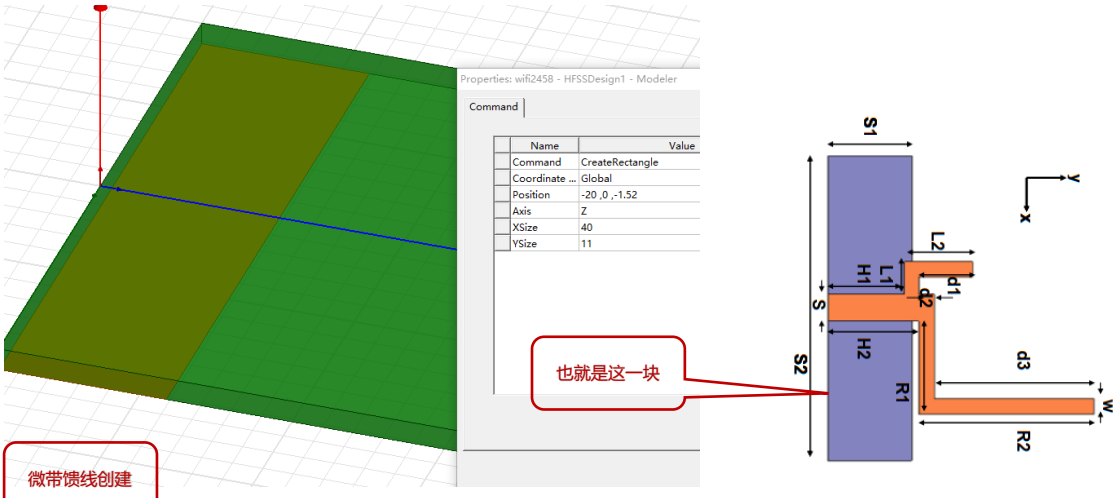
H1	10.00 mm	S	3.50 mm
H2	12.00 mm	w	2.00 mm
R1	12.25 mm	d1	7.00 mm
R2	23.00 mm	d2	2.00 mm
L1	4.25 mm	d3	21.00 mm
L2	9.00 mm	S1	11.00 mm
		S2	40.00 mm

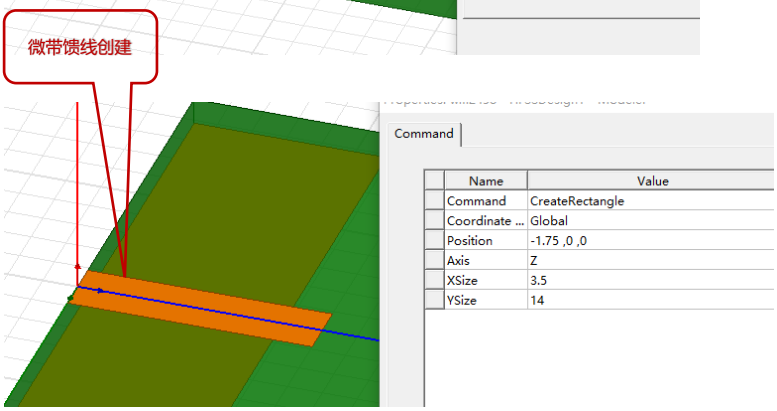


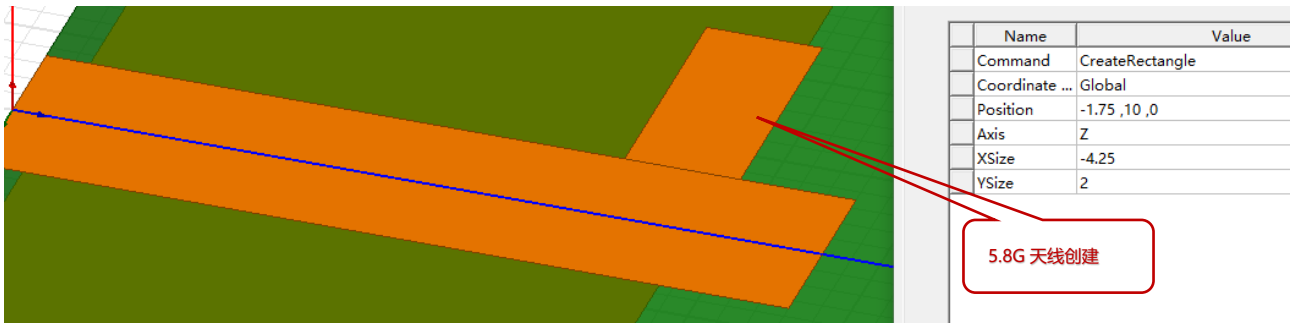
创建 PCB 基材

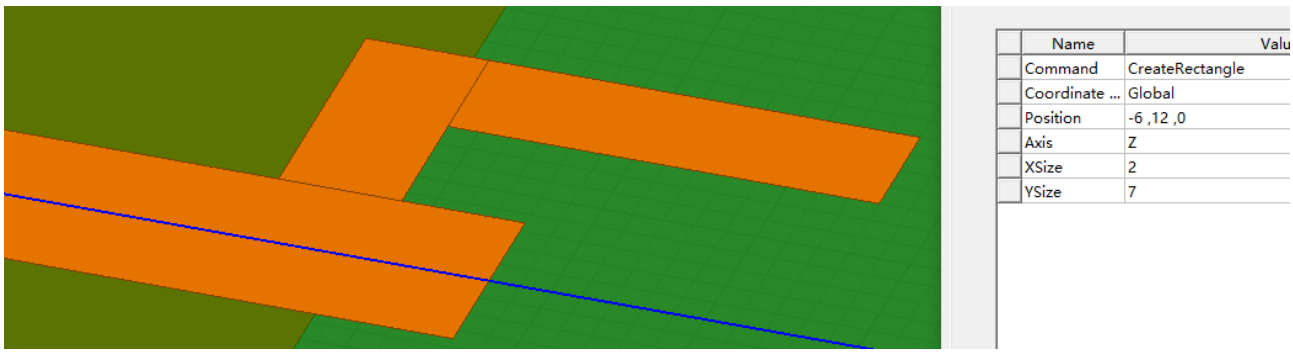


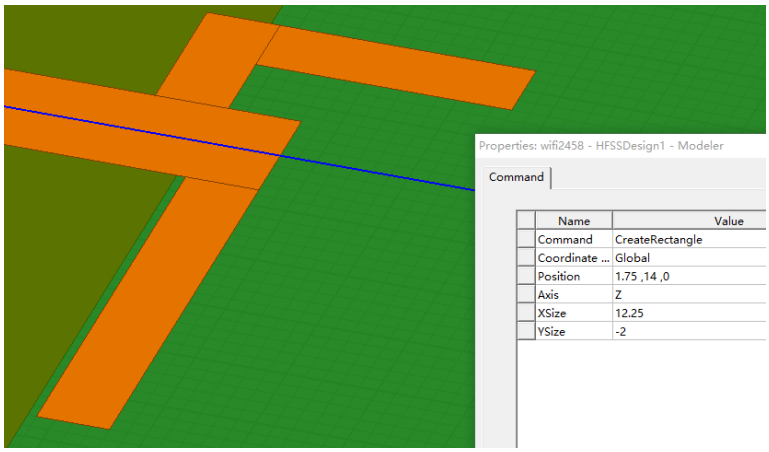
参考 GND 地设计

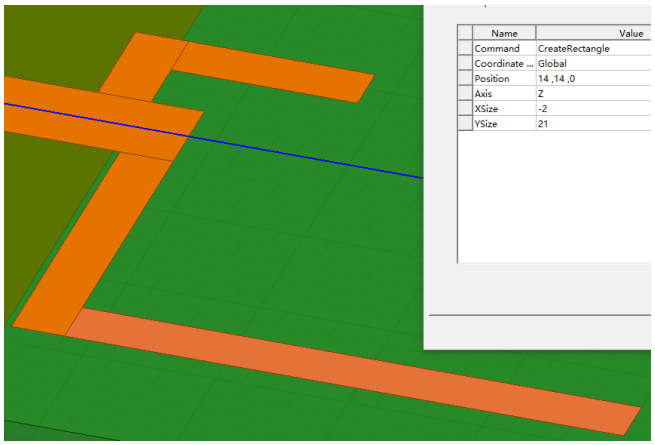





Name
Value
Command
CreateRectangle
Coordinate ...
Global
Position
-1.75 ,10 ,0
Axis
Z
XSize
-4.25
YSize
2
5.8G 天线创建

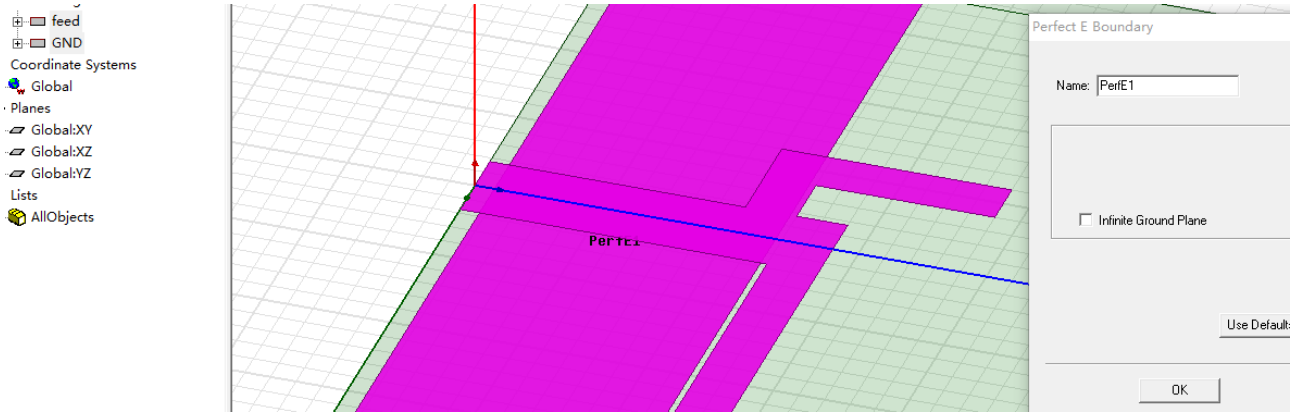

Name
Valu
Command
CreateRectangle
Coordinate ...
Global
Position
-6 ,12 ,0
Axis
Z
XSize
2
YSize
7


Properties: wifi2458 - HFSSDesign1 - Modeler
Command
Name
Value
Command
CreateRectangle
Coordinate ...
Global
Position
1.75 ,14 ,0
Axis
Z
XSize
12.25
YSize
-2

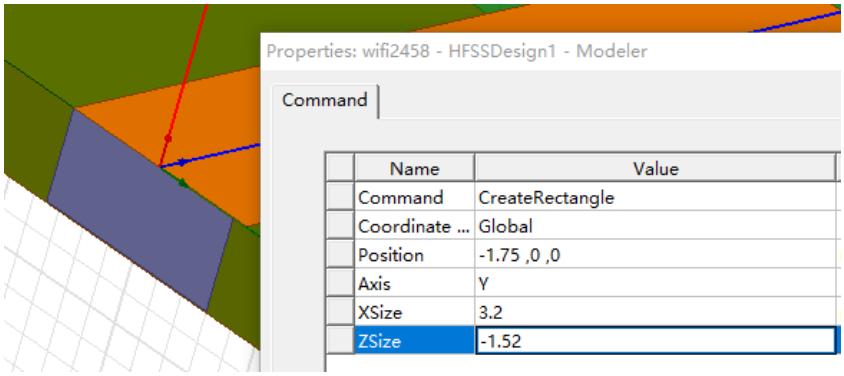

Name
Value
Command
CreateRectangle
Coordinate ...
Global
Position
14 ,14 ,0
Axis
Z
XSize
-2
YSize
21

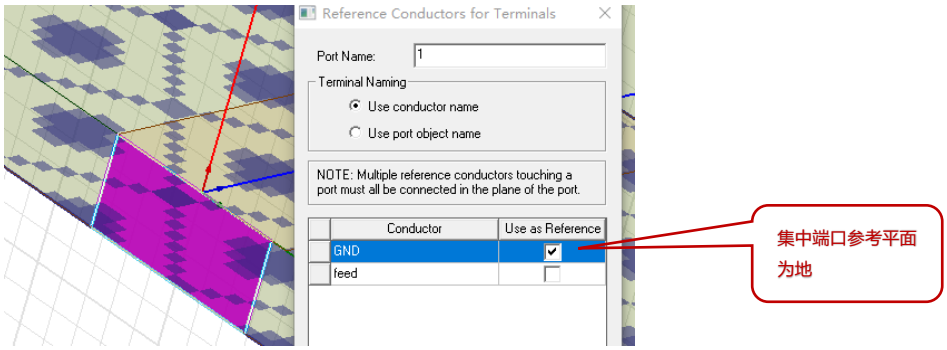
合并天线矩形面

导线和 GND 平面都设置成理想导体边界条件



设置集中参数端口(激励源)，激励源设置在馈线和 GND 之间，将馈线和 GND 连接起来





确保集中端口集阻抗为 50 欧姆

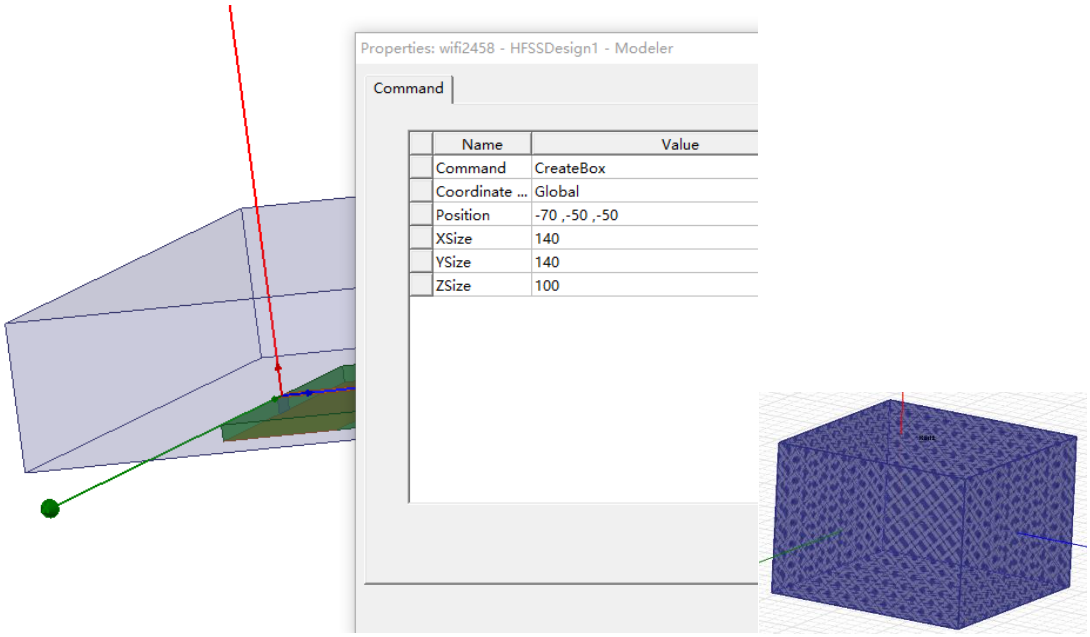
创建空气盒，空气盒与天线的距离要大于 1/4 波长

因为是双频天线 2.4G 和 5.8G，那么空气盒与 PCB 天线的距离取哪个频段呢?

这种双频就取低频的频段，这里取 2.4G 波长计算空气盒大小

$$\frac{3 \times 10^8}{2.4 \times 10^9} = 0.125m = 125mm$$

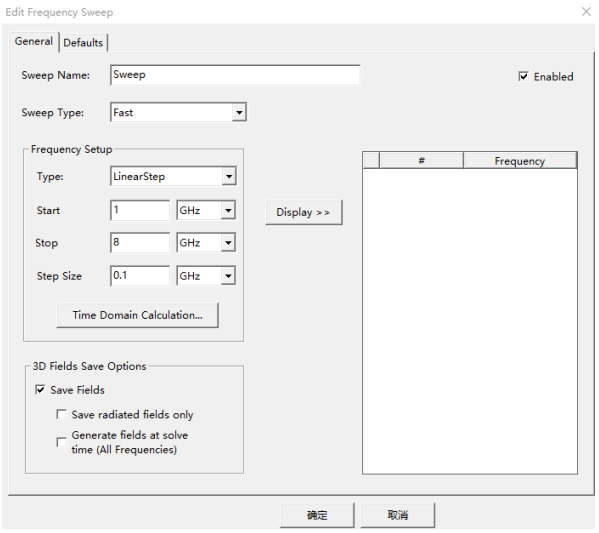
$$取 \frac{1}{4} = 31mm$$



设置空气盒辐射表面边界条件

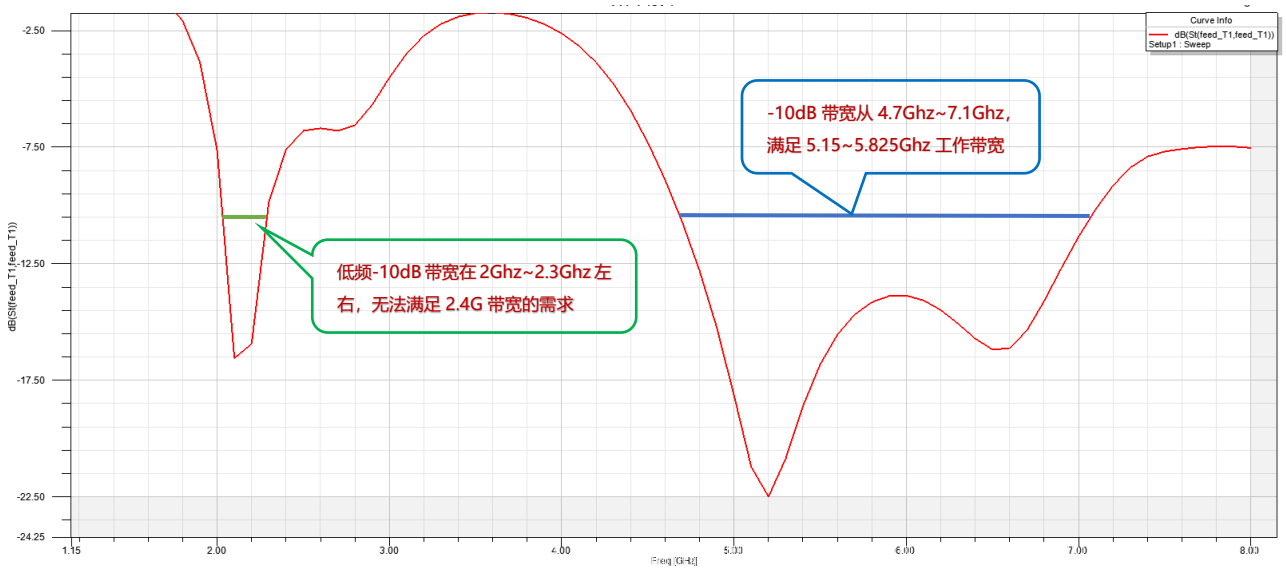
下面设置求解分析项

设置扫频分析项



开始仿真分析

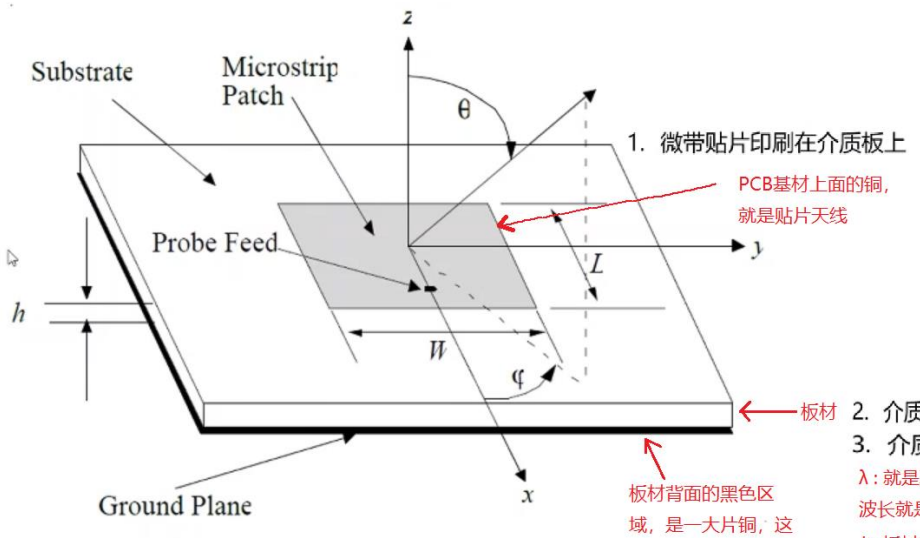
双频天线回波损耗分析



所以我们要改善 2.4Ghz 天线部分的尺寸和特性。5.8Ghz 频段天线部分就不需要修改了。

自己查阅资料修改……

微带贴片天线设计



1. 微带贴片印刷在介质板上

PCB基材上面的铜，就是贴片天线

板材

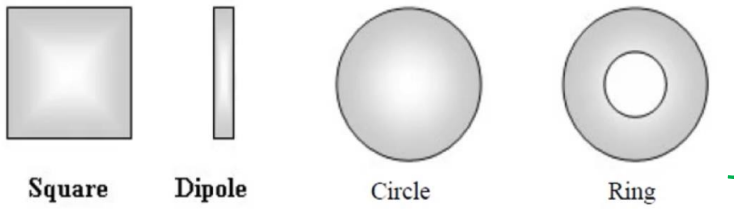
2. 介质板的相对介电常数通常是2到4
3. 介质板的厚度通常满足 $h << \lambda_0$

λ：就是真空中的波长，比如3Ghz频率在真空中波长就是100mm毫米 波长 = C / F

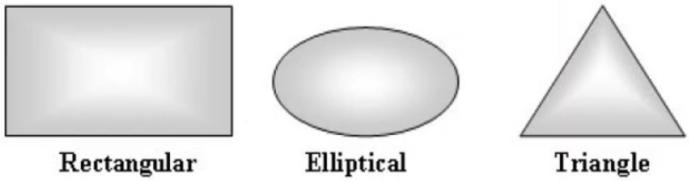
h: 板材厚度

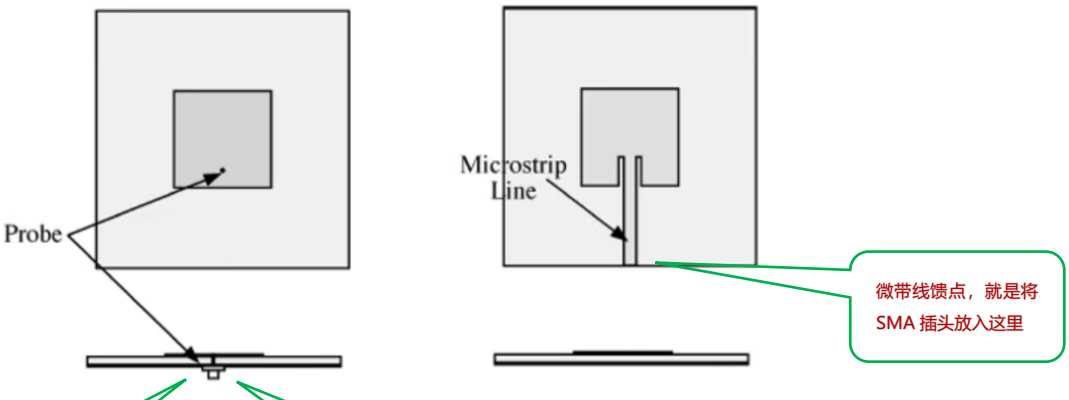
板材厚度要远远 < 真空中波长

板材背面的黑色区域，是一大片铜，这片铜当做GND处理



贴片天线做成这些形状都可以





微带线馈点，就是将SMA 插头放入这里

SMA 插座外壁与底部铜箔相连，底部铜箔相当于电源负极

同轴线馈点，将 SMA 插头内芯穿过基板到达顶部贴片铜箔

微带线设计公式如下:

$$W = \frac{c}{2f\sqrt{\frac{2}{\varepsilon_r + 1}}}$$

1.先确定线宽

W：微带天线线宽
f： 天线谐振频率
c：真空中光速
ε_r: PCB板材相对介电常数

一般都是用这个公式

2.根据已知的天线铜箔宽度W，计算有效介电常数

$$\varepsilon_{eff} = \begin{cases} \frac{\varepsilon_r + 1}{2} + \frac{\varepsilon_r - 1}{2}\left(1 + 12\frac{h}{W}\right)^{-\frac{1}{2}} & \text{for } \frac{W}{h} > 1 \\ \frac{\varepsilon_r + 1}{2} + \frac{\varepsilon_r - 1}{2}\left[\left(1 + 12\frac{h}{W}\right)^{-\frac{1}{2}} + 0.04\left(1 - \frac{W}{h}\right)^2\right] & \text{for } \frac{W}{h} < 1 \end{cases}$$

for $\frac{W}{h}>1$ 如果微带天线线宽 / 板材厚度 > 1就取上面这个公式

for $\frac{W}{h}<1$ 如果微带天线铜箔宽度 / 板材厚度<1 就用下面这个公式

很少看到PCB天线线宽比板材厚度还小的，一般用不到下面这个公式

ε_{eff}：得到有效介电常数

3.已知 天线线宽w， 板材厚度h， 有效介电常数 ε_{eff} ， 计算变化的ΔL长度

$$\Delta L = \frac{0.412h\,(\varepsilon_{eff} + 0.3)\,(\frac{w}{h} + 0.264)}{(\varepsilon_{eff} - 0.258)\,(\frac{w}{h} + 0.8)}$$

ΔL：变化的天线长度

4.已知有效介电常数 ， 光速， 天线谐振频率，求天线有效长度

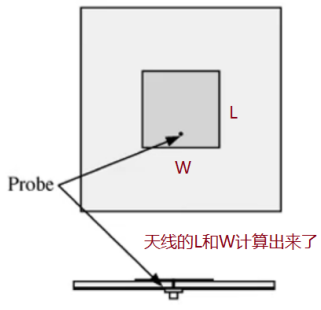
$$f = \frac{c}{2L_{eff}\sqrt{\varepsilon_{eff}}}$$

f： 天线谐振频率
c：真空中光速
ε_{eff}：有效介电常数
求L_{eff}：有效天线长度

5.已知变化天线长度，有效天线长度，计算天线真实长度

$L_{eff} = (L + 2\Delta L)$

L：就是要计算的天线实际长度



(a)Probe Feed

用 HFSS 仿真软件设计一个微带贴片天线

设计要求:

天线谐振频率: 2.45Ghz

板材: 型号(罗杰斯)Rogers RT /Duroid 5880。板厚 1.575mm

PCB介质基板尺寸确定

$L_{GND} > L + 6h$

L： 天线铜箔长度

L_{GND}: 基板长度

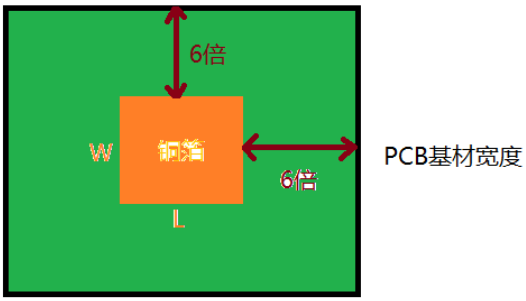
所以PCB基板长度 要 > 6倍的天线长度

$W_{GND} > W + 6h$

W: 天线铜箔宽度

W_{GND} : 基板宽度

基板宽度和长度同理都是要>6倍天线尺寸



PCB基材长度

所以PCB基板长宽比铜箔大6倍

根据前面的公式计算出

光速 c = 3x10^8

频率 f = 2.45Ghz

板材介电常数 e = 2.2

板厚 h=1.575mm

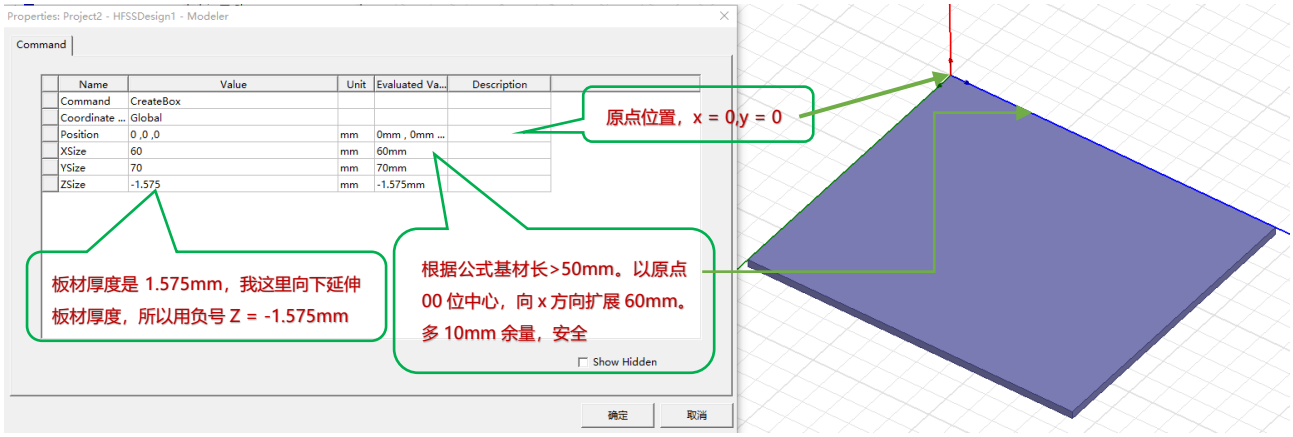
天线铜箔宽度 W = 0.484

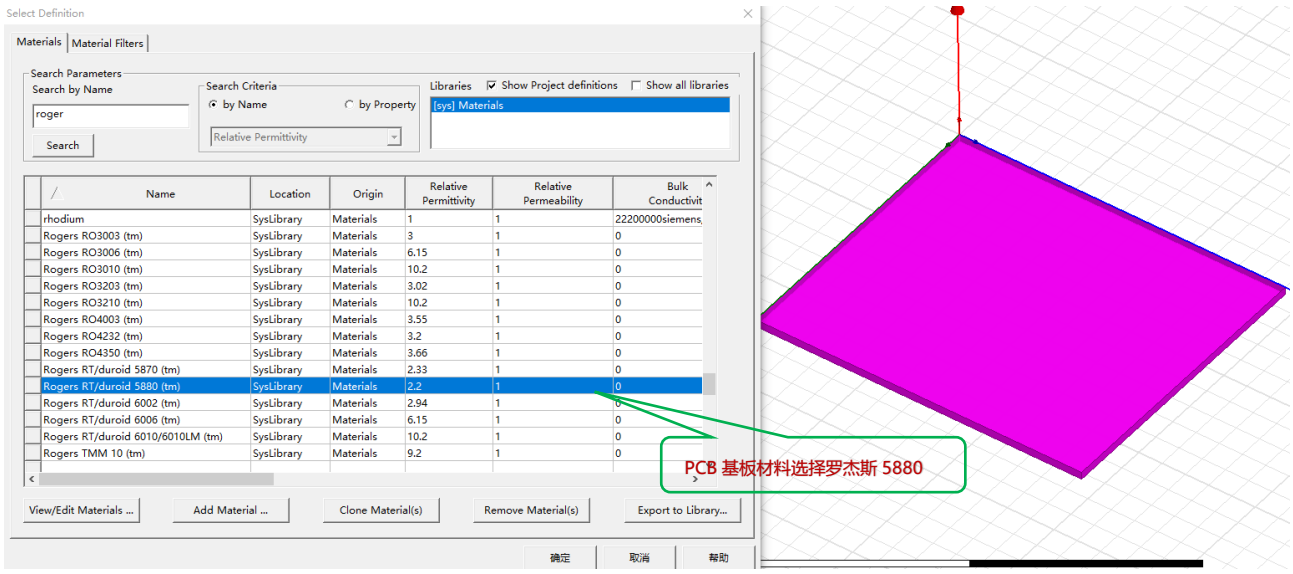
天线铜箔长度 L = 0.0405

LGND(板材长) >50mm

WGND(板材宽) > 57mm

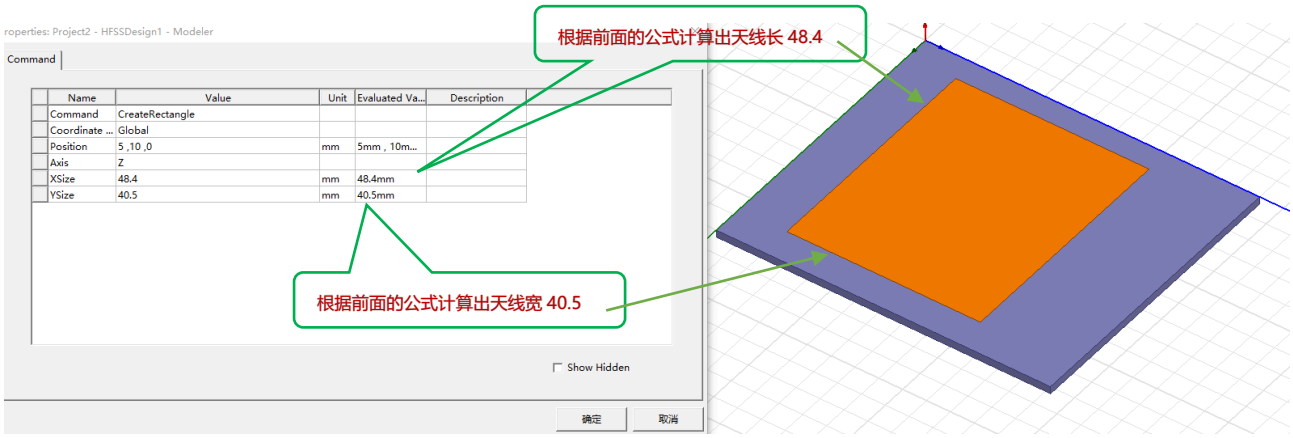
我建模用的是默认的模型求解器，而不是前面使用的终端 terminal 求解器，这点要注意哦





天线铜箔宽度 W = 0.484

天线铜箔长度 L = 0.0405



按照前面基板尺寸要求

$L_{GND} > L + 6h$

L：天线铜箔长度

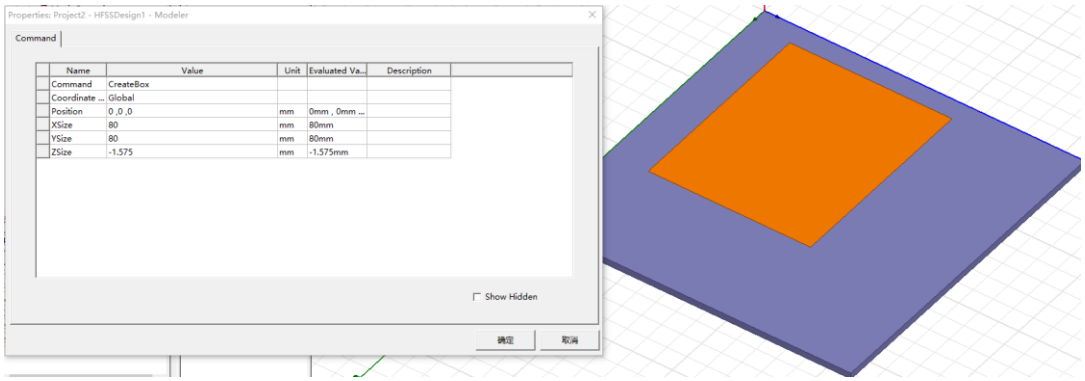
L_{GND}：基板长度

$W_{GND} > W + 6h$

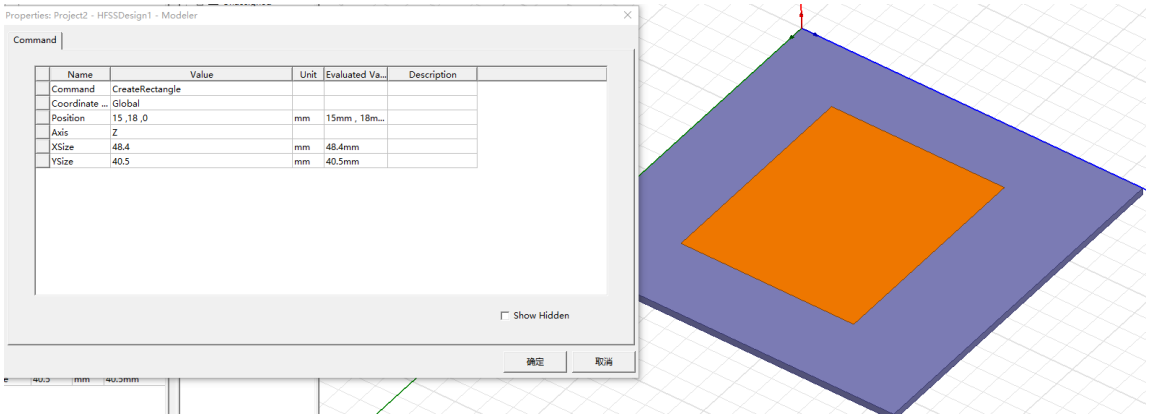
W：天线铜箔宽度

W_{GND}：基板宽度

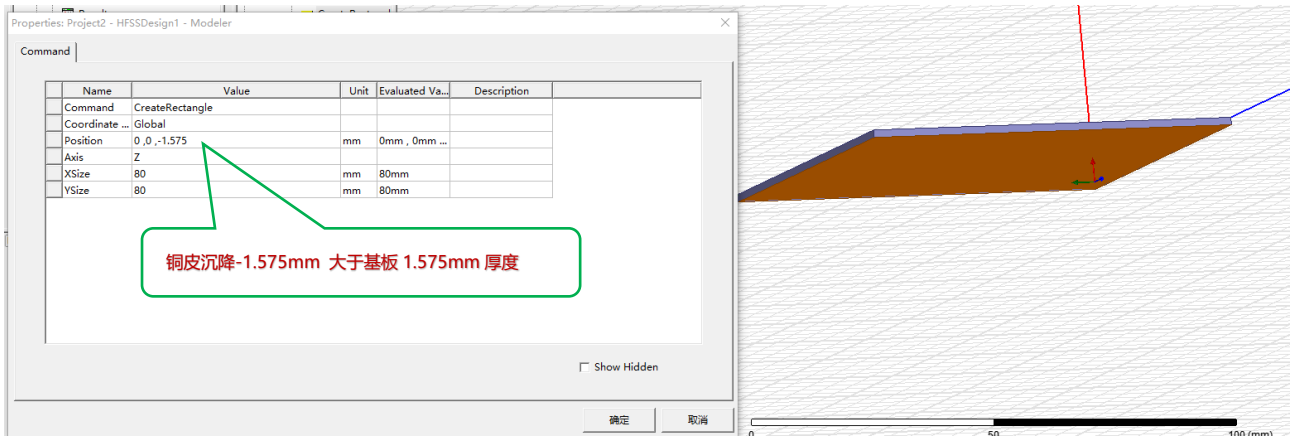
基板长宽必须大于天线尺寸 6 倍，我要修改下基板，长宽都设置成 80mm x 80mm

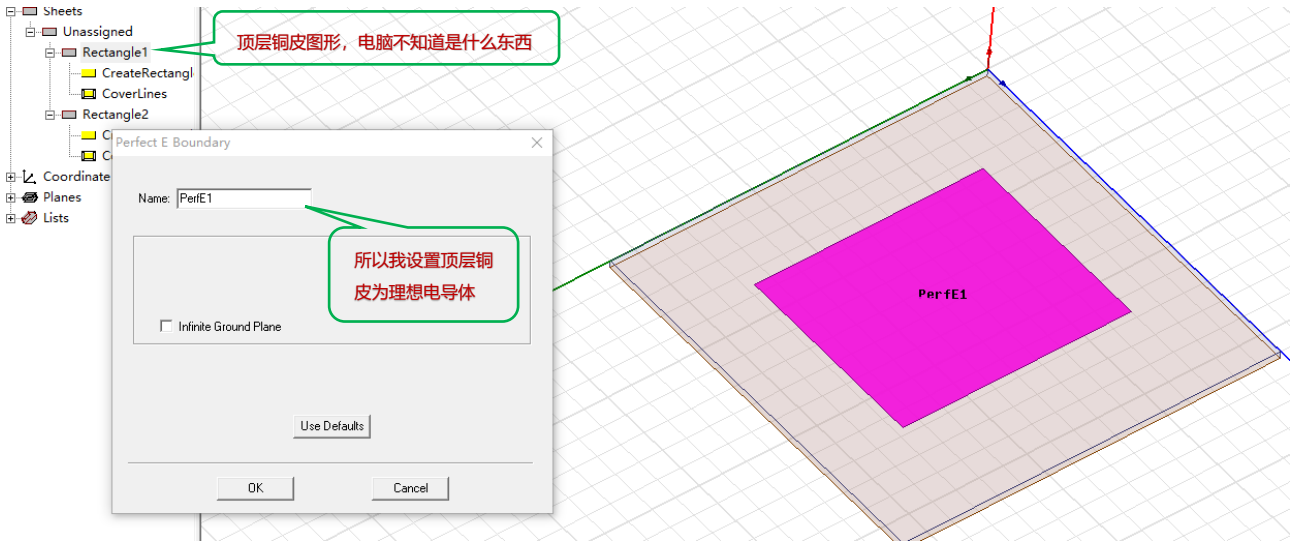


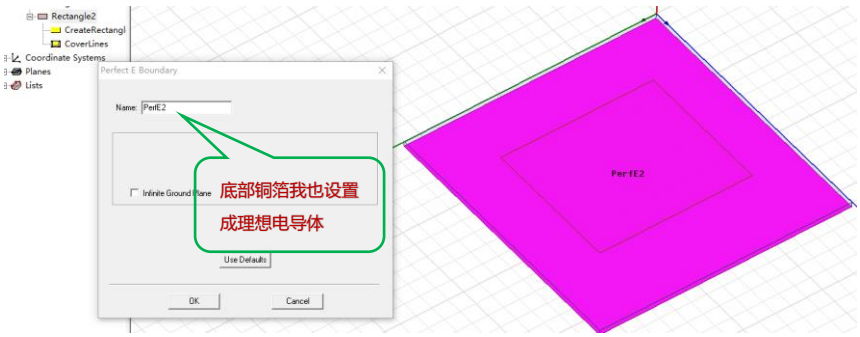
把天线铜箔放在基板中间



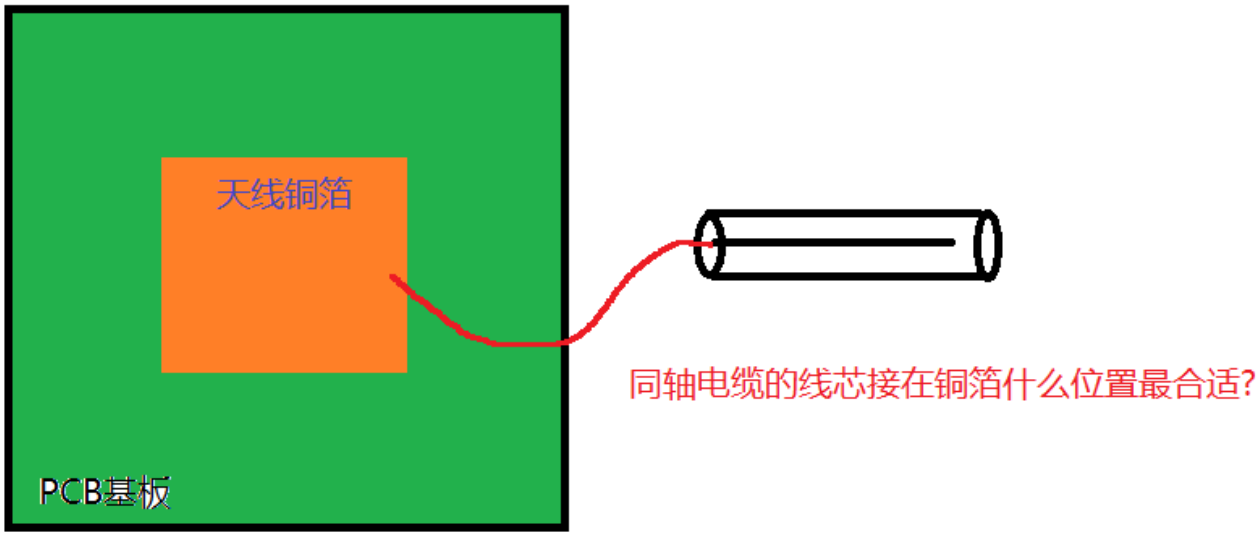
下面加入一块地铜皮，尺寸和基板一模一样，只是铜皮覆盖在基板下表面。

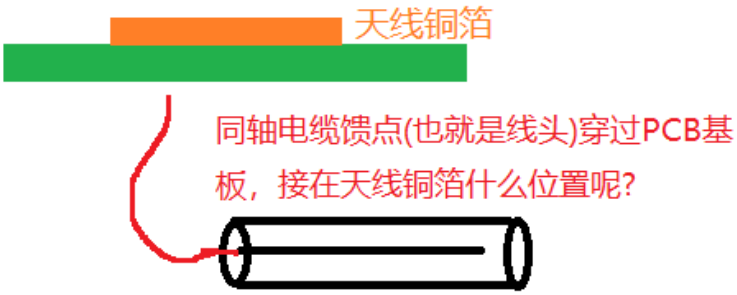


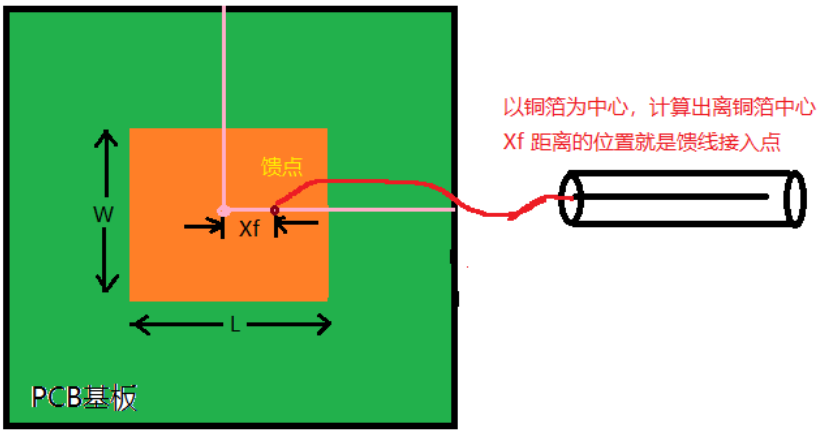




给天线增加电，没有电信号，天线无法工作。所以要确定同轴电缆的馈线接在天线铜箔的什么位置?





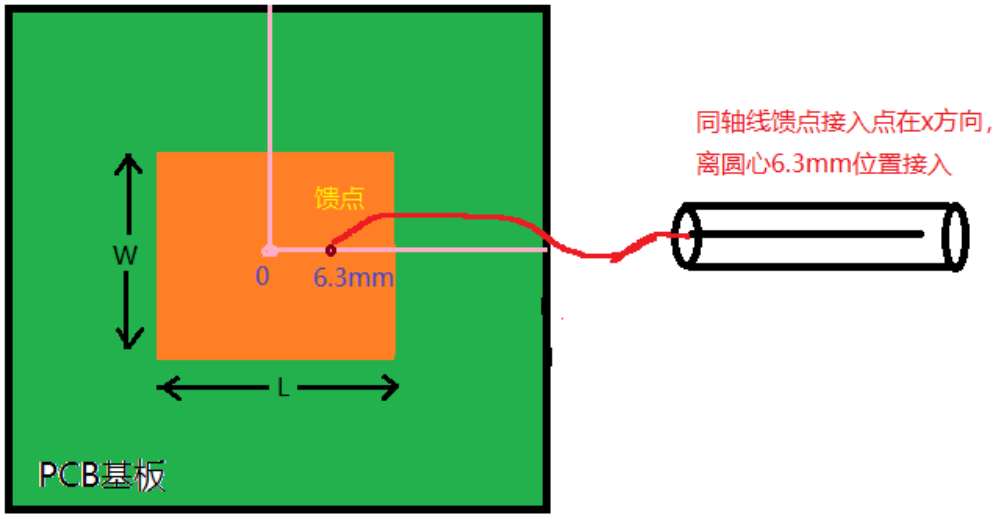


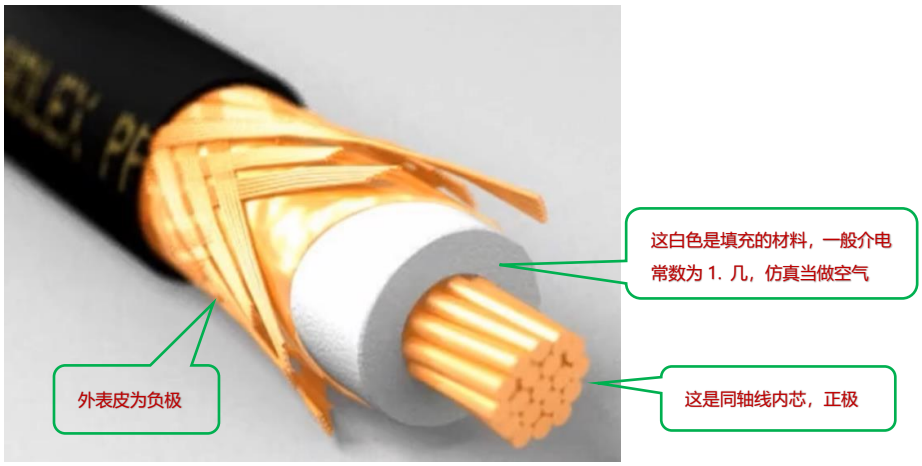
$$\varepsilon_{re}(L) = \frac{\varepsilon_r + 1}{2} + \frac{\varepsilon_r - 1}{2}\left(1 + 12\frac{h}{L}\right)^{-\frac{1}{2}}$$

$$Xf = \frac{L}{2} - \frac{L}{2\sqrt{\varepsilon_{re}}}$$

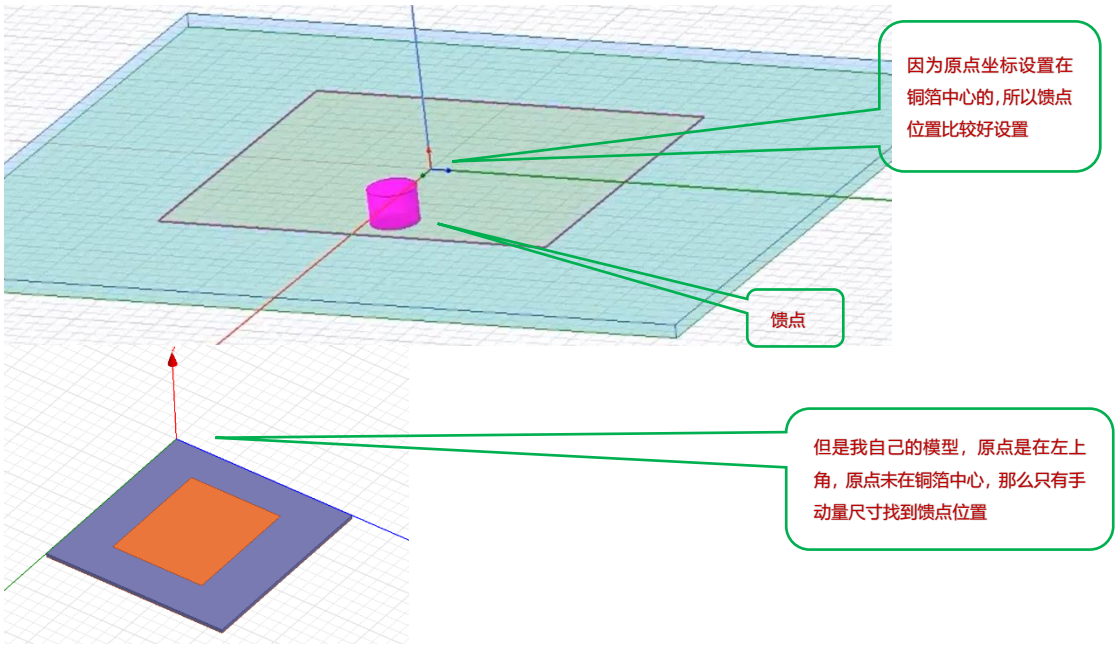
Xf 就是馈点处理铜箔中心位置相差多少mm，这个相差的位置
就是同轴电缆最适合接入的馈点处

本 2.4G 频率案例 Xf 算出来是 0.0063m(6.3m)



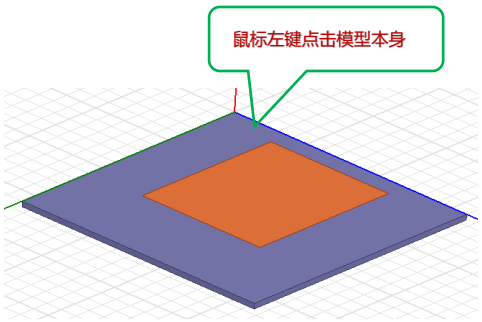


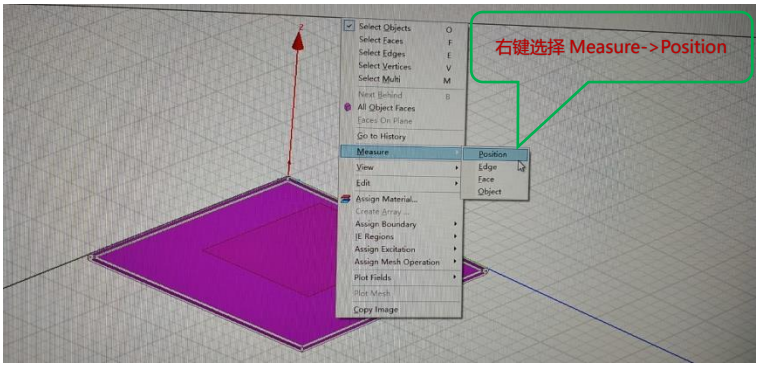
我们馈点应该从铜箔中心向外计算距离

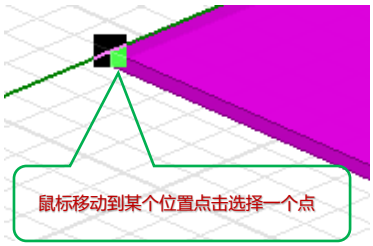


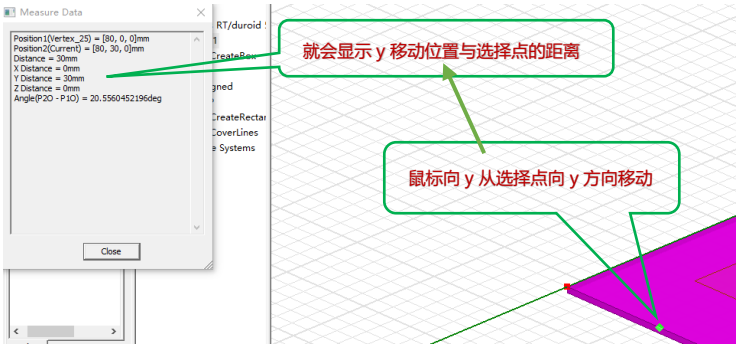
下面介绍下手动量尺寸的方法

手动测量模型尺寸

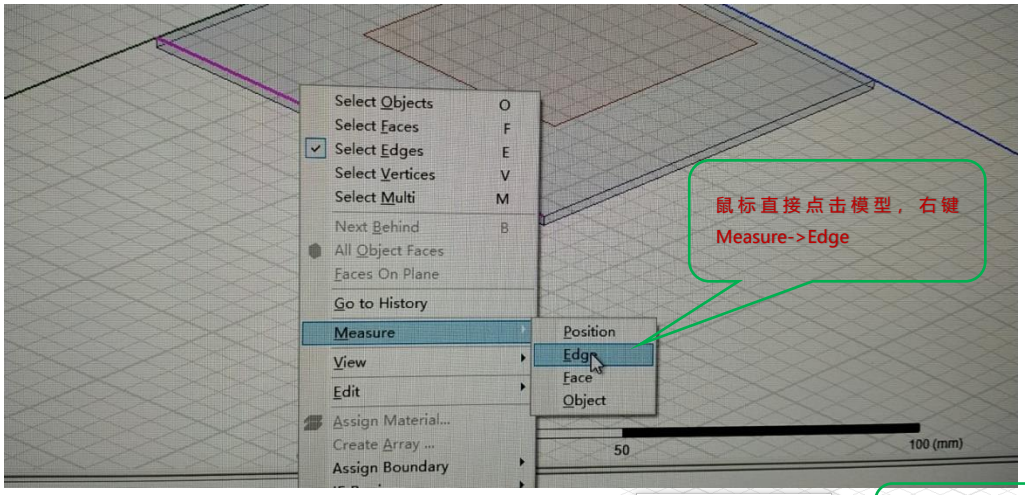


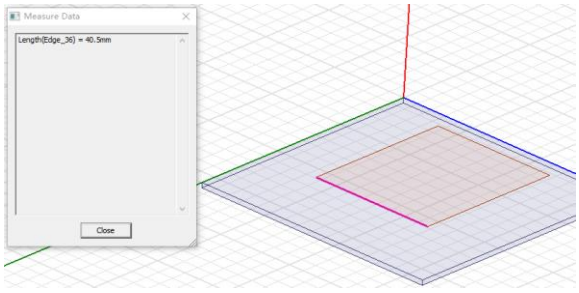


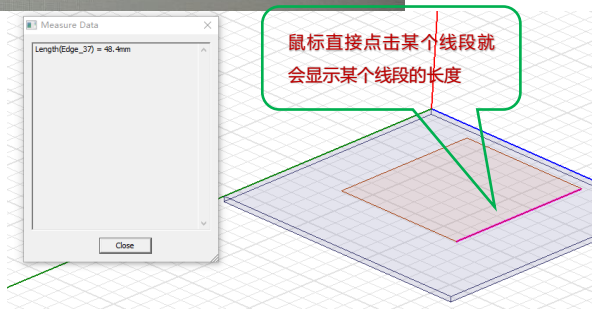




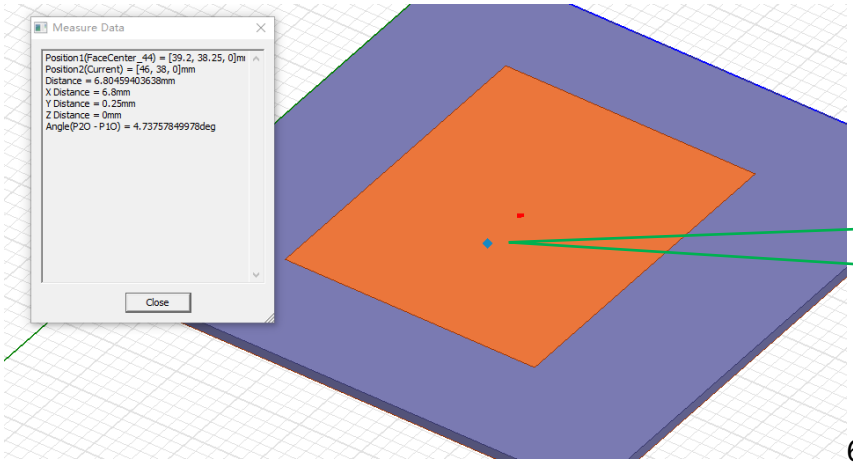
测量线段长度



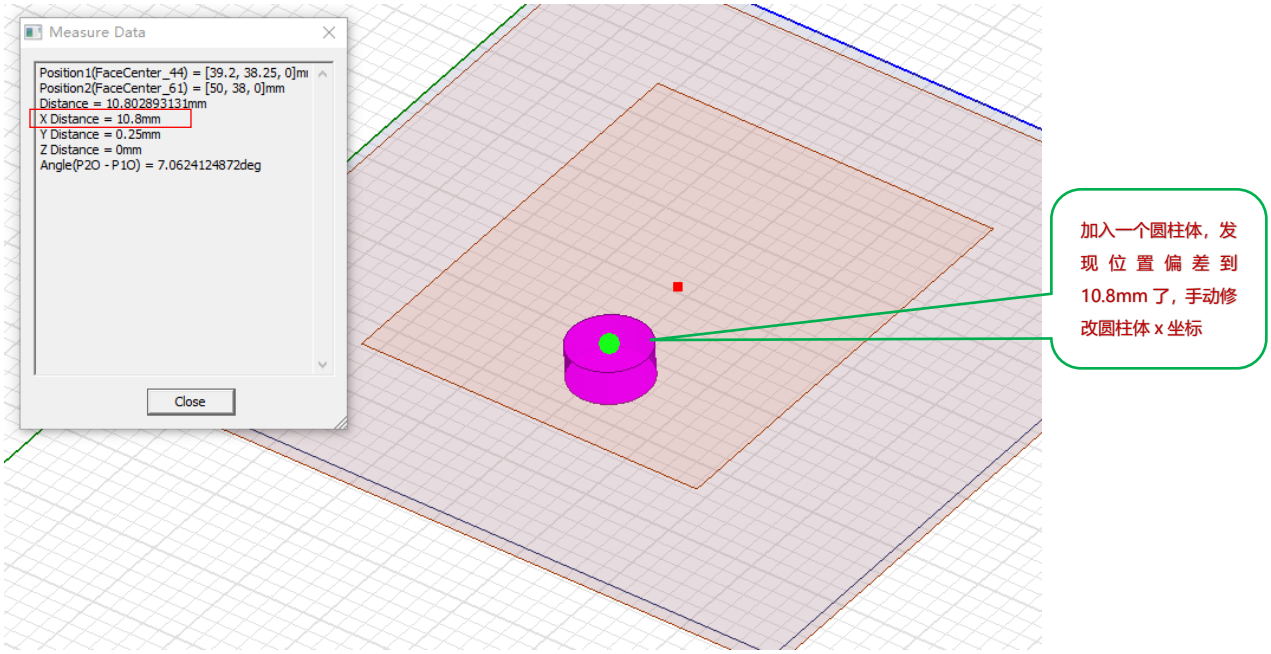


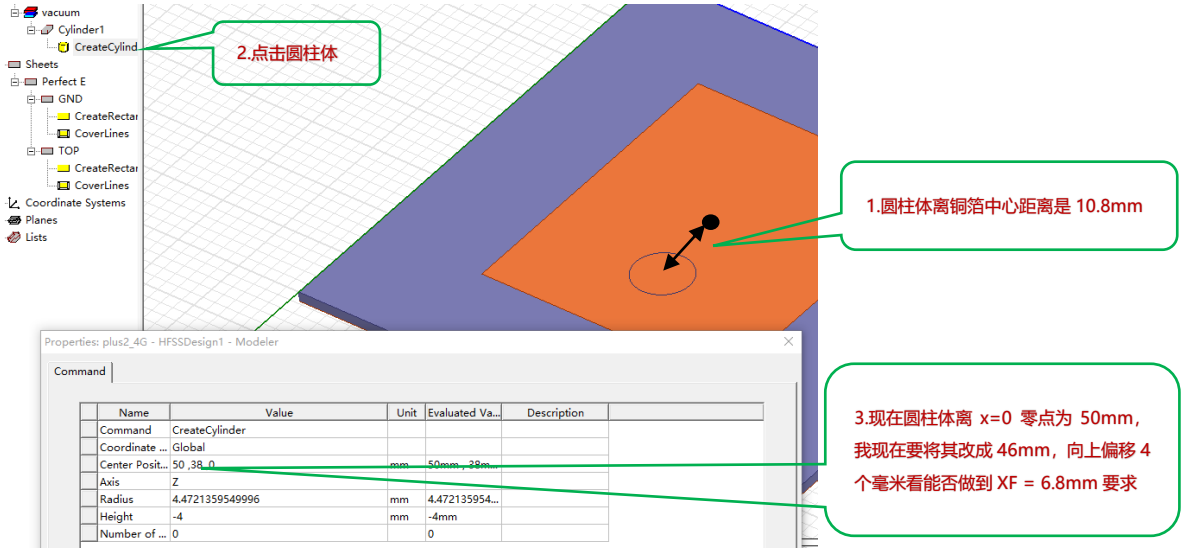


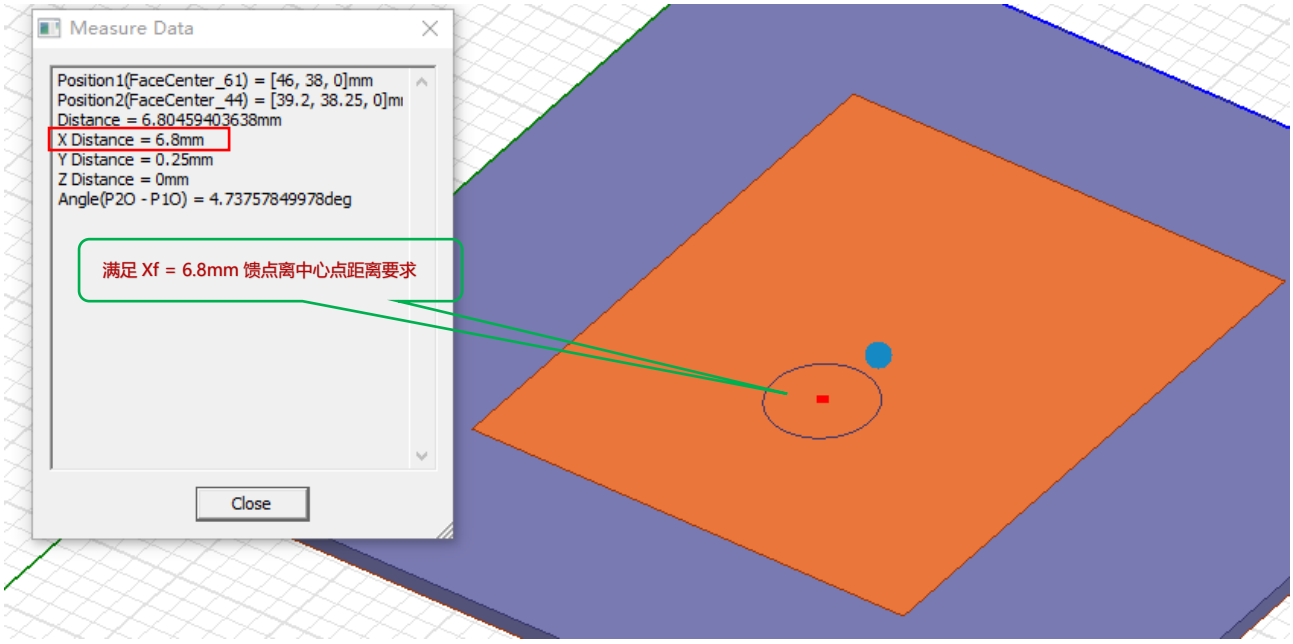
我们开始画馈点，根据前面公式介绍，我要从原点向 x 方向移动馈点，Xf = 6.3mm

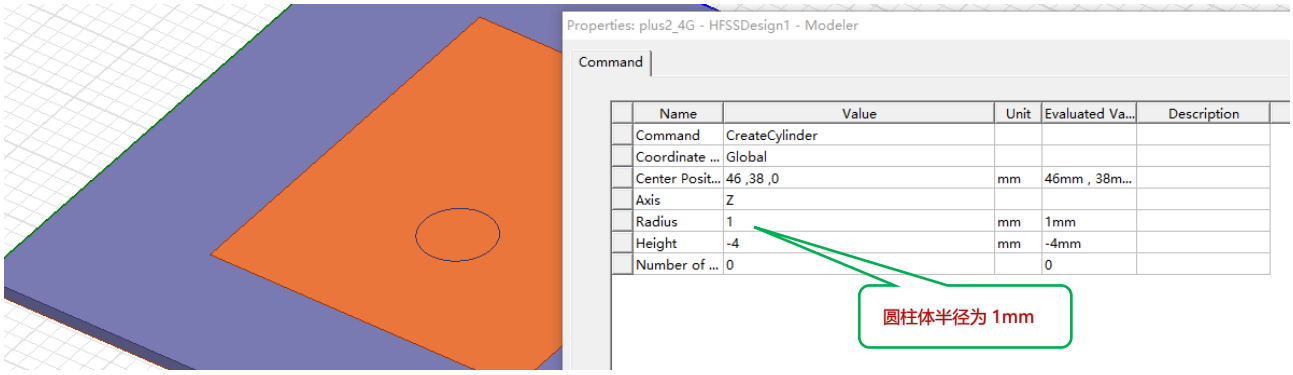


6.8mm 和 6.3mm 差不多，到时候根据仿真结果再调整。

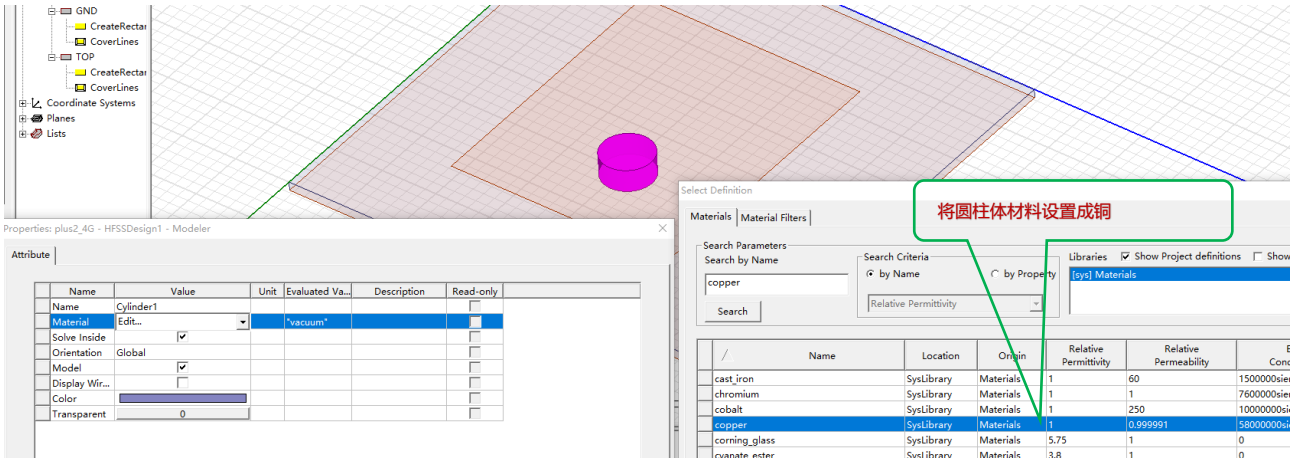


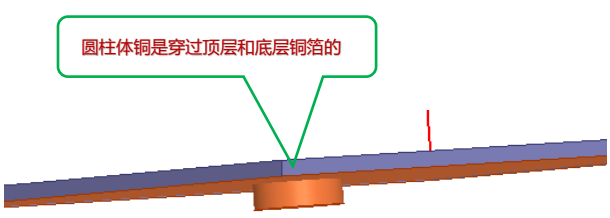


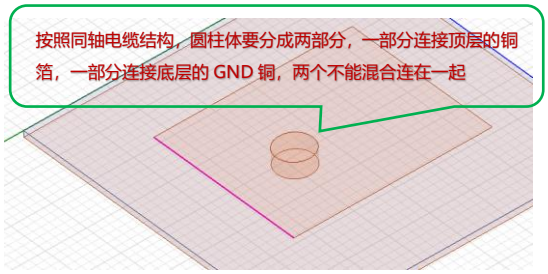


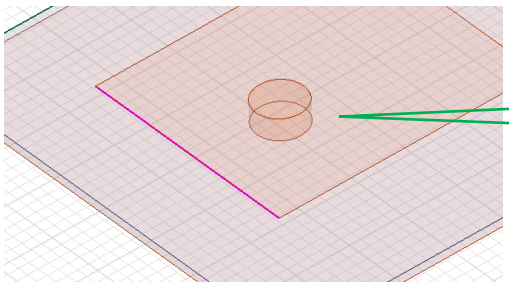


这个 1mm 圆柱体半径是为了我方便，实际应用中可以查标准(公式)确定。1mm 半径对仿真影响不大。



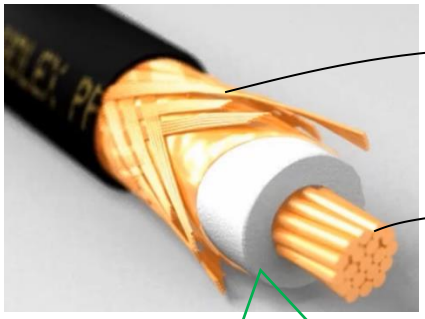


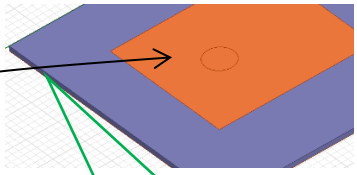


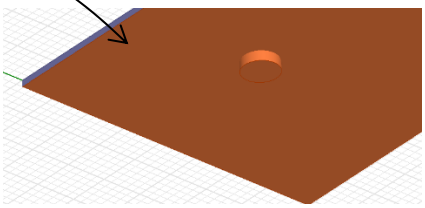


现在这个圆柱体和可能和底层 GND 铜箔和顶层中心铜箔连接在一起的

1.你看，同轴电缆外表层铜丝连接的是 PCB 背面(底层)的 GND 铜箔



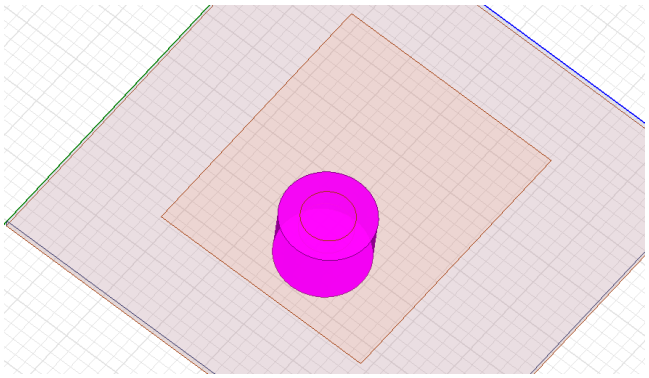


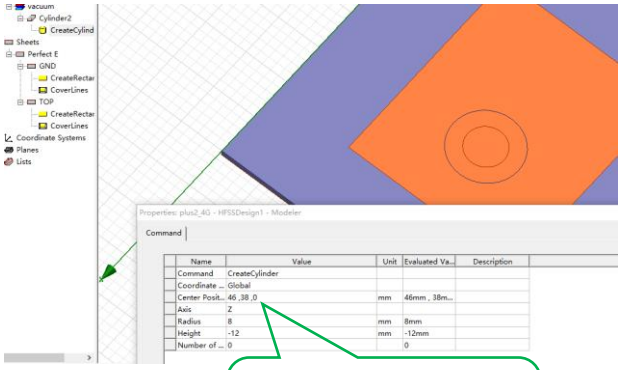


3.中间用空气或者介质隔离

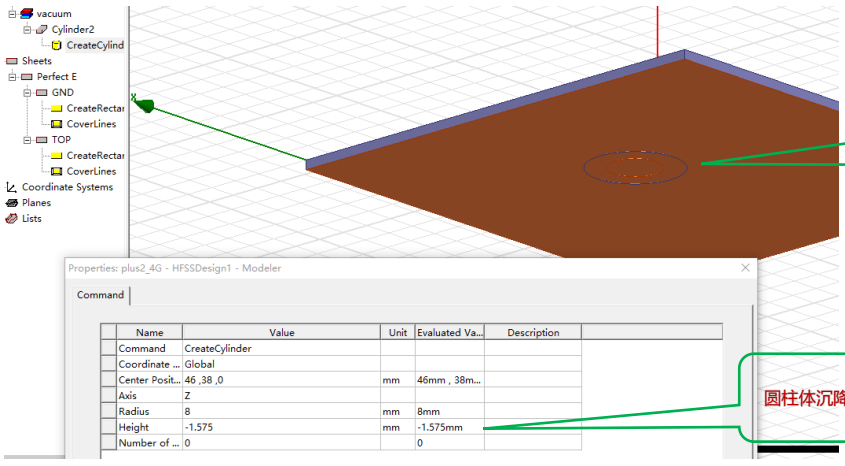
2.同轴电缆内芯铜丝连接的是 PCB 顶层的天线铜箔

所以 PCB 底层的 GND 铜箔和顶层的天线铜箔不能经过圆柱体造成短路



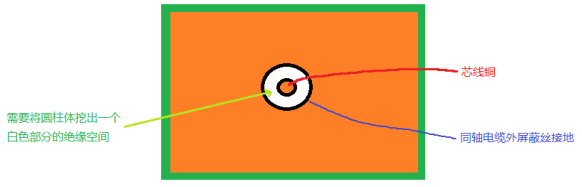


圆柱体向下沉降不是修改 Z 轴，如果修改 Z 轴向下沉降，反而把圆柱体表面沉降下去了

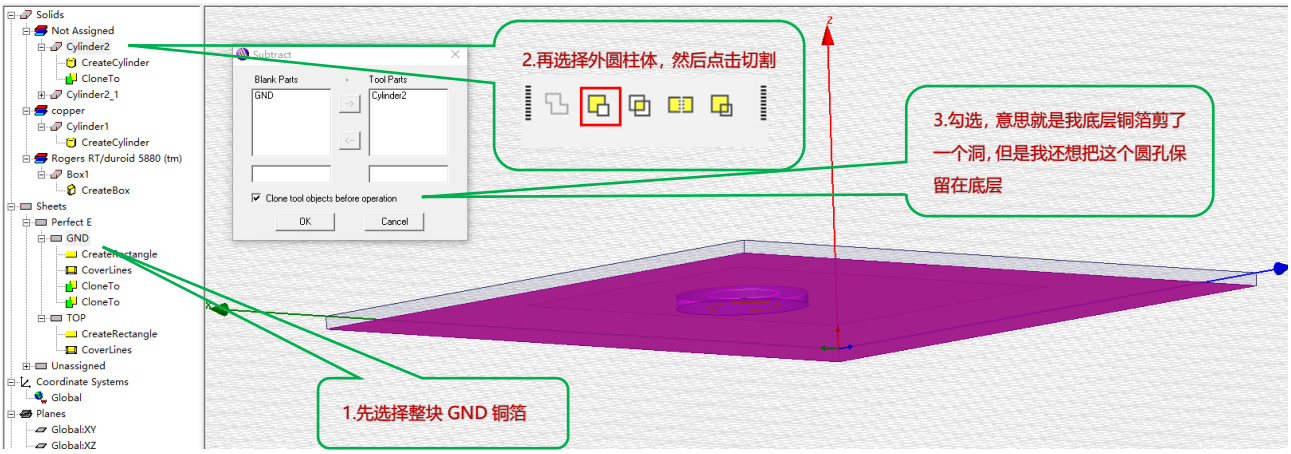


外圆柱体向下沉降-1.575mm，与底部铜箔持平

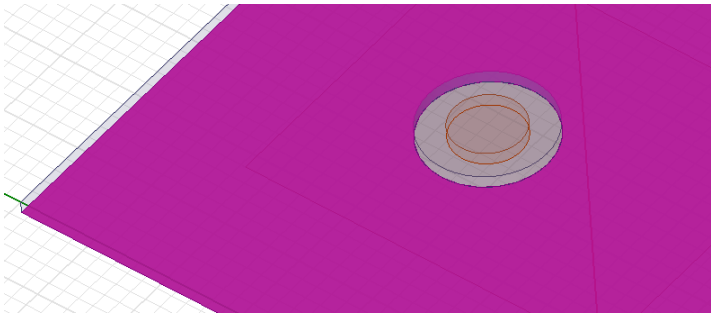
圆柱体沉降高度在 Height 设置

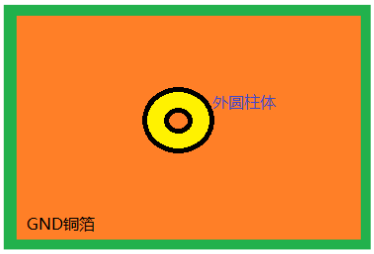


挖空圆柱体部分铜箔



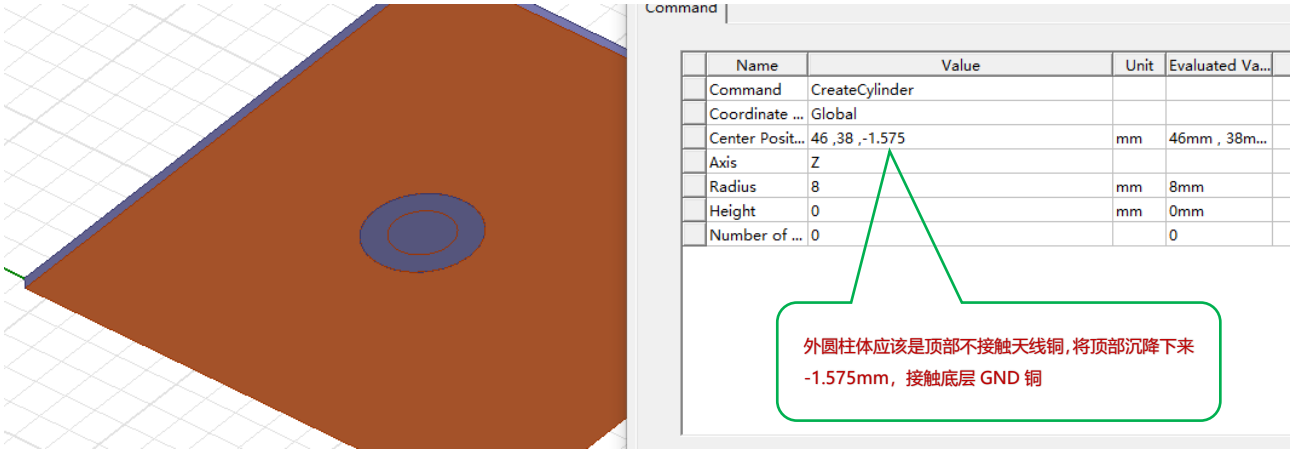
OK 之后，就将 GND 铜箔，与选择的外圆柱体部分隔离开来了。





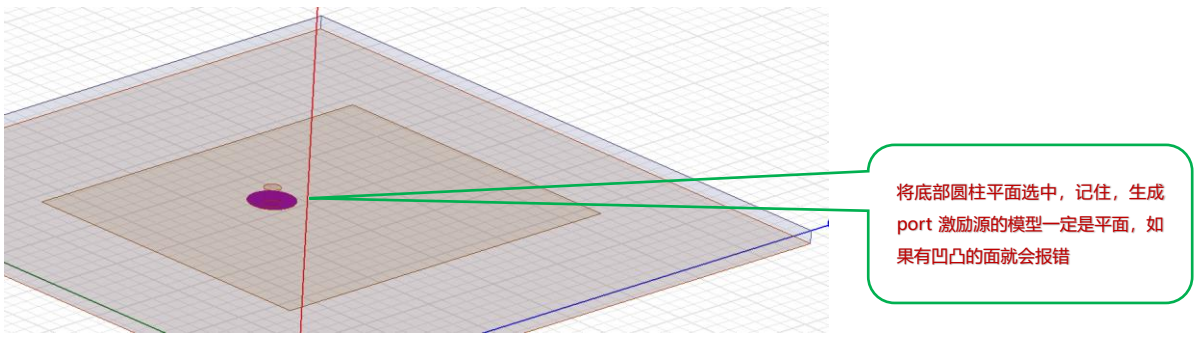
就是选择正个GND铜箔，然后按照外圆柱体的面积将GND铜箔切除个洞

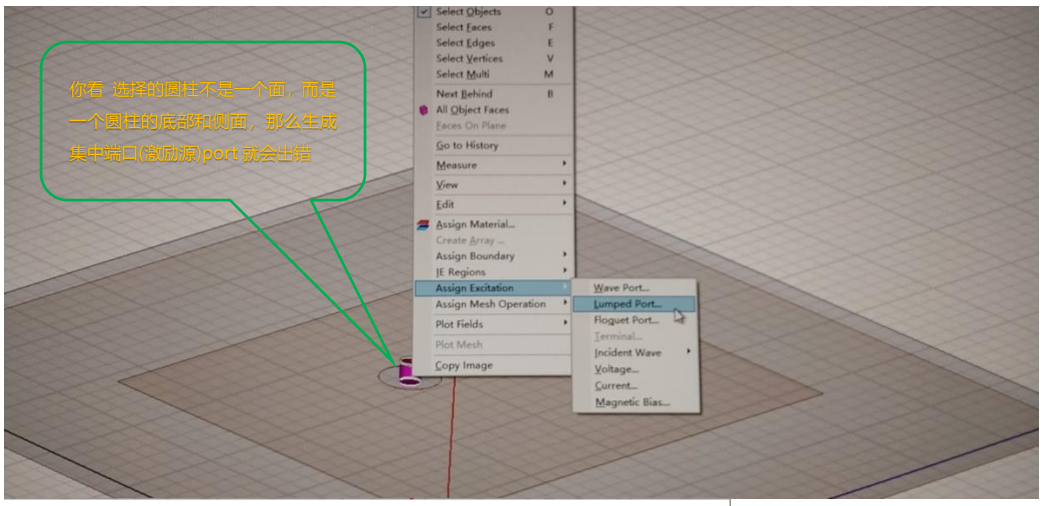
其实上面切割操作是对了的，但是外圆柱体沉降参数没有对

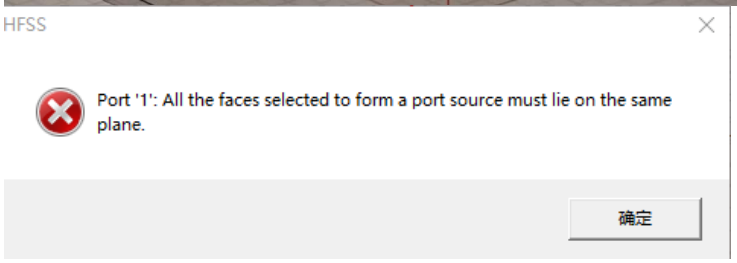


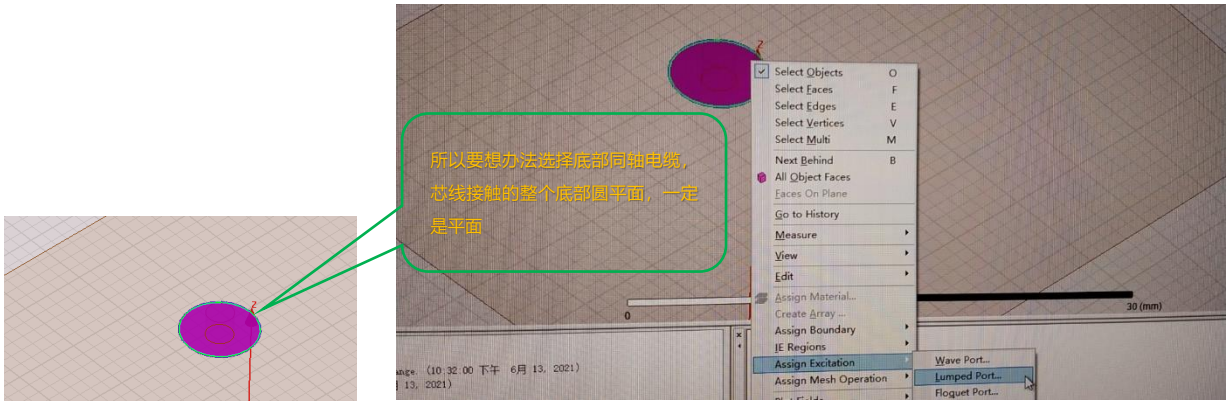
外圆柱 Height 高度可以设置为 0，这样在底层可以看到挖空的效果。

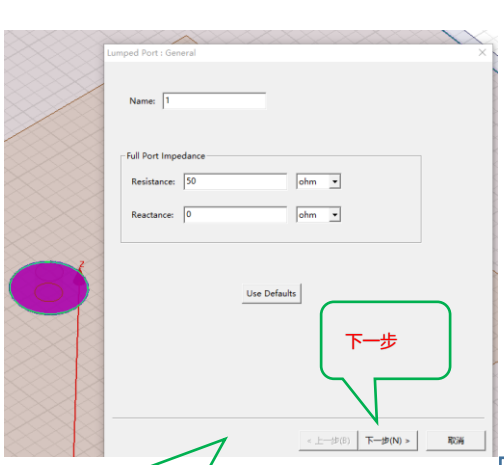
port ‘1’ All the faces **selected to form a port source must lie on the same plane** 激励源设置问题

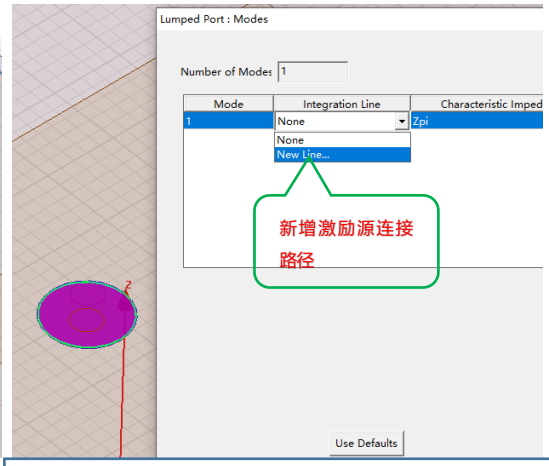


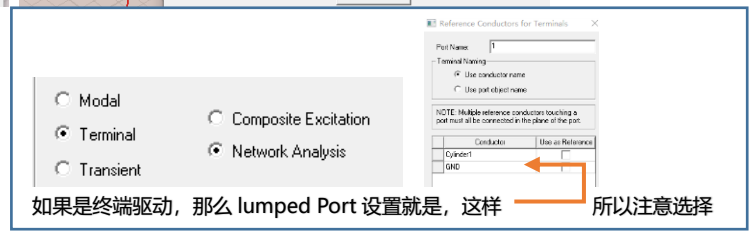

你看 选择的圆柱不是一个面，而是一个圆柱的底部和侧面，那么生成集中端口(激励源)port 就会出错
Select Objects
Select Faces
Select Edges
Select Vertices
Select Multi
Next Behind
All Object Faces
Faces On Plane
Go to History
Measure
View
Edit
Assign Material...
Create Array ...
Assign Boundary
IE Regions
Assign Excitation
Assign Mesh Operation
Plot Fields
Plot Mesh
Copy Image
Wave Port...
Lumped Port...
Floquet Port...
Terminal...
Incident Wave
Voltage...
Current...
Magnetic Bias...

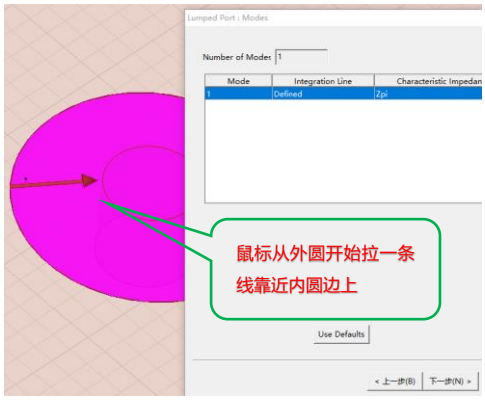

HFSS
Port '1': All the faces selected to form a port source must lie on the same plane.
确定

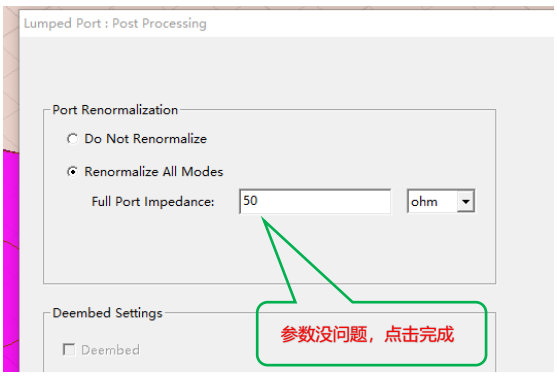

所以要想办法选择底部同轴电缆，芯线接触的整个底部圆平面，一定是平面
Select Objects
Select Faces
Select Edges
Select Vertices
Select Multi
Next Behind
All Object Faces
Faces On Plane
Go to History
Measure
View
Edit
Assign Material...
Create Array ...
Assign Boundary
IE Regions
Assign Excitation
Assign Mesh Operation
Wave Port...
Lumped Port...
Floquet Port...
0
30 (mm)


Lumped Port : General
Name: 1
Full Port Impedance
Resistance: 50 ohm
Reactance: 0 ohm
Use Defaults
下一步
这种画激励线的设置来至于 modal 模式驱动，在 HFSS->solution Type 设置


Lumped Port : Modes
Number of Modes: 1
Mode
Integration Line
Characteristic Imped
1
None
Zpi
None
New Line...
新增激励源连接路径
Use Defaults

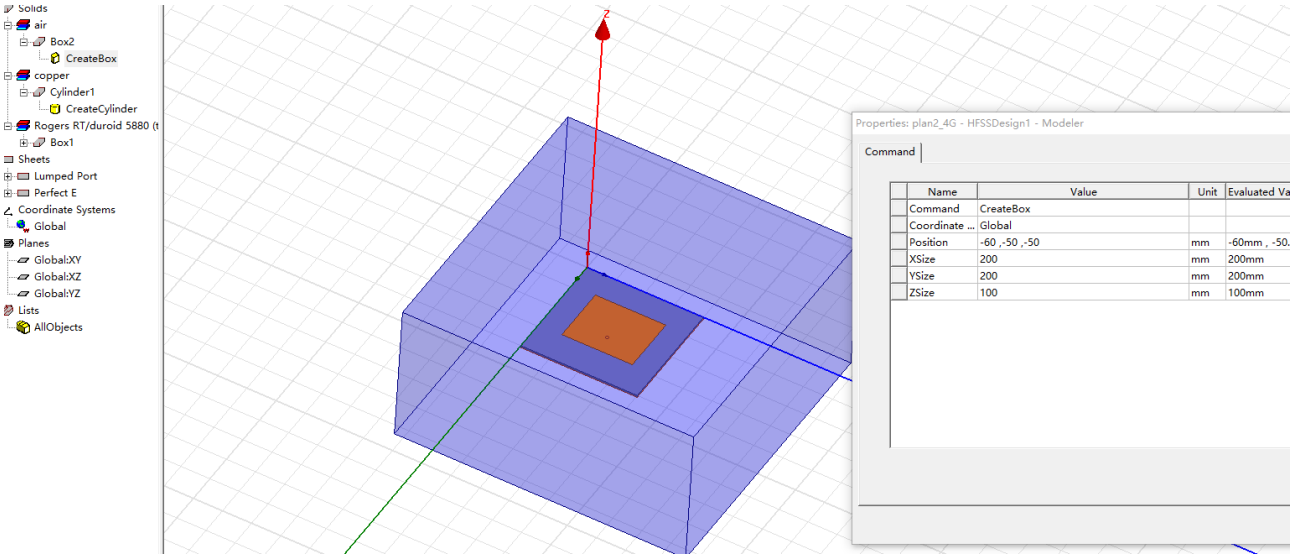

Modal
Terminal
Transient
Composite Excitation
Network Analysis
如果是终端驱动，那么 lumped Port 设置就是，这样
所以注意选择



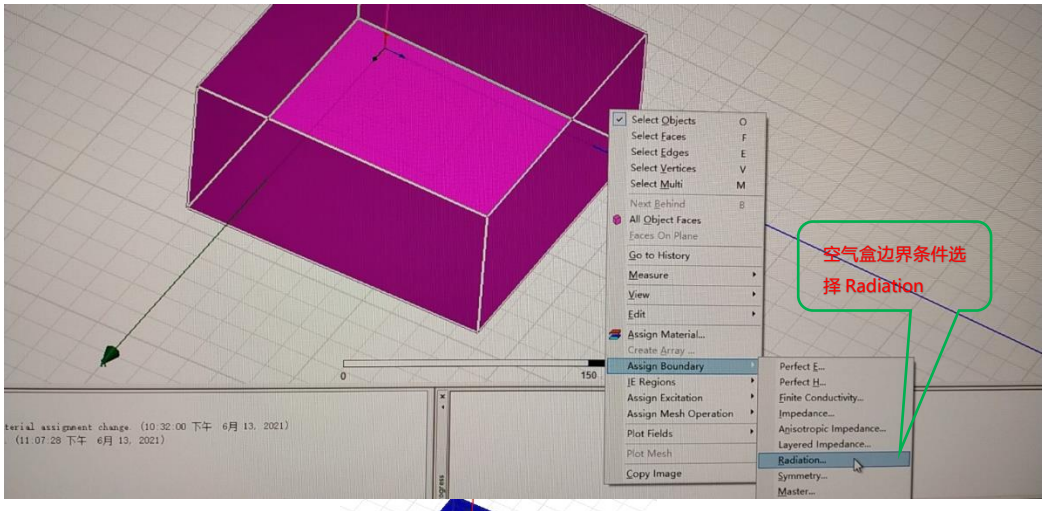


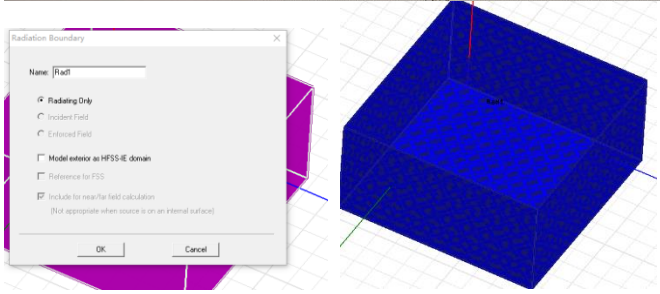
这个圆就把内外导体连接起来了

创建空气盒



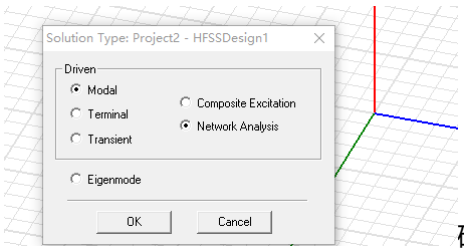
空气盒尺寸为 200mm x 200mm x 100mm，用矩形创建空气盒



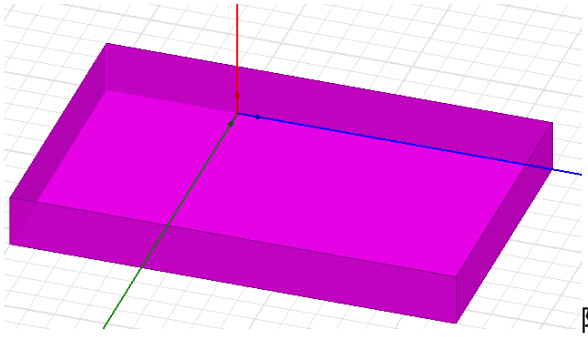


注意，到了这一步，发现以上模型尺寸设计有一定的出入，但是操作流程是一样的，下面我们用中心线的方式重新设计一次。

下面设计我就简化快捷

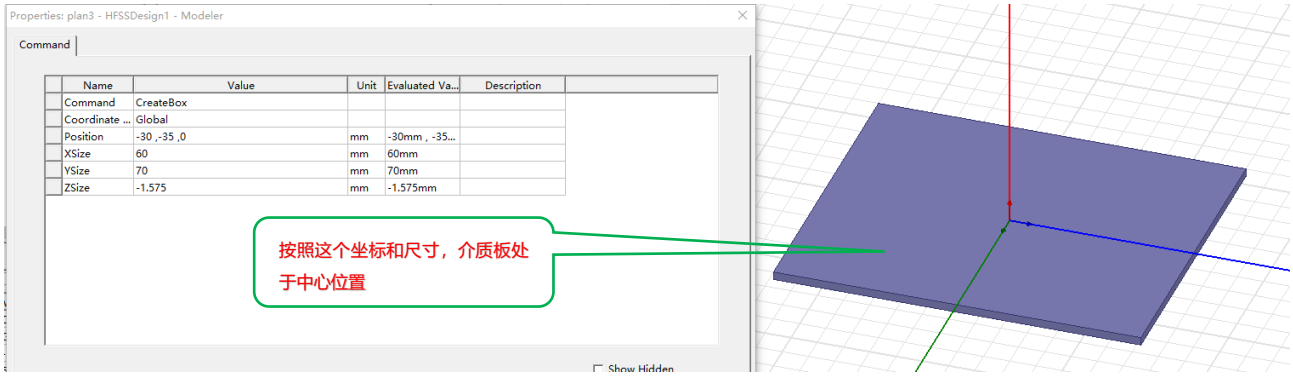


确认还是用模式驱动

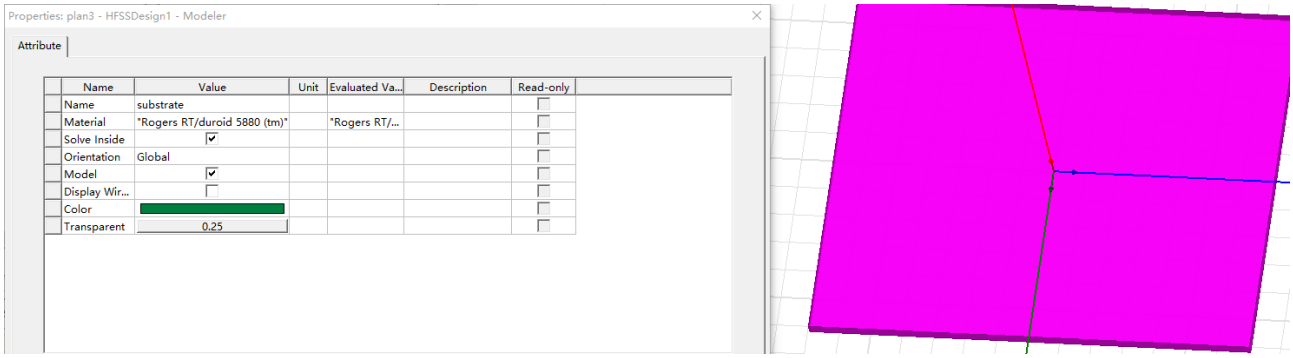


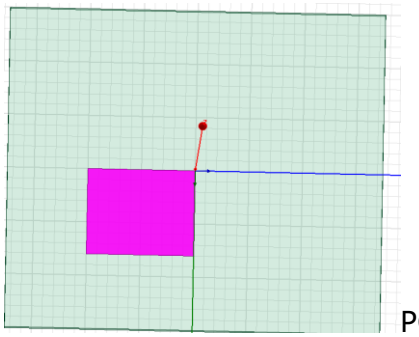
随意画一个立方体 PCB 基材(介质板)。也可以是其它板子，比如木板，塑料板。

设置介质板尺寸



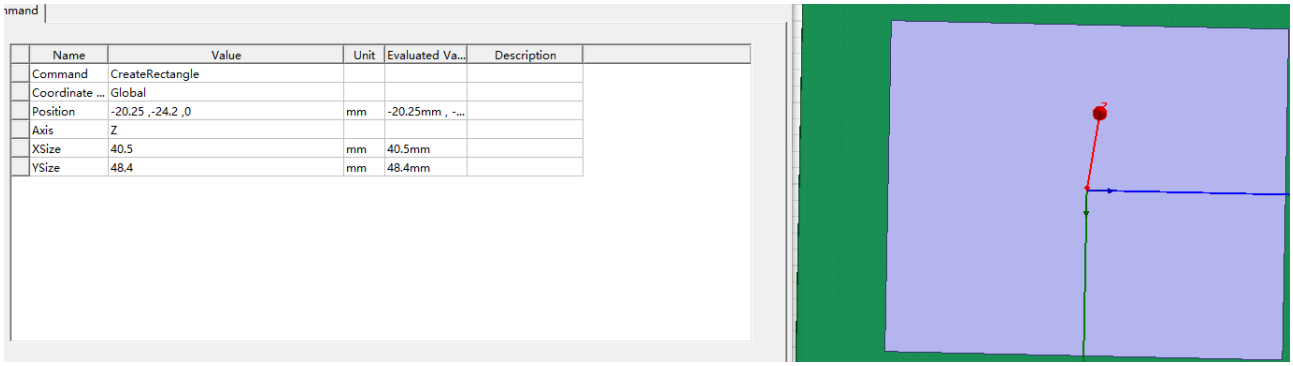
PCB 板基材选择





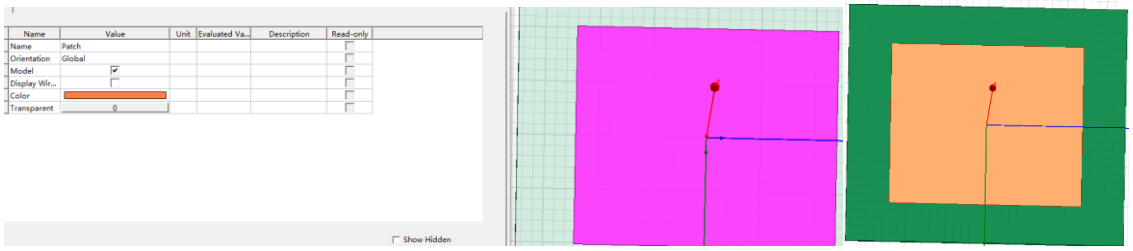
PCB 基材上随意画一个铜箔，用方形画，这里是仿真，就不需要设置厚度了，不然还需要用立方体来画。我这就用长方形

PCB 表面天线铜箔设置

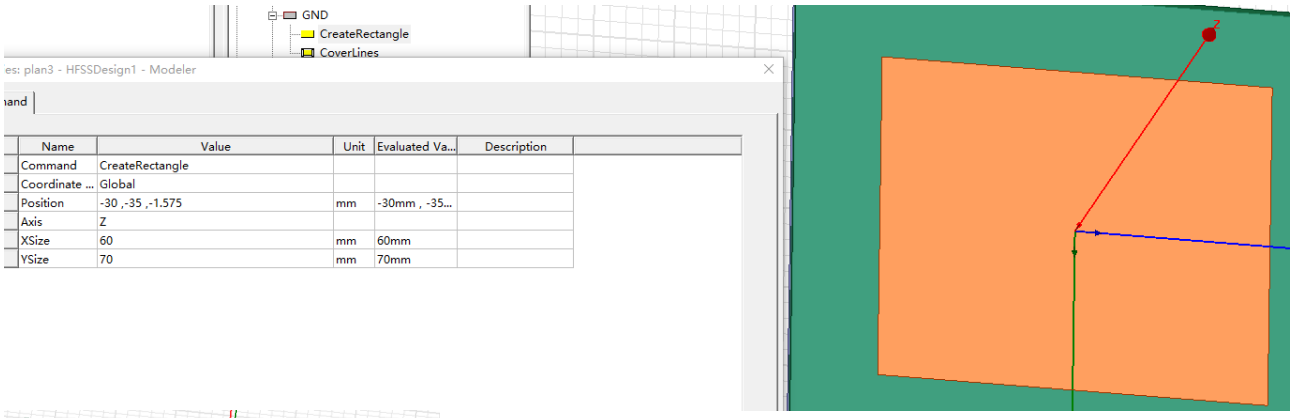


Name	Value	Unit	Evaluated Va...	Description
Command	CreateRectangle			
Coordinate ...	Global			
Position	-20.25 ,-24.2 ,0	mm	-20.25mm , -...	
Axis	Z			
XSize	40.5	mm	40.5mm	
YSize	48.4	mm	48.4mm	

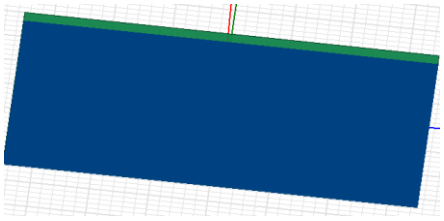
PCB 表面天线铜箔名称和颜色设置



PCB 地基材设置，就是画一个和 PCB 基材尺寸一样的铜，然后沉降到 PCB 基材背面

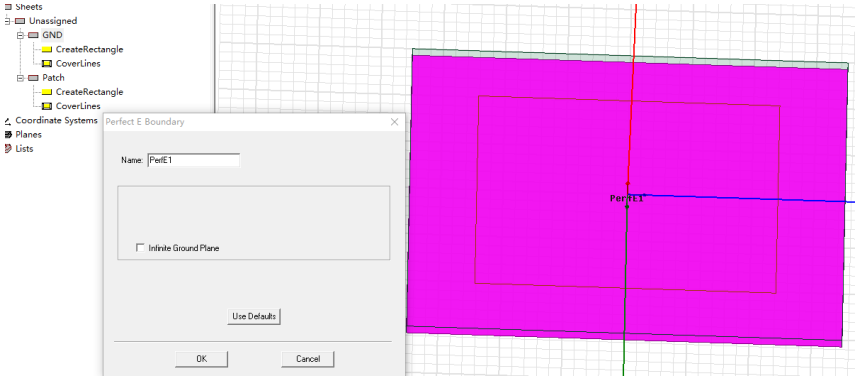


Name	Value	Unit	Evaluated Va...	Description
Command	CreateRectangle			
Coordinate ...	Global			
Position	-30 ,-35 ,-1.575	mm	-30mm , -35...	
Axis	Z			
XSize	60	mm	60mm	
YSize	70	mm	70mm	

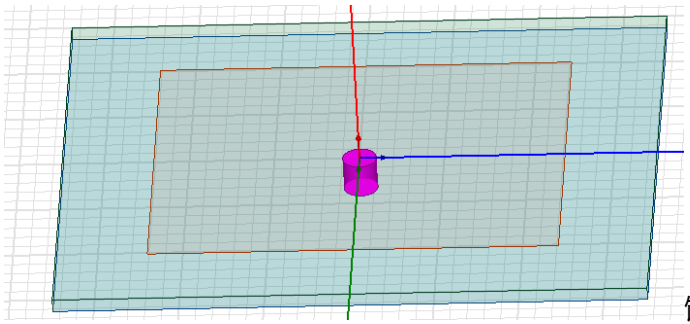


PCB 背面有 GND 地了

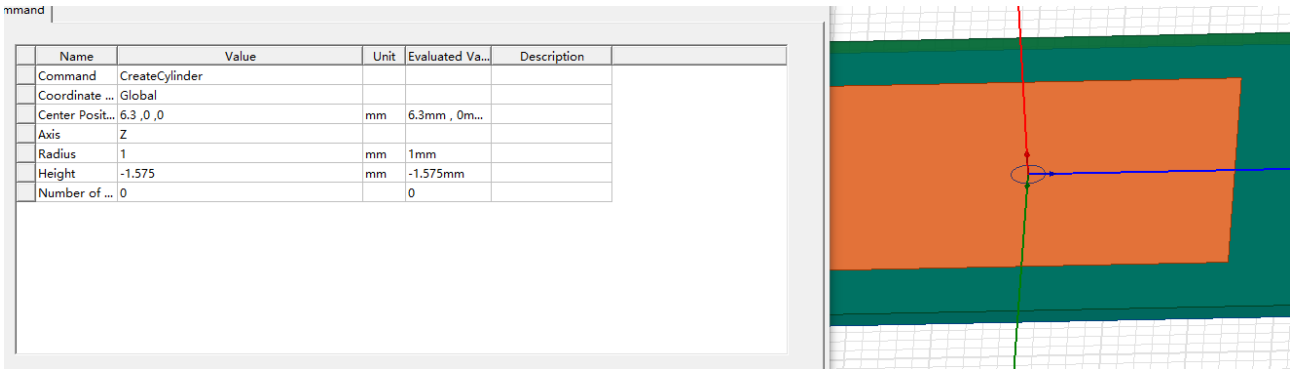
将 PCB 表层铜箔和 PCB 背面的地，设置成理想电导体。



馈点内芯设计



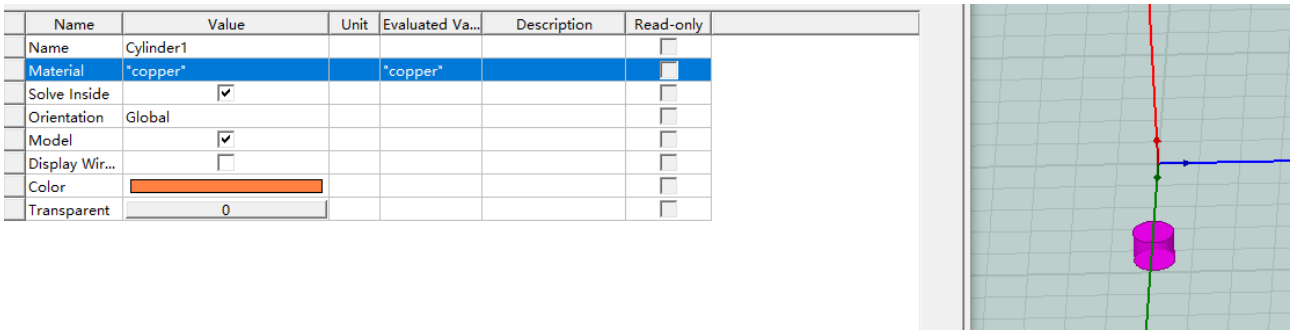
馈点就是一个圆柱体，穿透整个 PCB 板材和铜箔。用来焊接同轴电缆的。



Name	Value	Unit	Evaluated Va...	Description
Command	CreateCylinder			
Coordinate ...	Global			
Center Posit...	6.3 ,0 ,0	mm	6.3mm , 0m...	
Axis	Z			
Radius	1	mm	1mm	
Height	-1.575	mm	-1.575mm	
Number of ...	0		0	

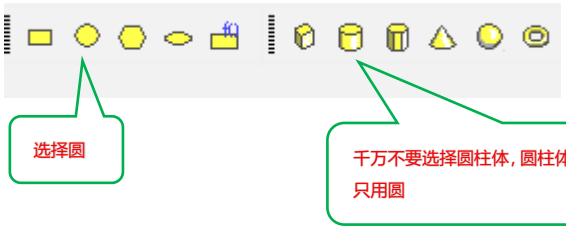
内芯圆柱体离中心 6.3mm，这个距离是前面计算之后的馈点位置，内芯直径 1mm，内芯圆柱体长度贯穿整个 PCB 基板所以使用-1.575mm 厚度。

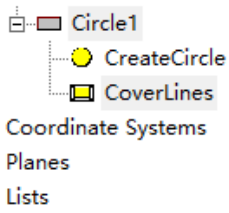
内芯材料设置为铜，copper

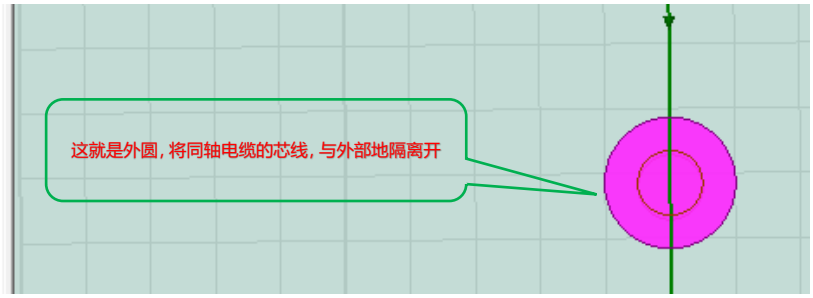


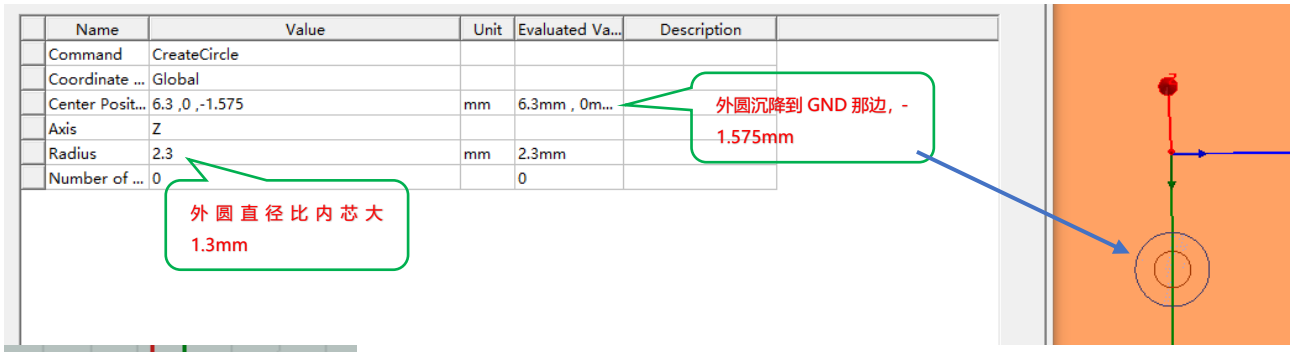
Name	Value	Unit	Evaluated Va...	Description	Read-only
Name	Cylinder1				
Material	"copper"		"copper"		
Solve Inside	☑				
Orientation	Global				
Model	☑				
Display Wir...	☐				
Color					
Transparent	0				

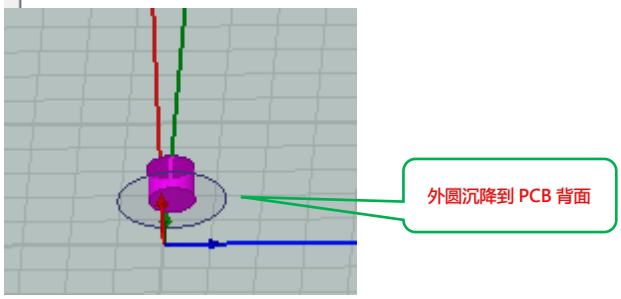
内芯外圆设计，就是连接同轴电缆外表皮的地(金属丝)



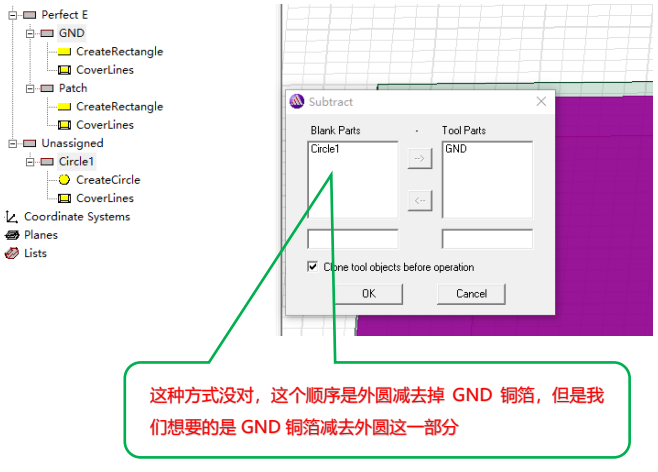


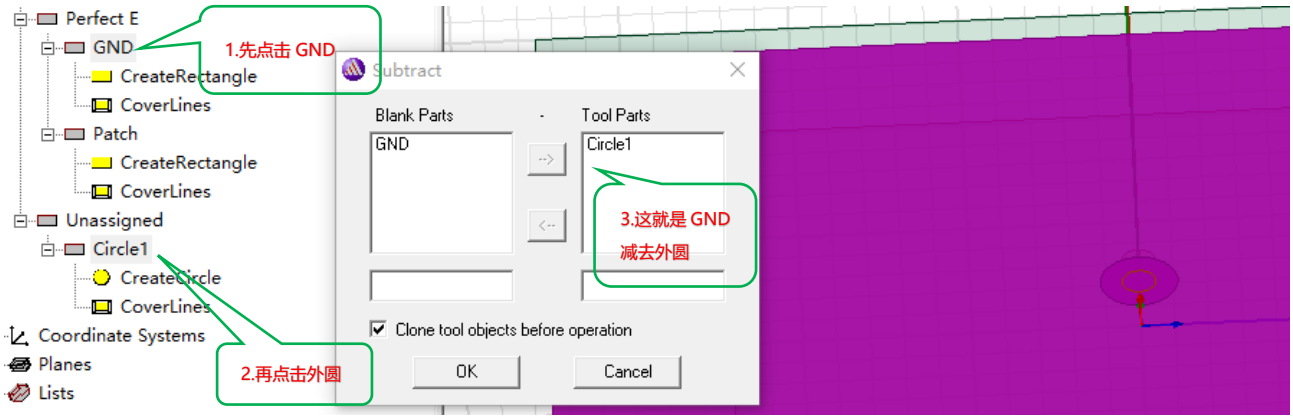




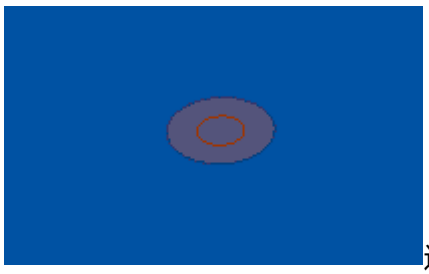


将外圆和 PCB 背面的地，连接在一起，外圆与内芯进行隔离。



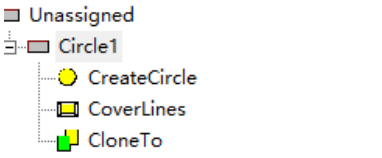


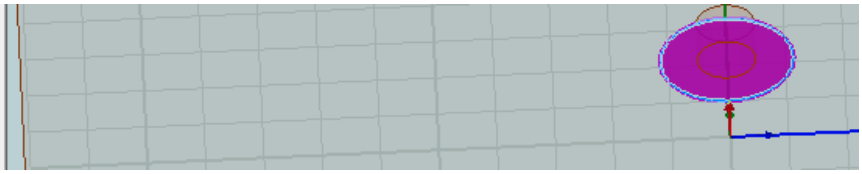
这样就形成 GND 整个铜箔只剪切掉外圆部分，其余的 GND 还是保留。

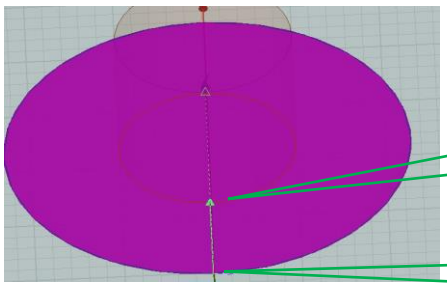


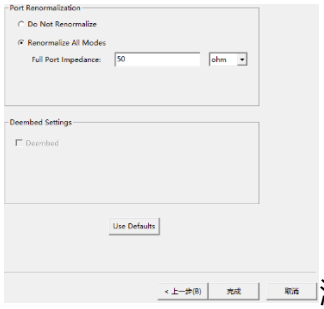
这就是对了。只有外圆的部分剪切掉。

点击外圆，进行端口设置



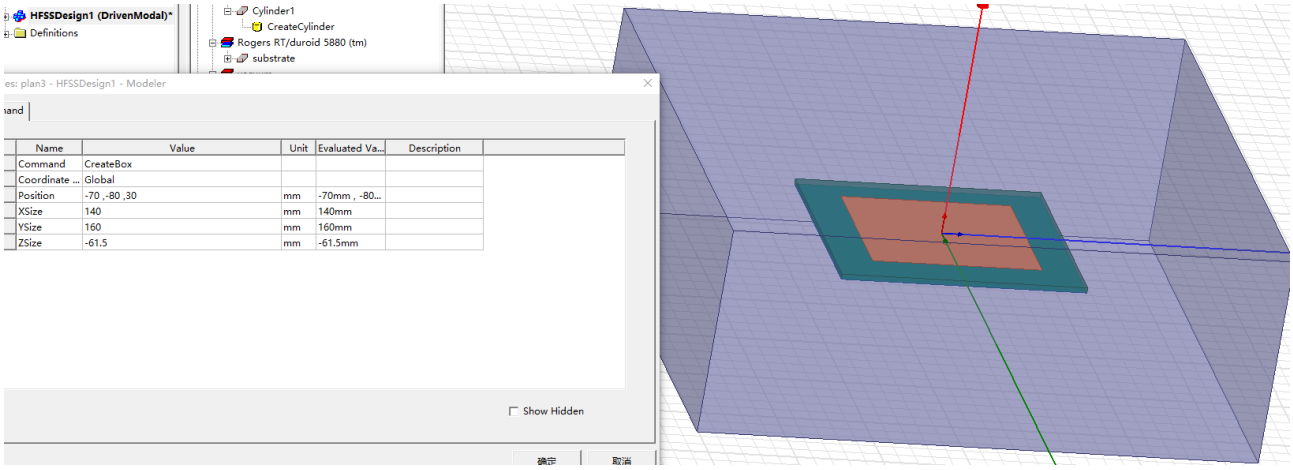




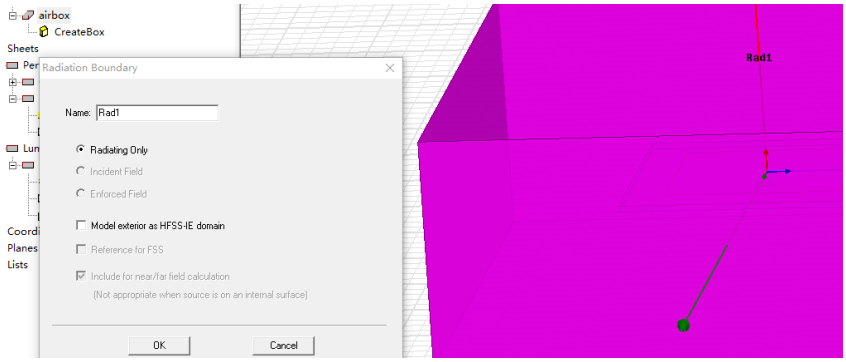


激励端口设置完成。

加入空气盒子

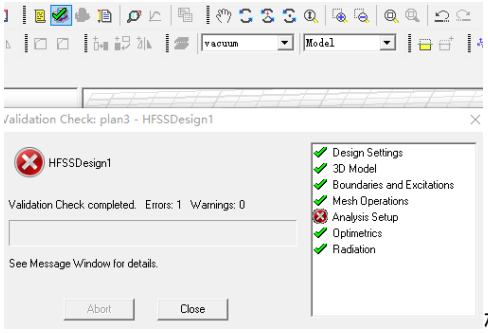


设置空气盒子辐射边界



如果想要隐藏某个模型

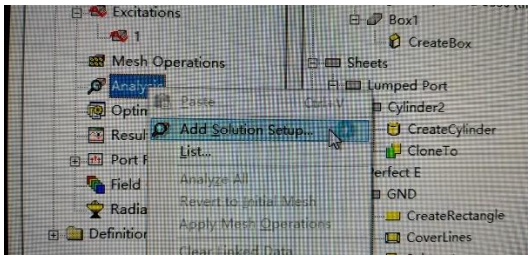
如果向要还原某个模型

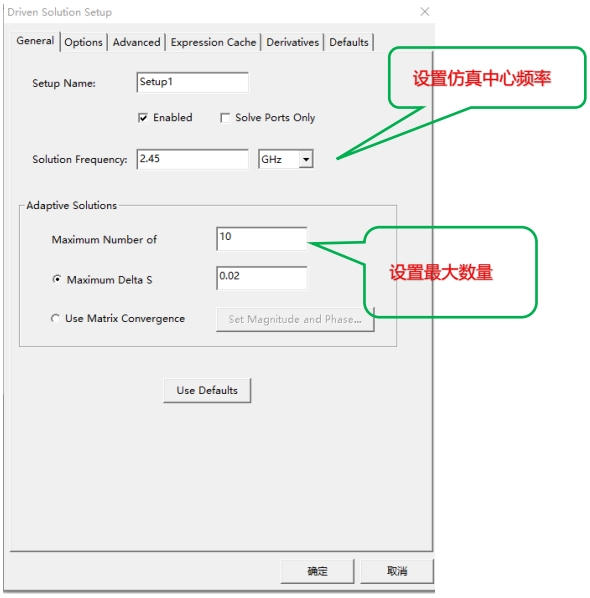


检查一下是否可以仿真，发现仿真的频率参数还未设置

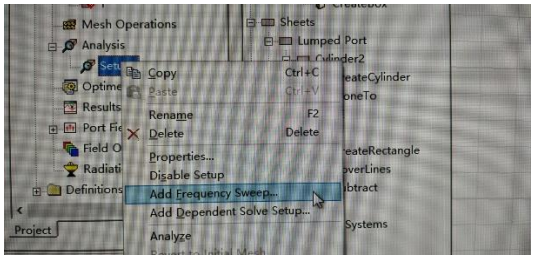
下面来设置求解器，完善频率参数

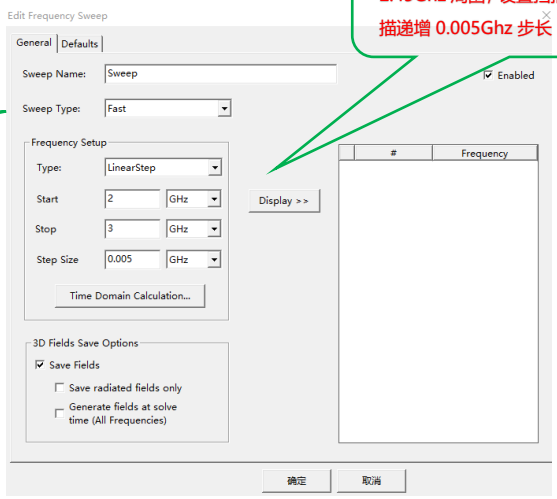
设置求解器

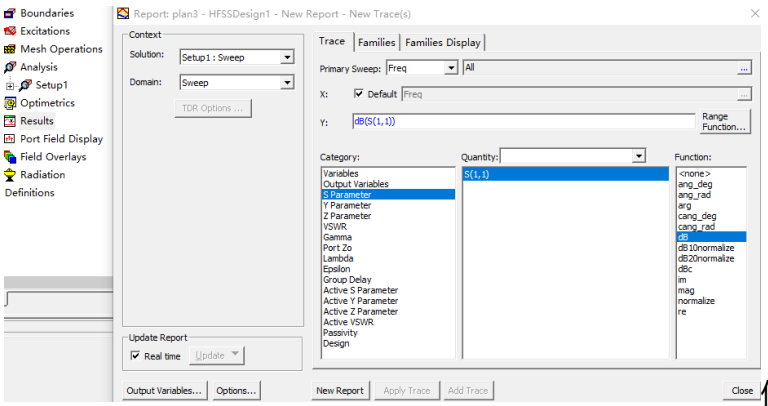




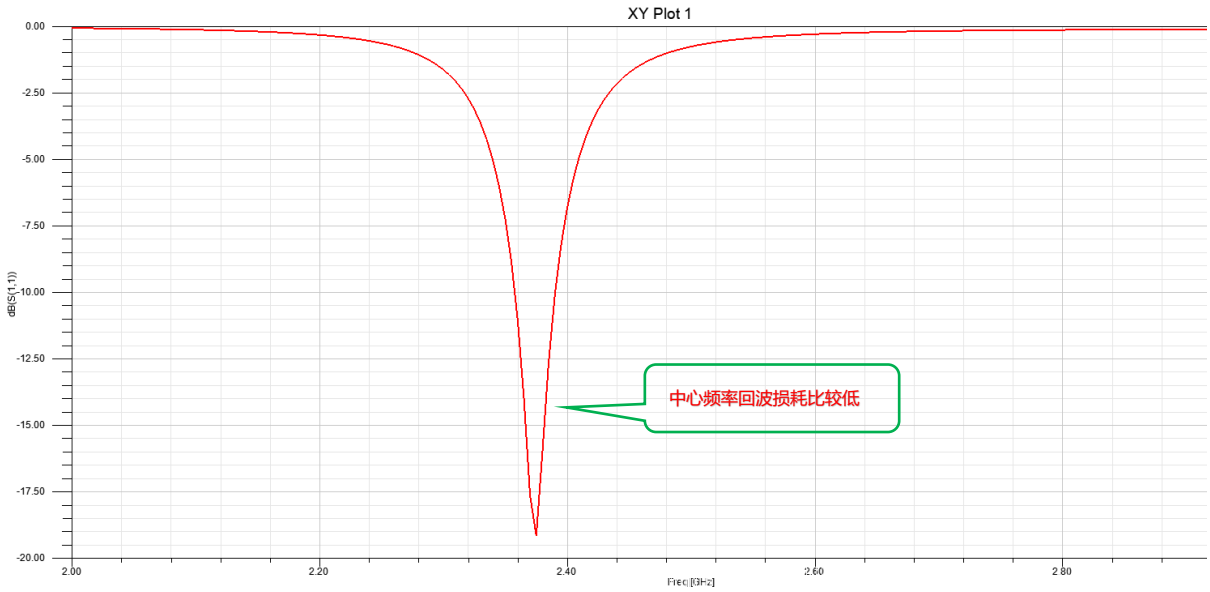
设置扫频范围



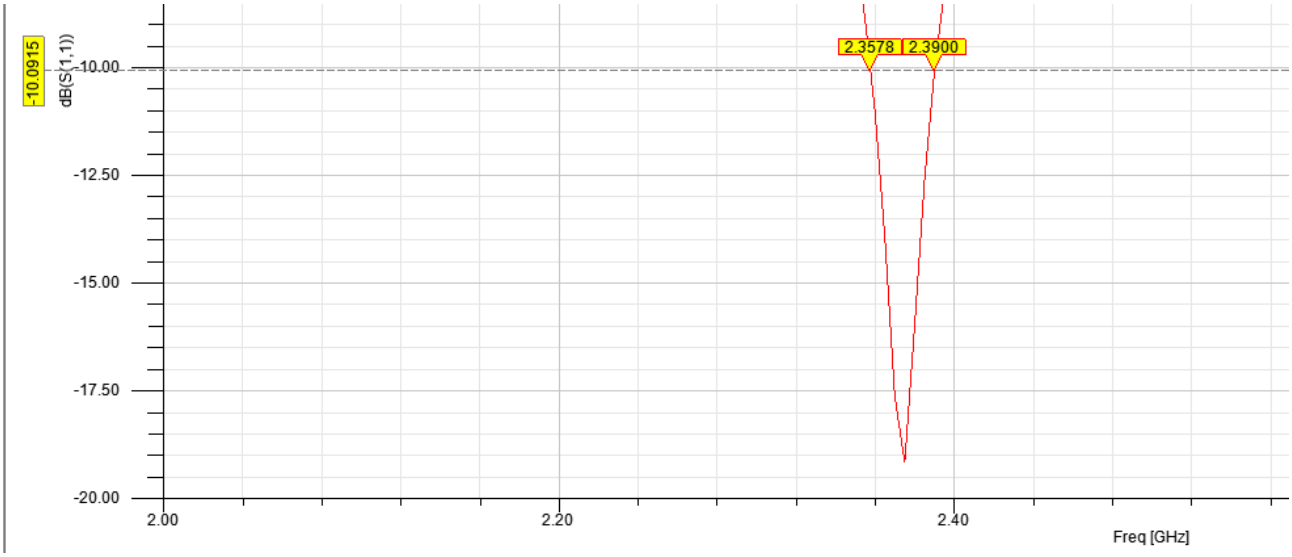




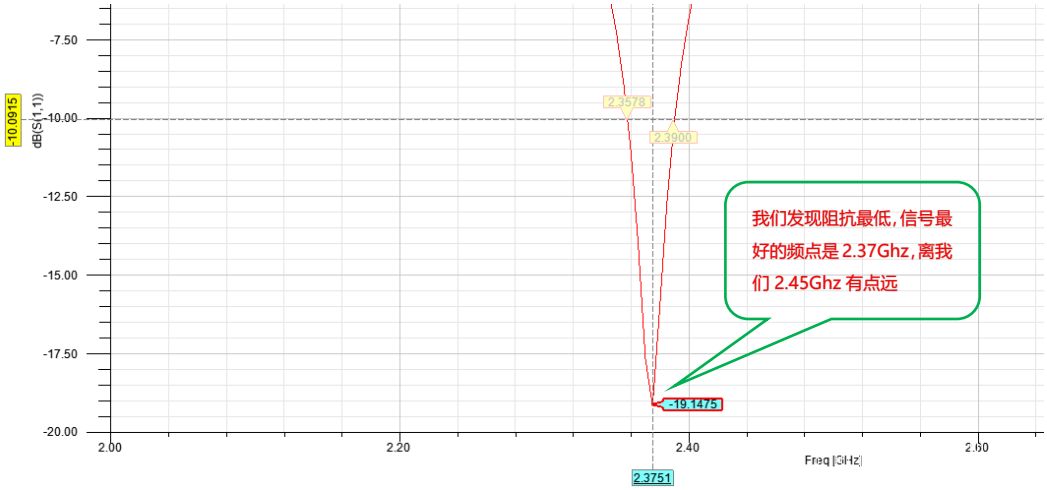
仿真完成，查看 S11(回波损耗)



鼠标右键->Marker ->Add Y Maker



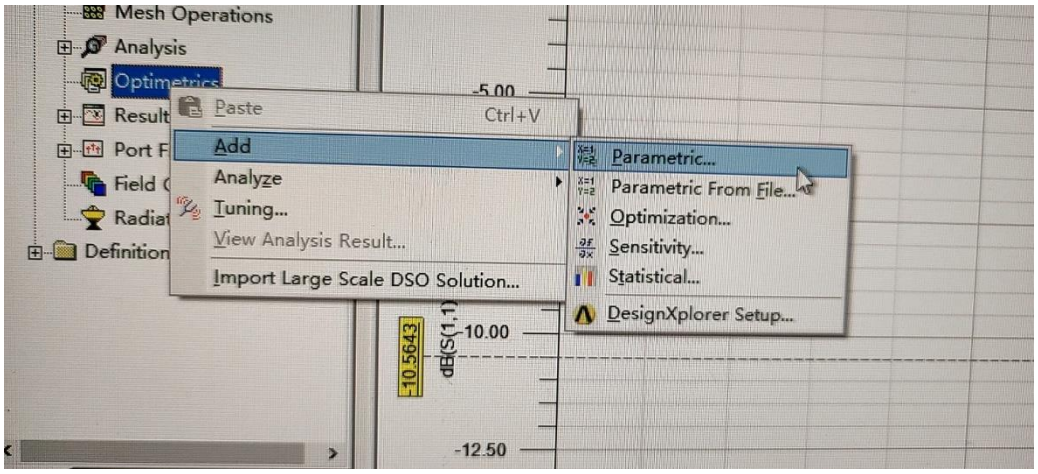
在 2.35Ghz ~ 2.39Ghz 回波损耗都在-10dB，表示信号 9/10 都入射进天线，只有 1/10 被反射回来，证明效果很好。



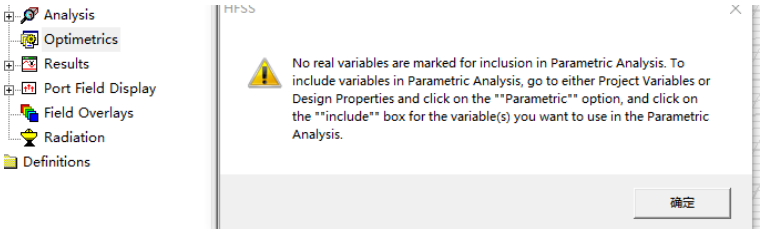
所以前面的表面贴片天线设计公式只是经验公式，不是绝对的，所以存在误差。

优化天线参数

我们来改善上面 2.37Ghz 离 2.45Ghz 较远的问题。



Parametric...是优化参数选项。我们现在修改天线表面铜箔的宽度，看看有没有频率改善。

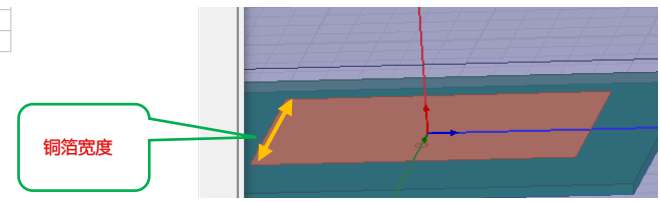


优化参数功能打开，出现报错。系统要求你必须有需要优化的变量，就是将导线宽度和 PCB 厚度等等....用变量来表示，我前面为了方便，都是直接写值，所以没有定义变量。所以我干脆直接修改模型尺寸来手动优化。

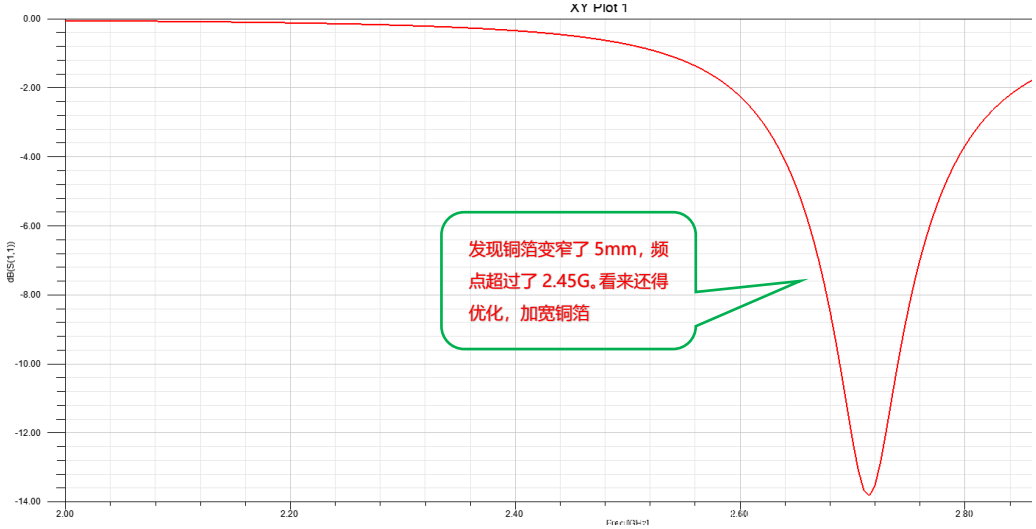
	XSize	40.5	mm	40.5mm
	YSize	48.4	mm	48.4mm

本身铜箔宽度 40.5mm

	XSize	35	mm	35mm
	YSize	48.4	mm	48.4mm

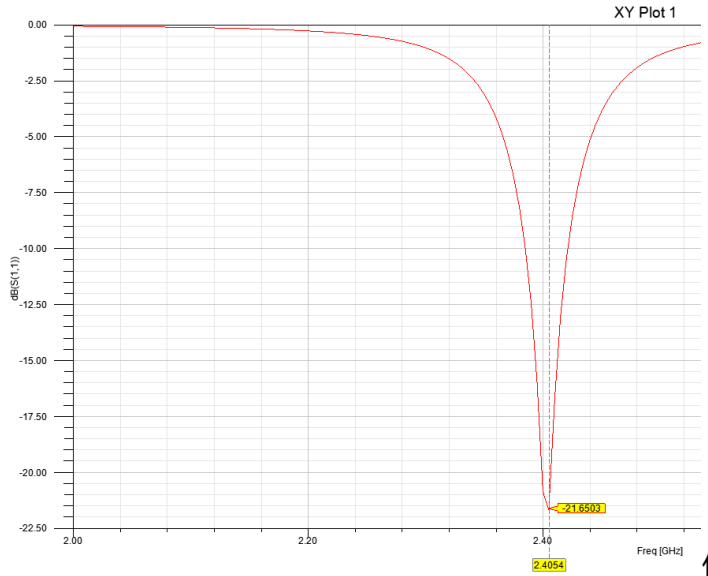


将铜箔宽度改成 35mm，然后重新仿真。



	XSize	40	mm	40mm	
	YSize	48.4	mm	48.4mm	

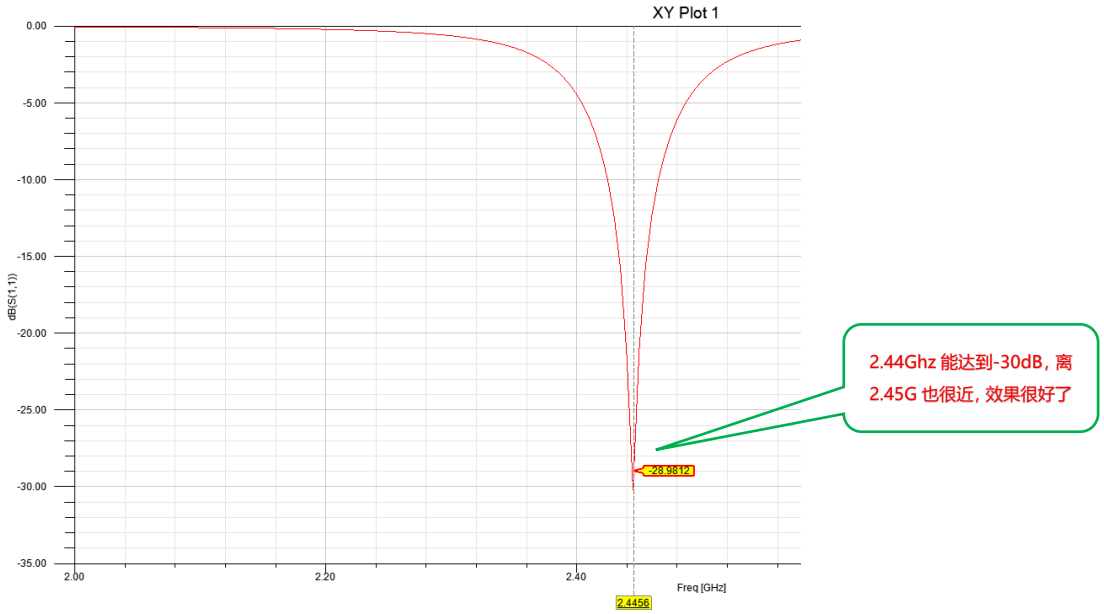
把铜箔宽度 XSize 减小 0.5mm，就剩 40mm



你看，0.5mm 的铜箔就可以让频点上移这么多，从 2.375Ghz 移动到 2.40Ghz

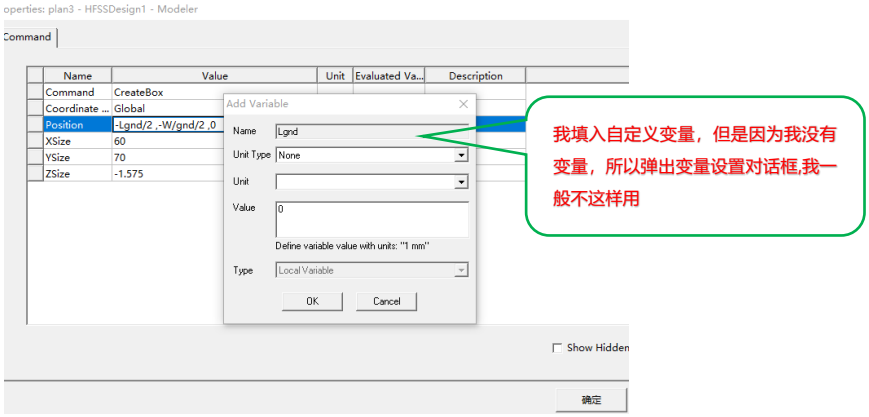
	XSize	39.3	mm	39.3mm	
	YSize	48.4	mm	48.4mm	

改成 39.3mm 宽度

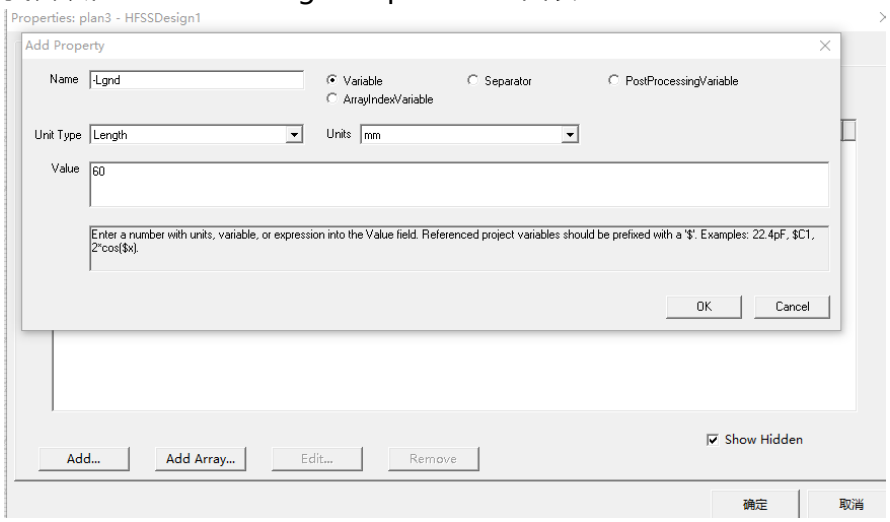


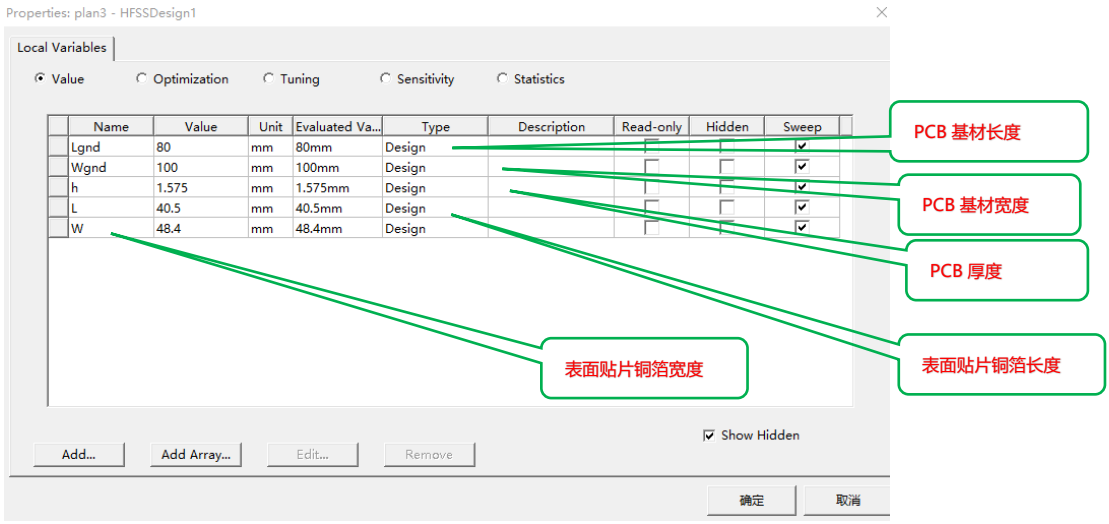
我觉得这种改参数，然后重新仿真太麻烦了，我们还是加入变量，来做自动参数优化。

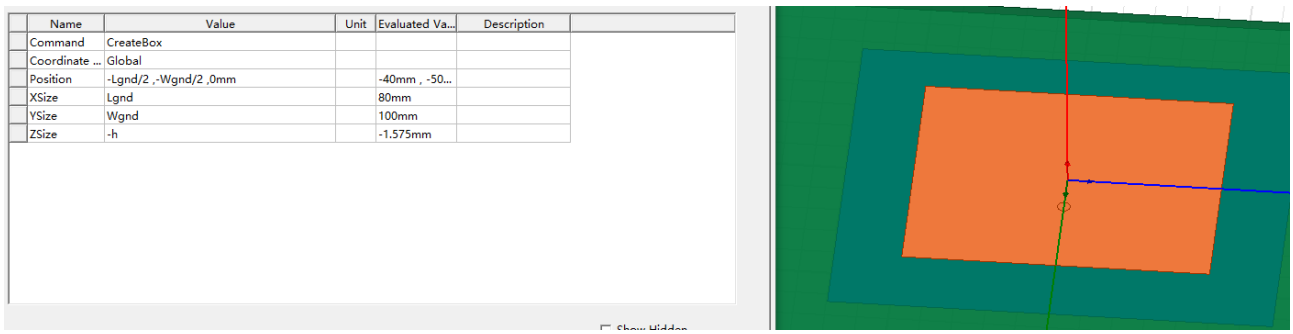
1.修改 PCB 板基材尺寸



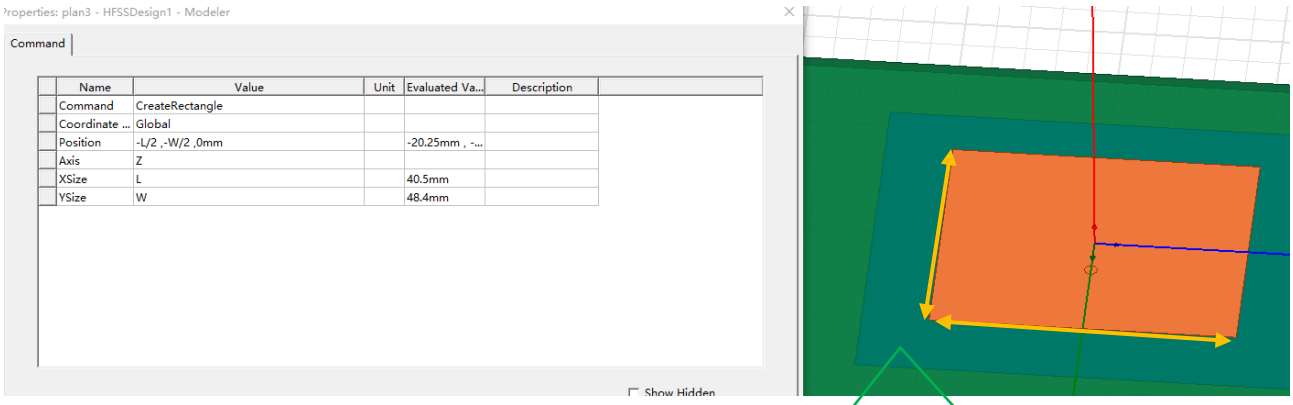
我喜欢在 HFSS -> Design Properties 里面设置





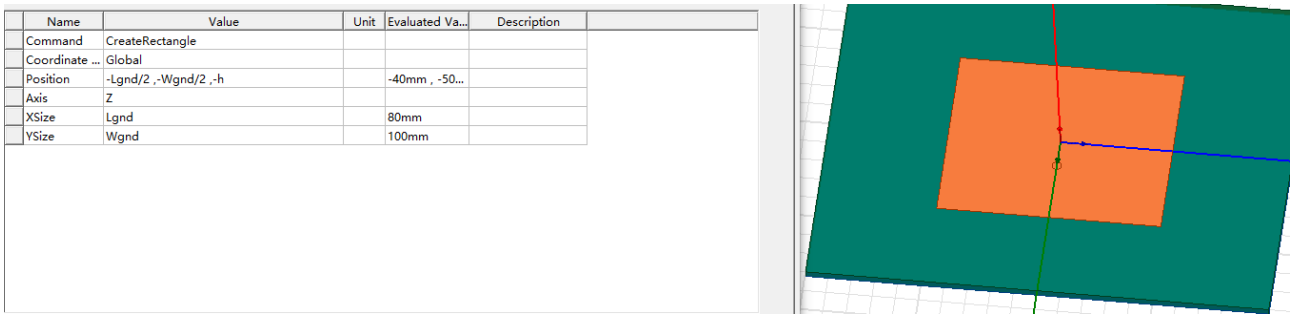


PCB 基材尺寸

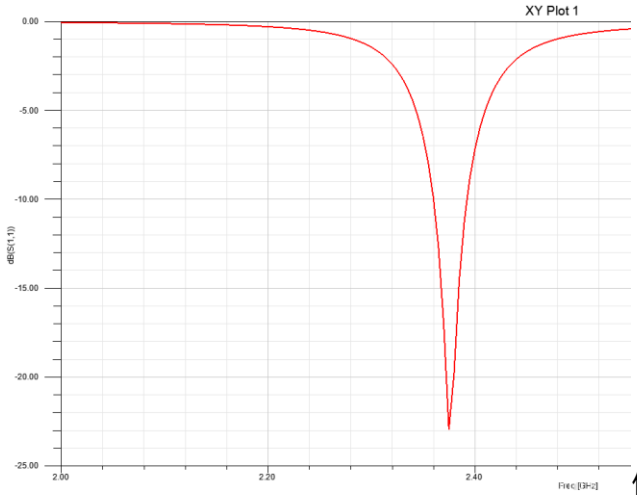


表贴铜箔尺寸

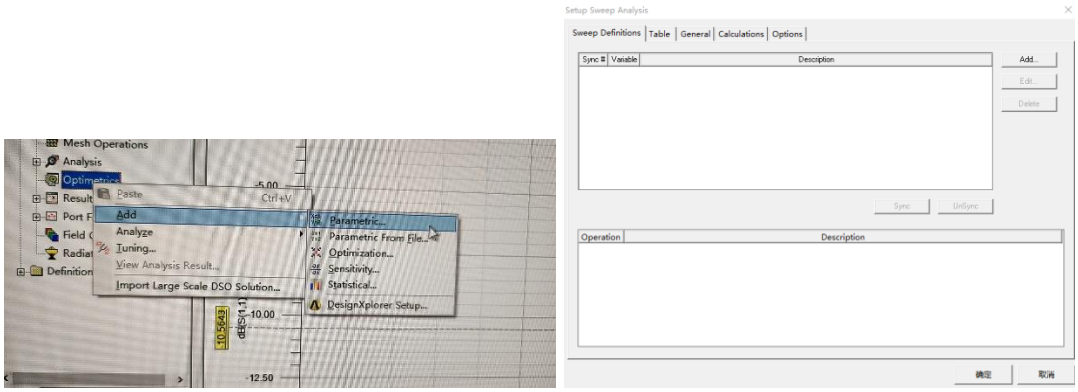
我发现背面 GND 尺寸比 PCB 基材小，修改下



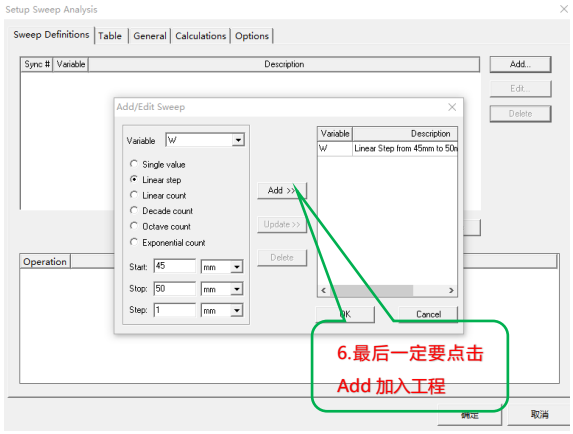
背面 GND 铜箔尺寸也修改了，其余的过孔这些不用去修改，可以开始仿真了

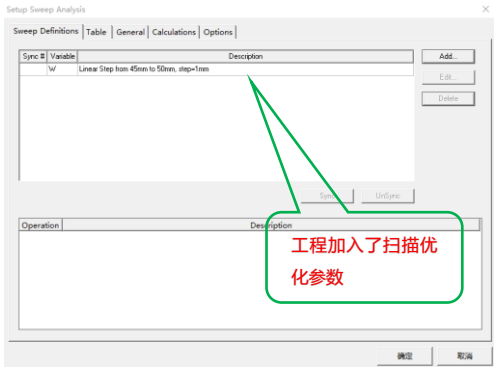


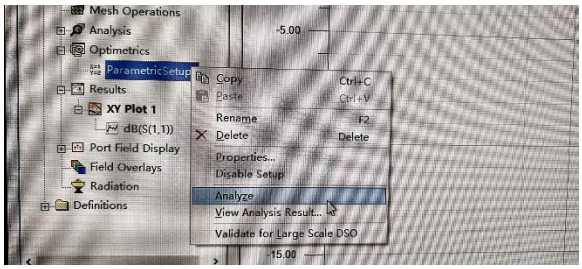
仿真结果和前面一样，下面开始进行仿真优化



这次点击仿真优化就不会报错了，因为有了 Lgnd，Wgnd，L，W，h 等变量。

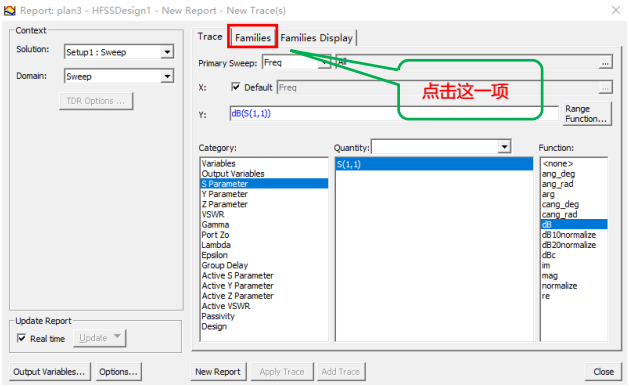


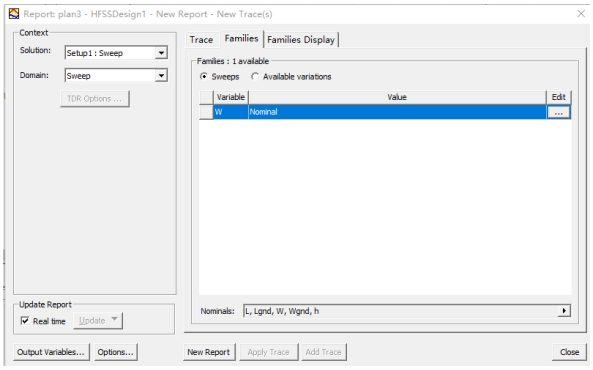


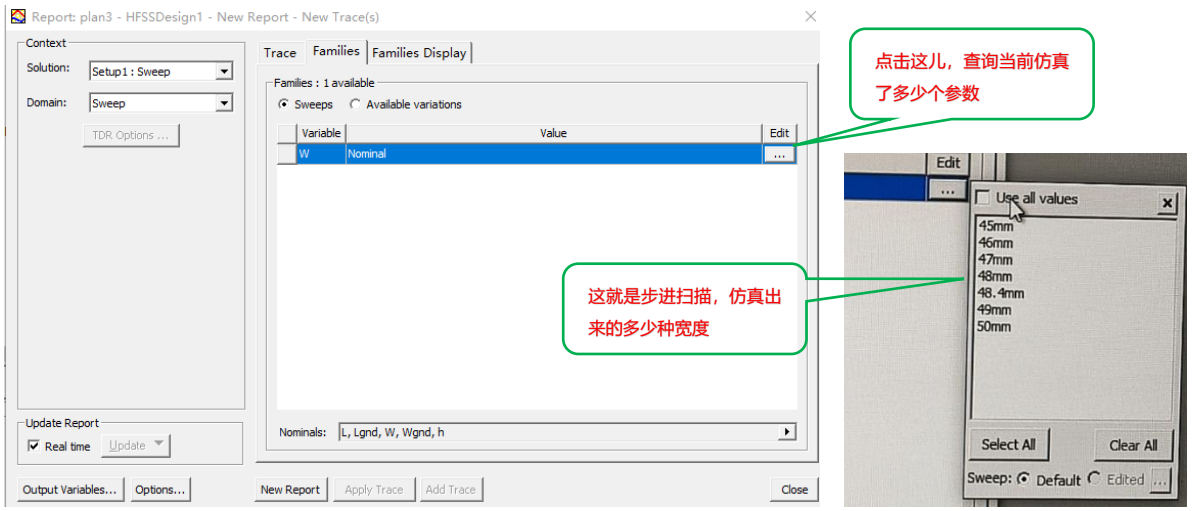


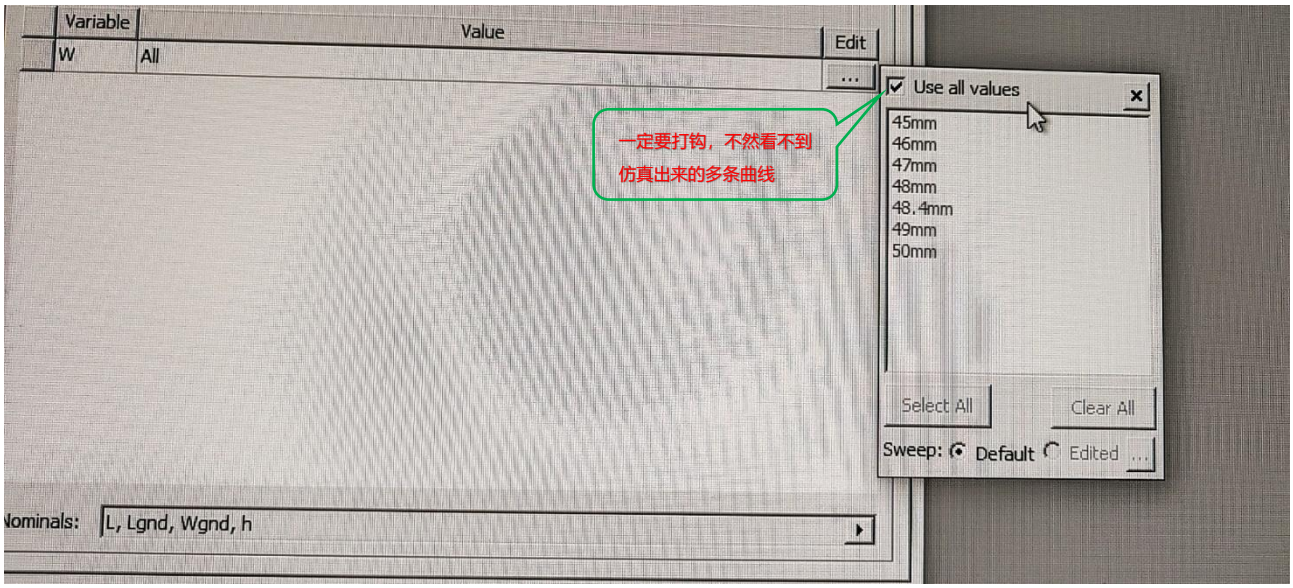
开始优化仿真，仿真时间很长，我们可以变仿真边看。

点击 Results->Create Modal Solution.... ->Rectangular Plot 生成报告

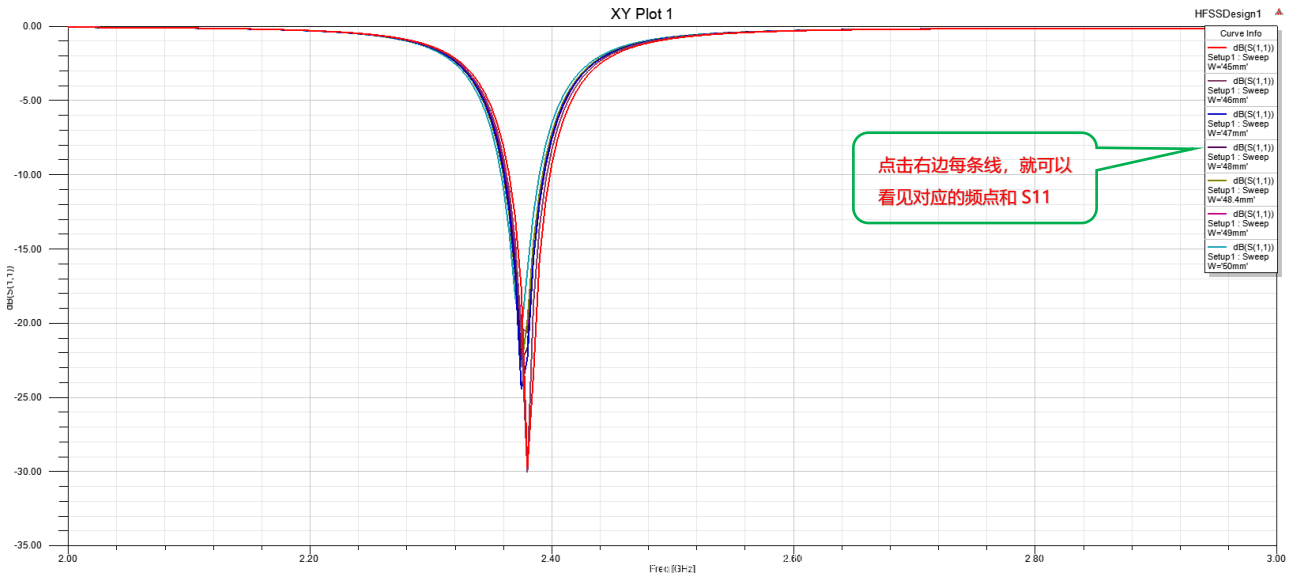


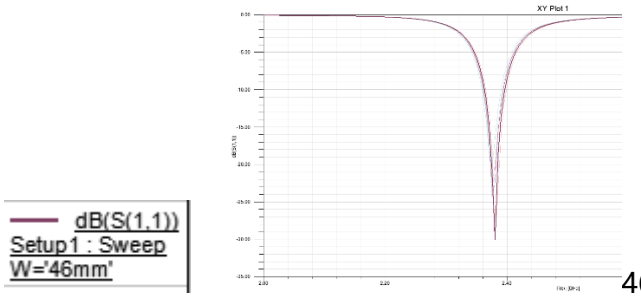






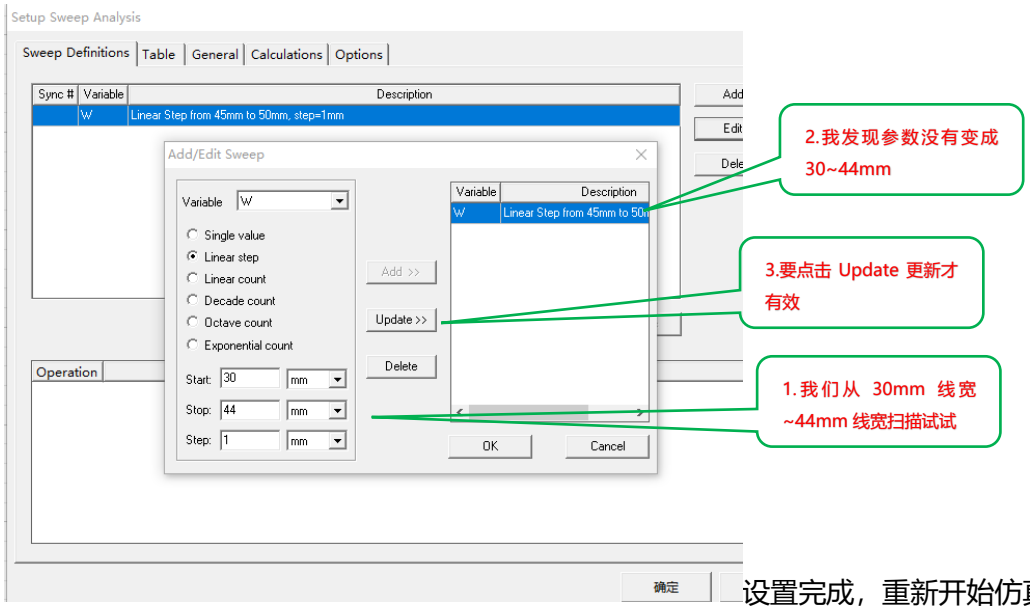
点击 New Report，生成仿真优化曲线



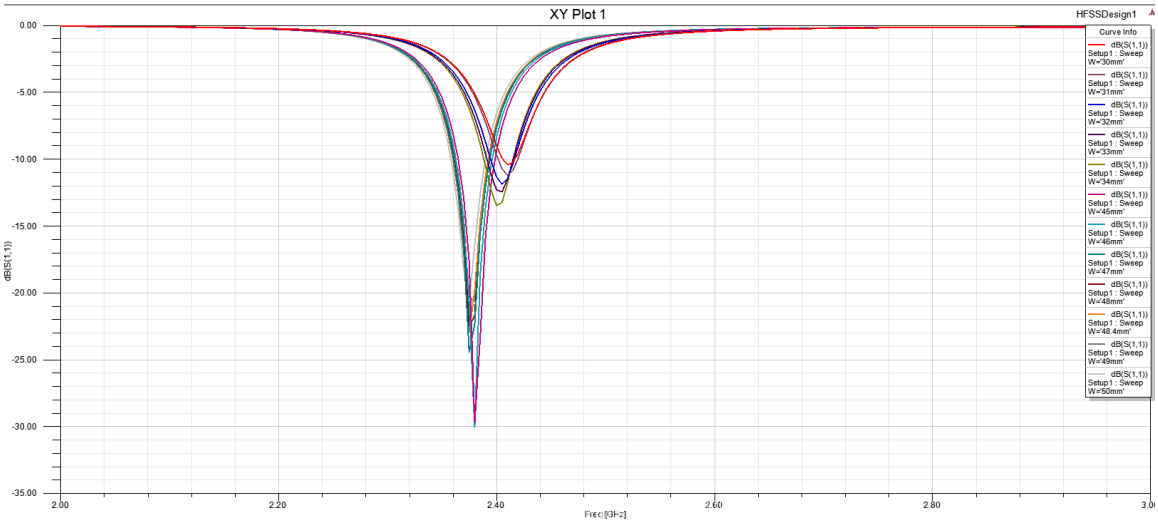


46mm 线宽效果最好，但是还是没达到 2.45Ghz 频点

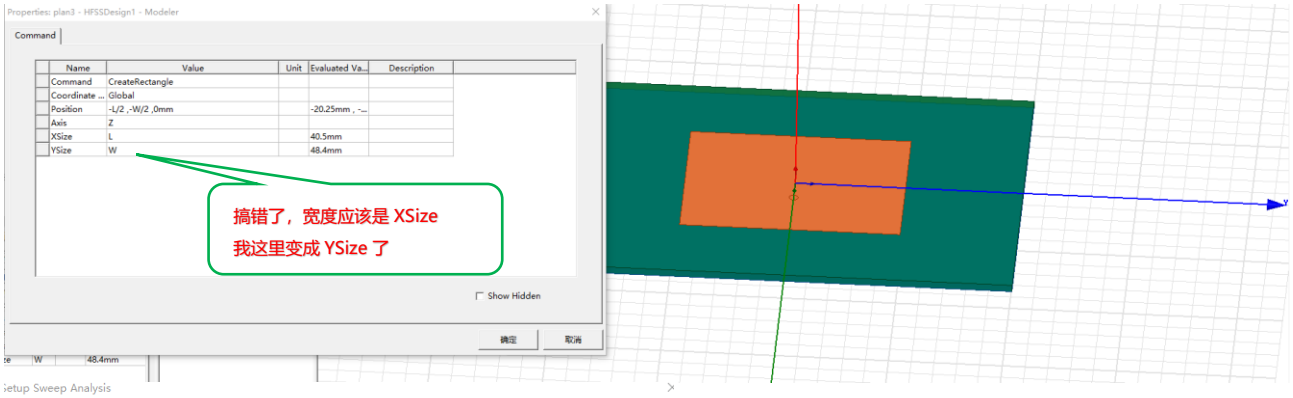
证明 45~50mm 的线宽范围都达不到要求。

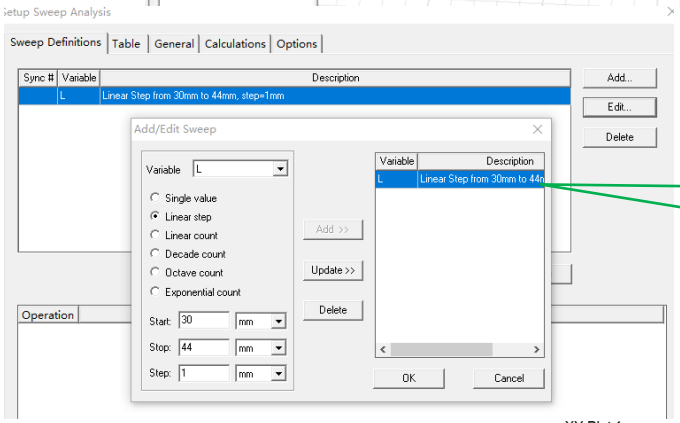


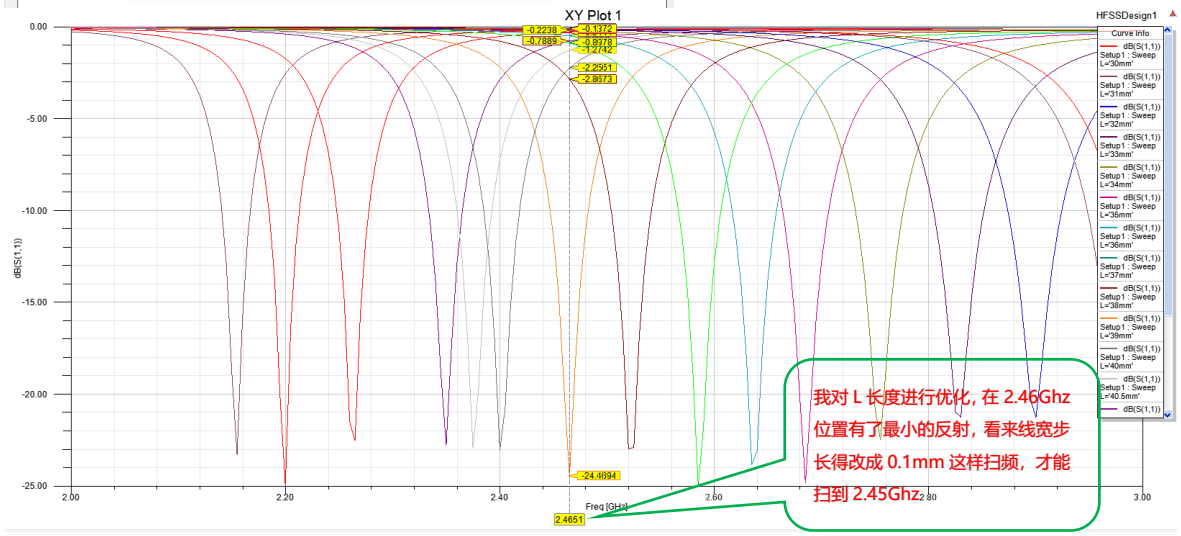
设置完成，重新开始仿真优化



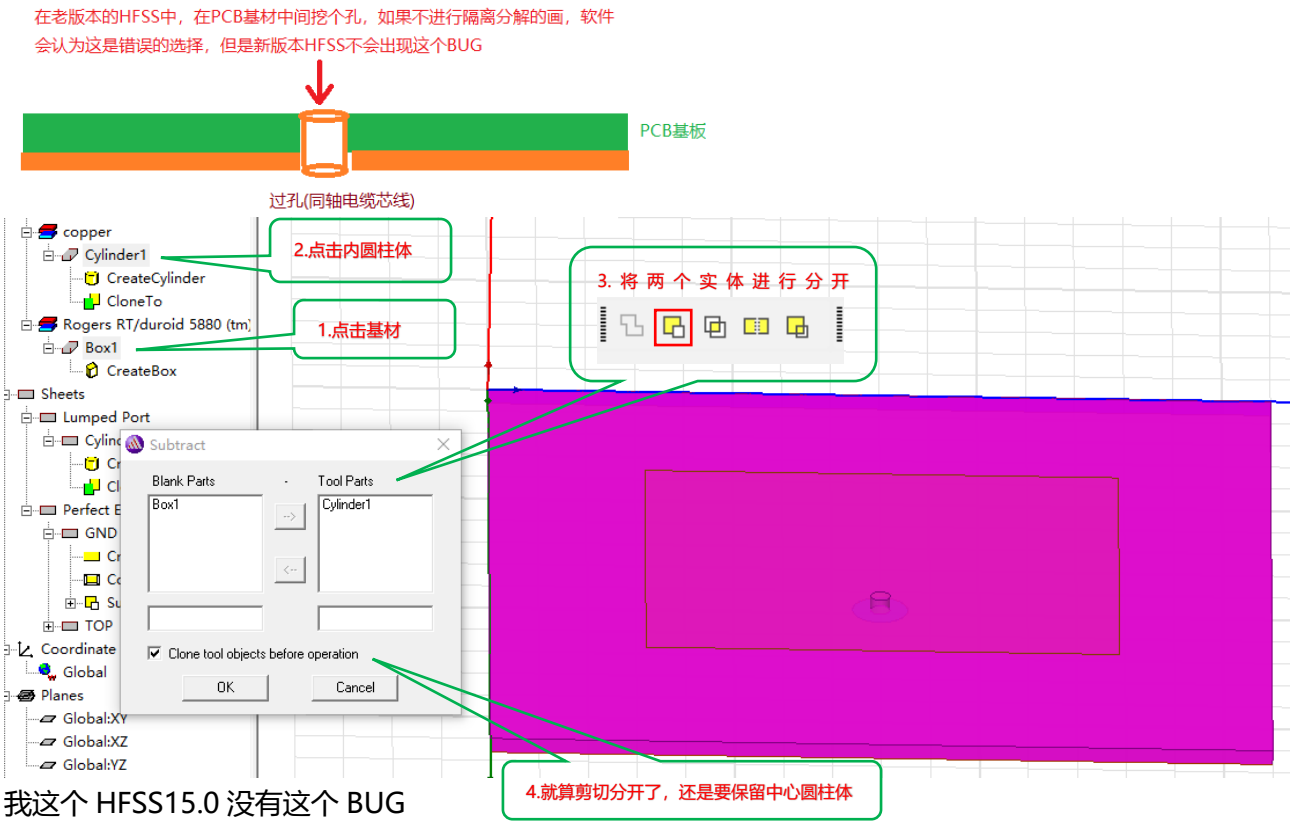
我发现仿真之后这么多曲线，都离 2.45Ghz 很远，我是不是把表面贴片铜箔的宽度和长度搞反了？





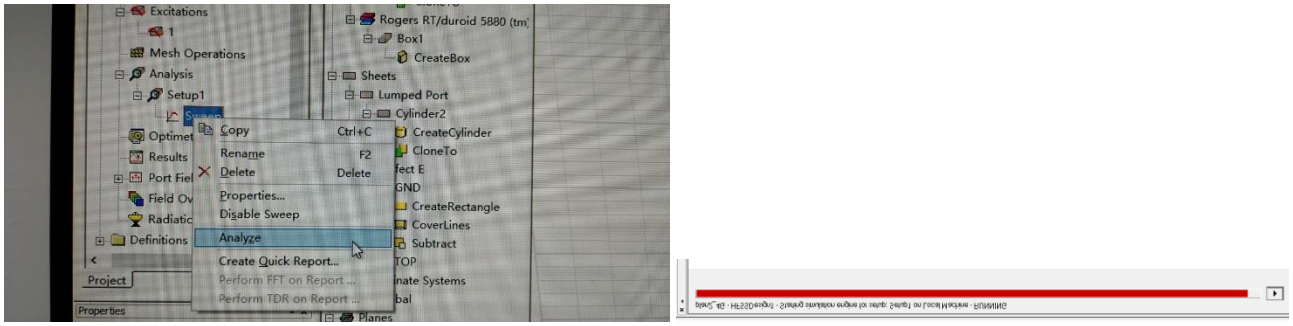


这里要提醒一下，有些老版本的软件会在 Analysis 仿真的时候出现过孔和基板不兼容的 BUG

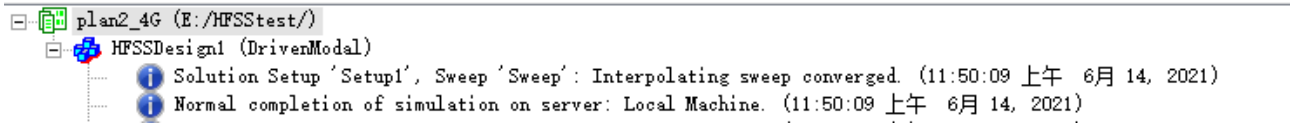


我这个 HFSS15.0 没有这个 BUG

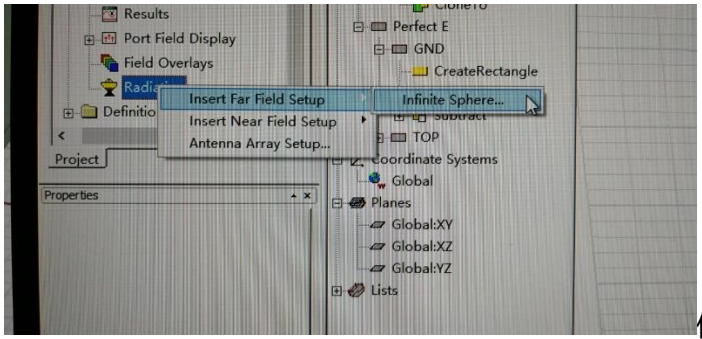
开始仿真



仿真显示红色进度条是正确的，我这个 HFSS 版本是红色进度条

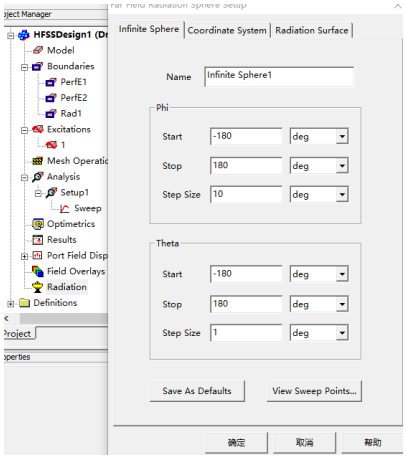


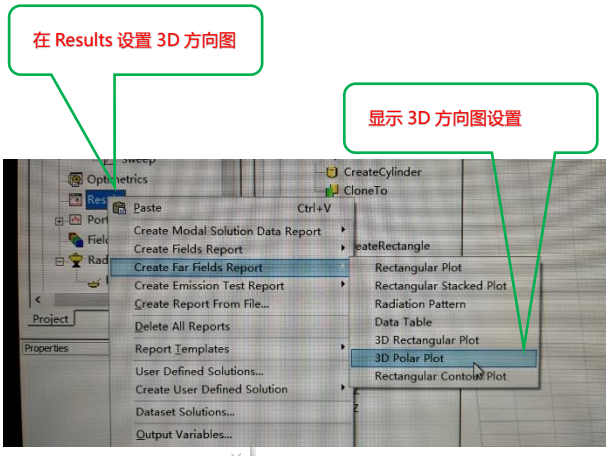
仿真结果没有出现 Error 就证明可以使用。

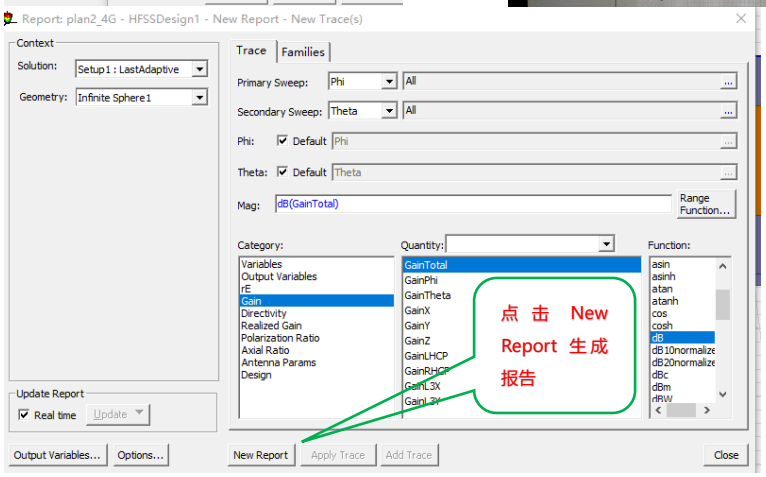


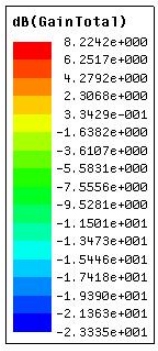
仿真完成后可以设置辐射方向

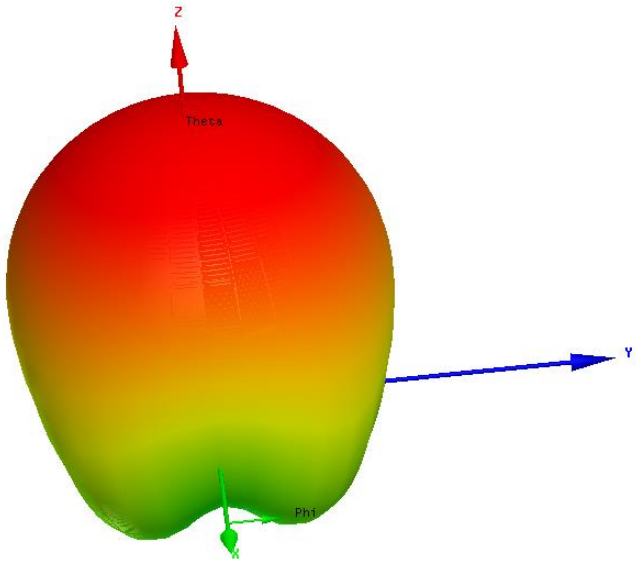
查看天线辐射方向图


Infinite Sphere
Coordinate System
Radiation Surface
Name
Infinite Sphere1
Phi
Start
-180
Stop
180
Step Size
10
Theta
Start
-180
Stop
180
Step Size
1
Save As Defaults
View Sweep Points...

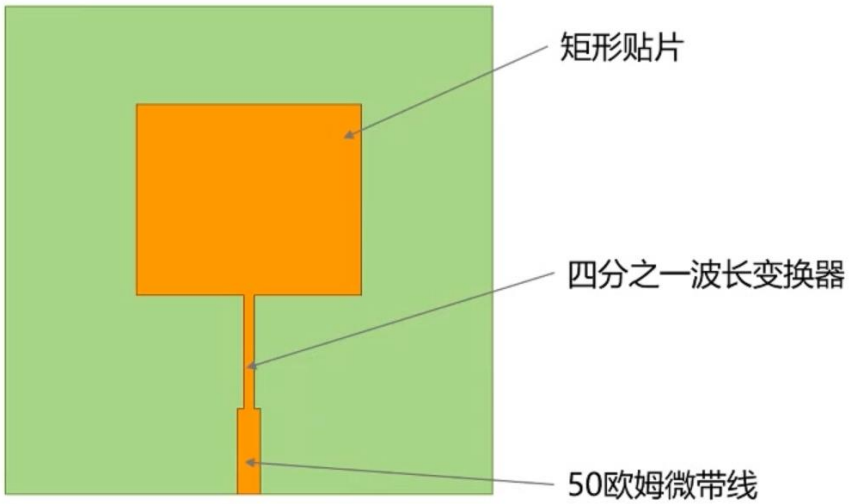

在 Results 设置 3D 方向图
显示 3D 方向图设置
Paste
Ctrl+V
Create Modal Solution Data Report
Create Fields Report
Create Far Fields Report
Create Emission Test Report
Create Report From File...
Delete All Reports
Report Templates
User Defined Solutions...
Create User Defined Solution
Dataset Solutions...
Output Variables...
Rectangular Plot
Rectangular Stacked Plot
Radiation Pattern
Data Table
3D Rectangular Plot
3D Polar Plot
Rectangular Contour Plot
CreateCylinder
CloneTo


Report: plan2_4G - HFSSDesign1 - New Report - New Trace(s)
Context
Solution:
Setup1 : LastAdaptive
Geometry:
Infinite Sphere1
Trace
Families
Primary Sweep:
Phi
All
Secondary Sweep:
Theta
All
Phi:
Default
Theta:
Default
Mag:
dB(GainTotal)
Range Function...
Category:
Quantity:
Function:
Variables
Output Variables
rE
Gain
Directivity
Realized Gain
Polarization Ratio
Axial Ratio
Antenna Params
Design
GainTotal
GainPhi
GainTheta
GainX
GainY
GainZ
GainLHCP
GainRHCP
asin
asinh
atan
atanh
cos
cosh
dB
Update Report
Real time
Update
点 击 New Report 生成报告
Output Variables...
Options...
New Report
Apply Trace
Add Trace
Close

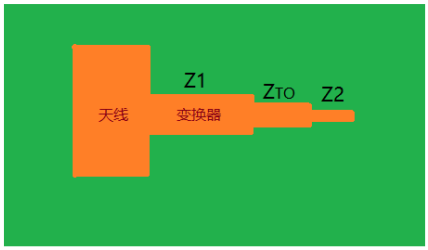

dB(GainTotal)
8.2242e+000
6.2517e+000
4.2792e+000
2.3068e+000
3.3429e-001
-1.6382e+000
-3.6107e+000
-5.5831e+000
-7.5556e+000
-9.5281e+000
-1.1501e+001
-1.3473e+001
-1.5446e+001
-1.7418e+001
-1.9390e+001
-2.1363e+001
-2.3335e+001


Z
Theta
Y
Phi
X

微带天线铜皮馈线仿真(侧馈)



四分之一波长变换器讲解

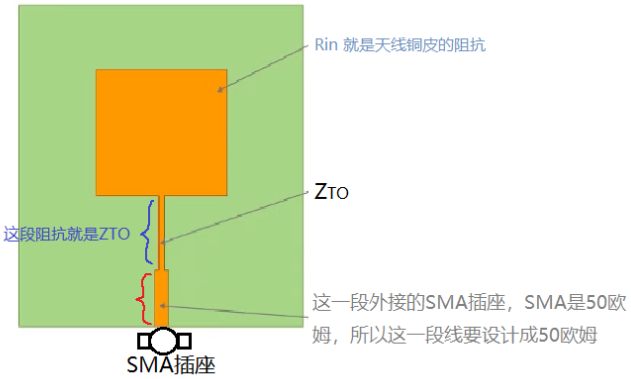


$$Z_{TO} = \sqrt{Z1 \times Z2}$$

$$\lambda = \frac{c}{f}$$ 光速 / 频率

W = 线宽

$$G = \begin{cases} \dfrac{W^2}{90\lambda_0^2} & w \le \lambda 0 \quad \text{如果W < 真空中波长，就用这行公式} \\ \dfrac{W^2}{120\lambda_0^2} & w > \lambda 0 \quad \text{如果W > 真空中波长，就用这行公式} \end{cases}$$



$$Yin = 2G$$

$$Rin = \frac{1}{Yin}$$ Rin 天线铜皮阻抗

$$Z_{TO} = \sqrt{50 \times Rin}$$ ZTO就是中间线段阻抗，就是我们要调试的阻抗变换器

实例操作:

如:光速 c = 10^{8}

天线工作频率 ; 2.45Ghz

PCB基材介电常数 : Er = 2.2

PCB基材板厚: 1.575mm

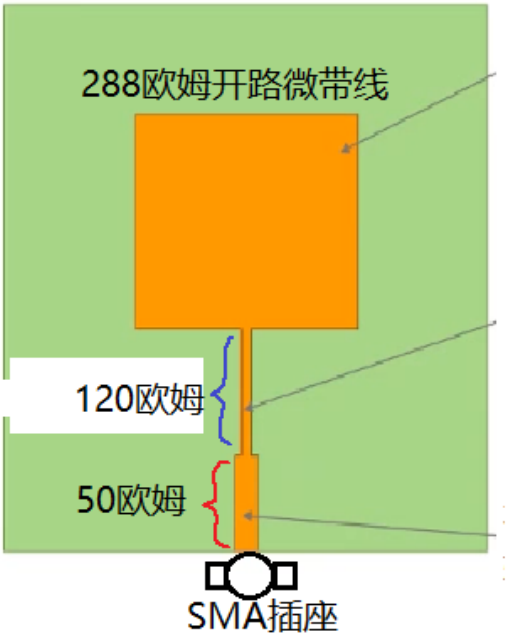
$$W = \frac{c}{2f}\sqrt{\frac{2}{\varepsilon r + 1}}$$ 线宽就用前面计算微带贴片天线的公式 W = 48.4mm

L = 天线长度 (也用前面计算微带贴片天线的公式) L = 40.5mm

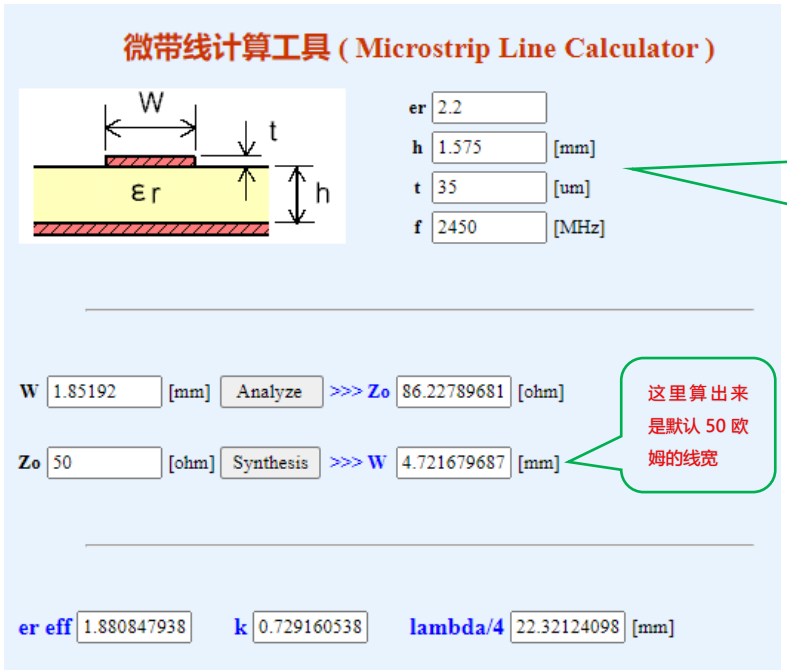
G 根据前面公式选择计算

阻抗 Rin = 288 欧姆

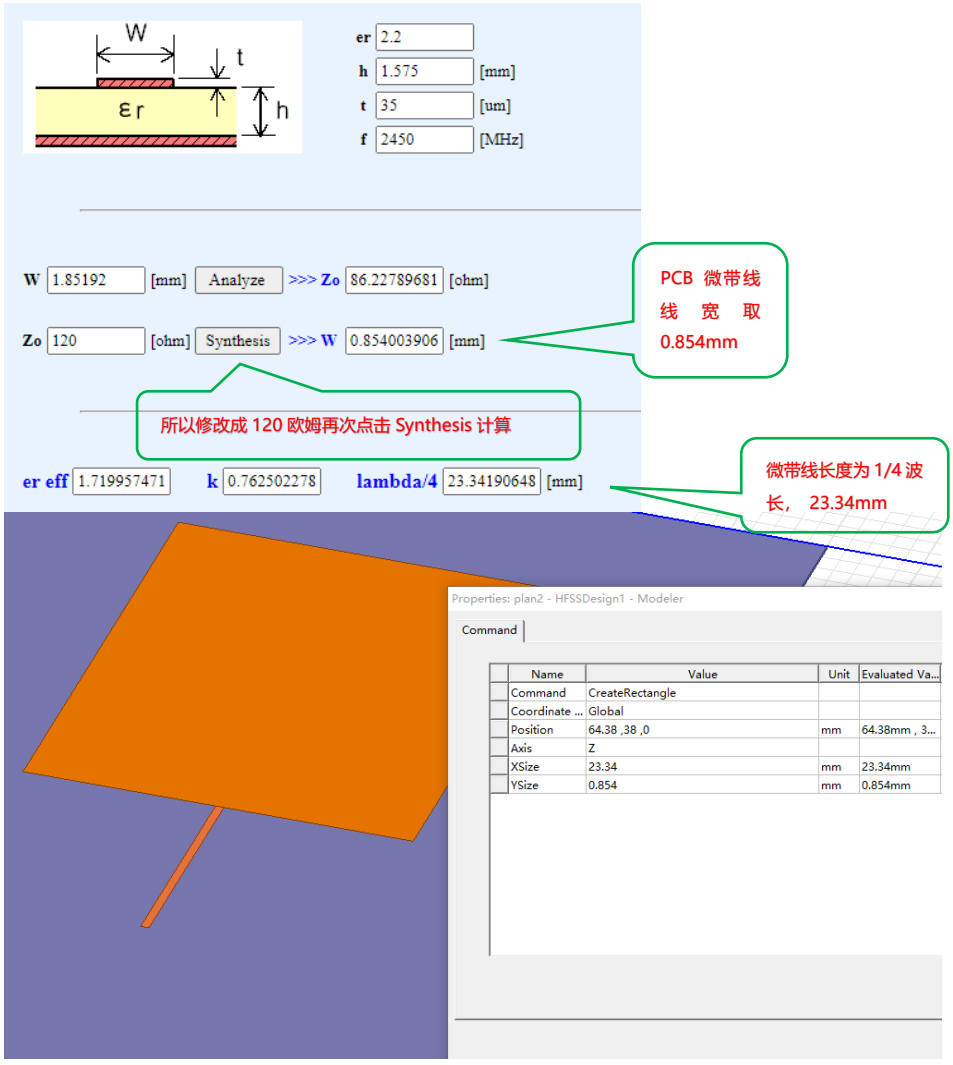
阻抗 ZTo = 120 欧姆

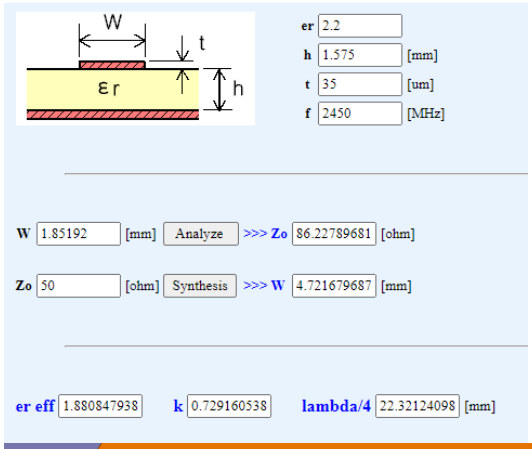


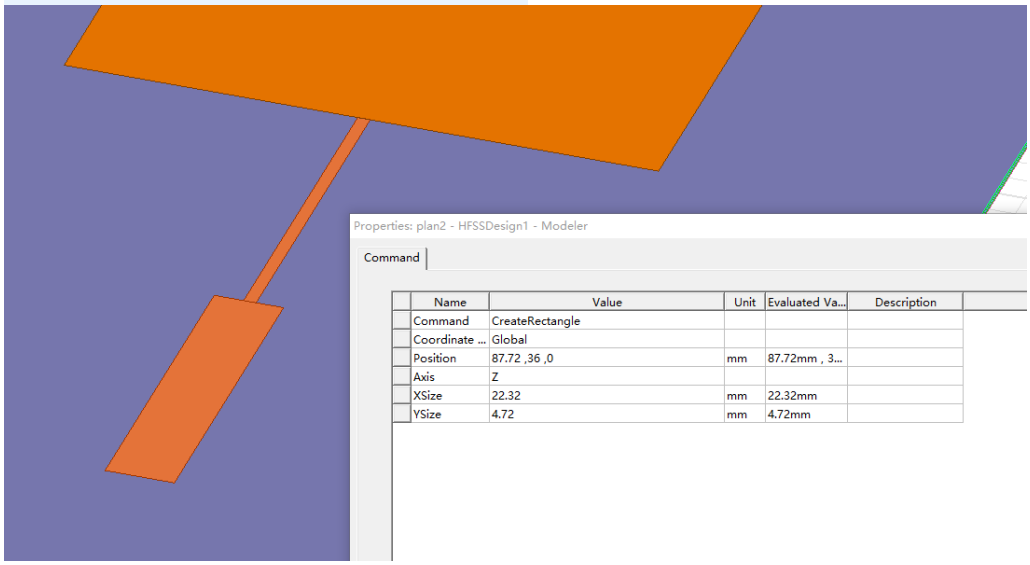
我们在微波 EDA 网站找到微带线计算工具

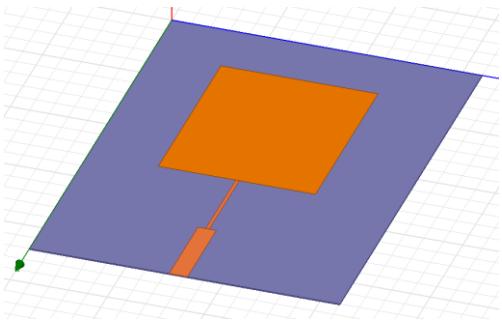


但是我的 ZTO 是要设计 120 欧姆的

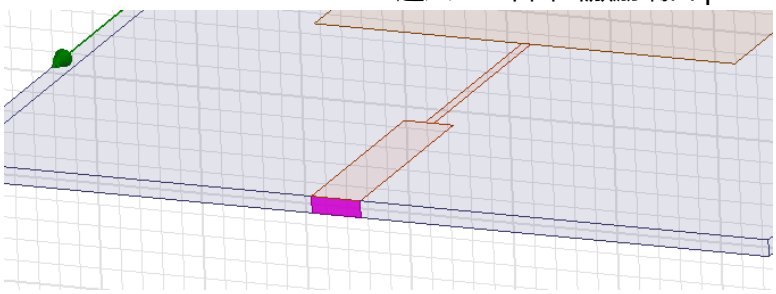




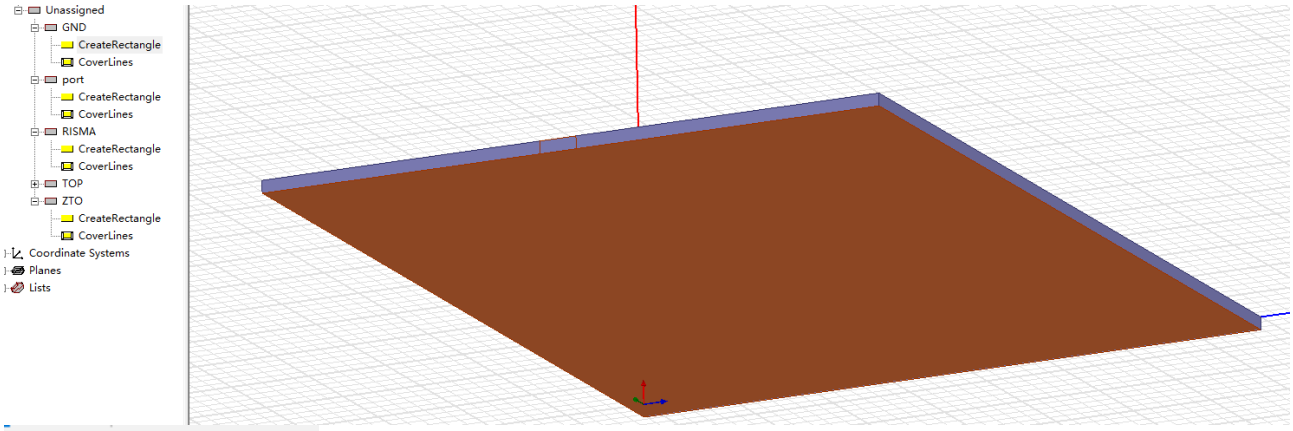




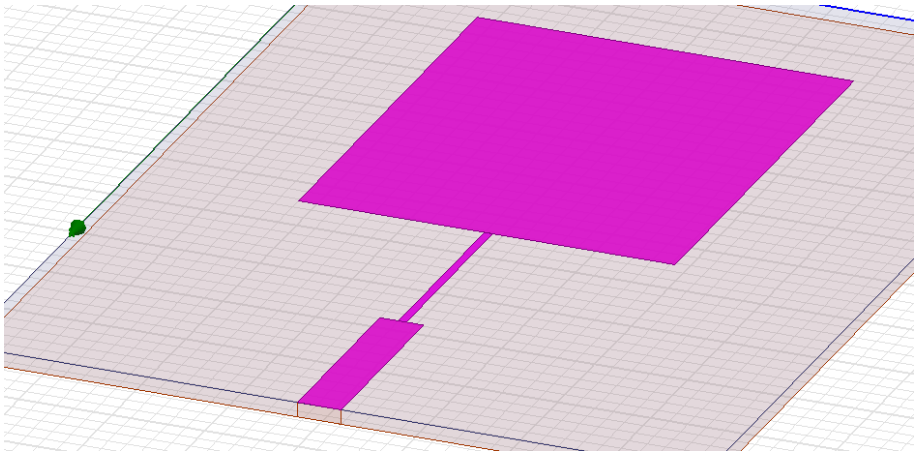
进入 YZ 平面画激励端口 port

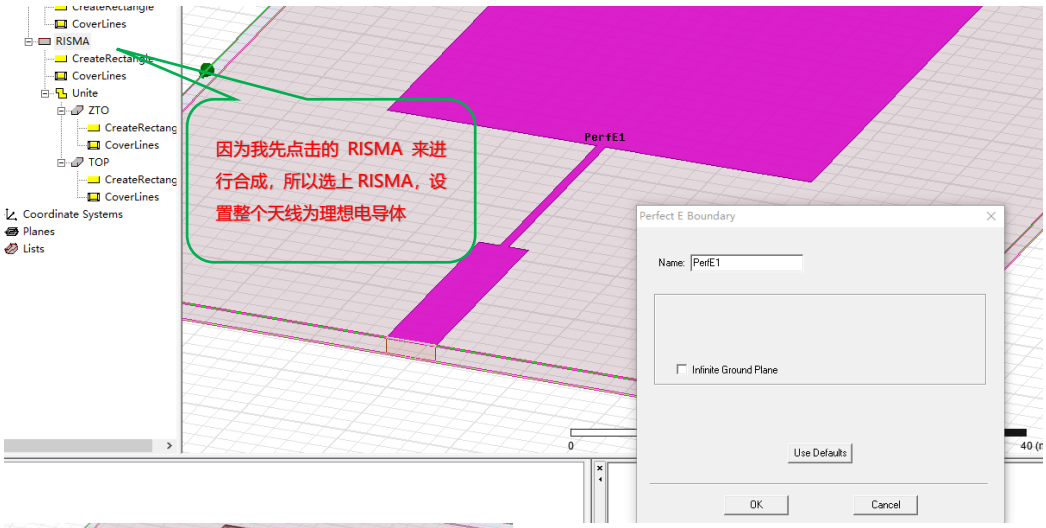


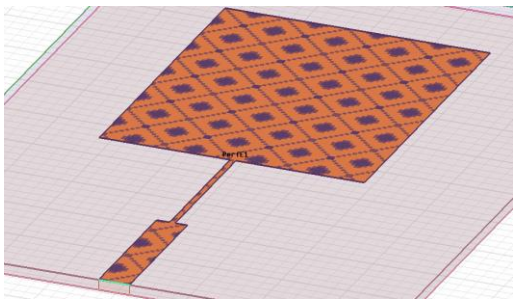
还是要加入底层GND参考平面,记住设置成理想电导体,如果是在激励端口之后设置GND为理想电导体，有可能会报错，这时候只有删除激励端口，从新画/设置 GND 为理想电导体

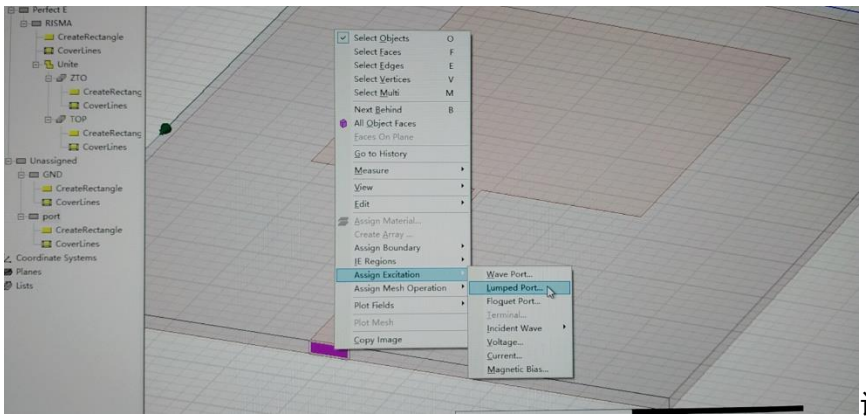


用合成工具，将天线铜，阻抗变化铜 ZTO 和 50 欧姆输入铜合并成一个导体



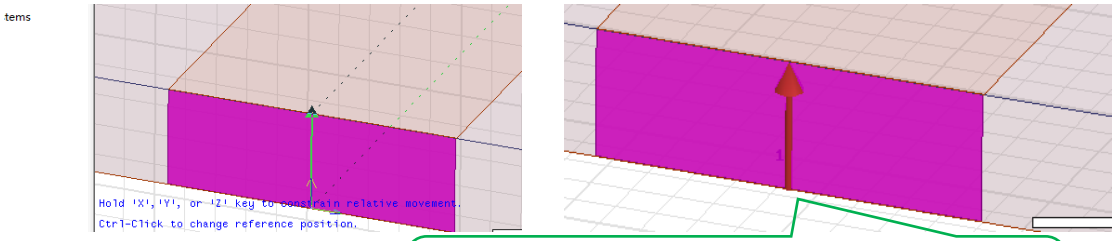






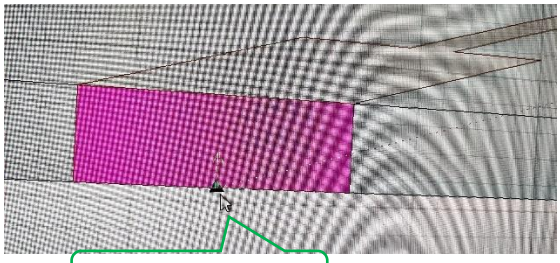
设置 port 铜皮为激励端口

激励端口的箭头到底该怎么画?



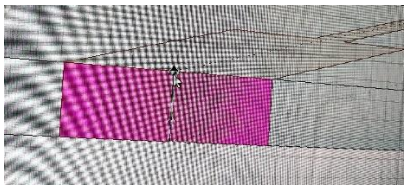
其实 Lumped port 箭头不管是从下向上画还是从上向下画都没有关系，只要保证箭头一端接触的是 GND(天线负极)参考面，另一端接触的是天线正极即可

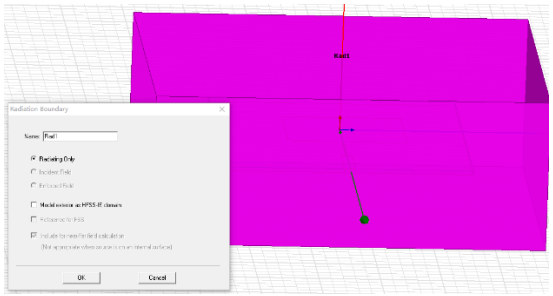
激励端口注意事项

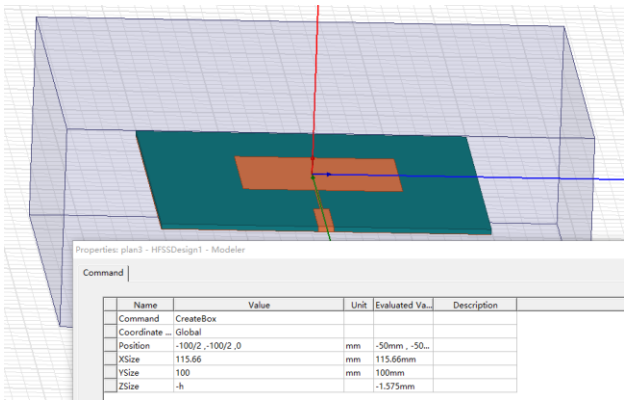


一定是三角形开始画

如果是原点开始，就表示鼠标没有瞄对，一定要三角形

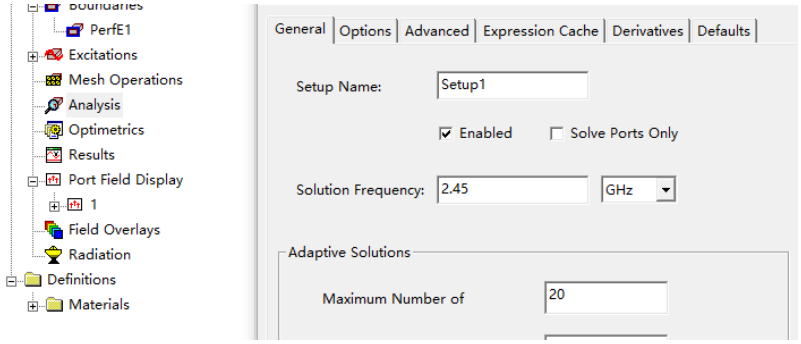




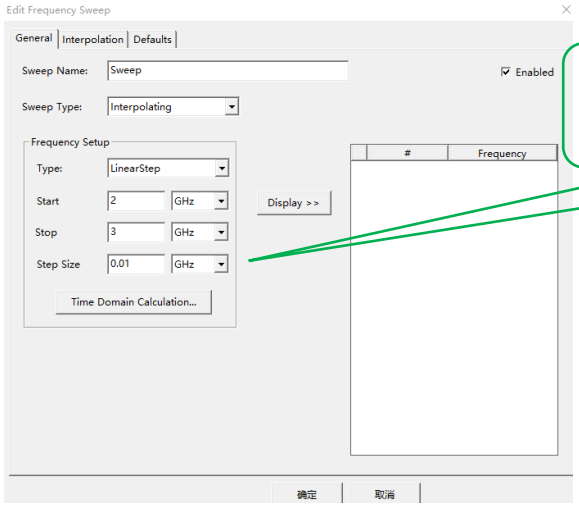


加入空气盒子

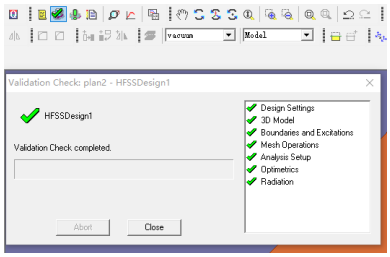
设置仿真频率

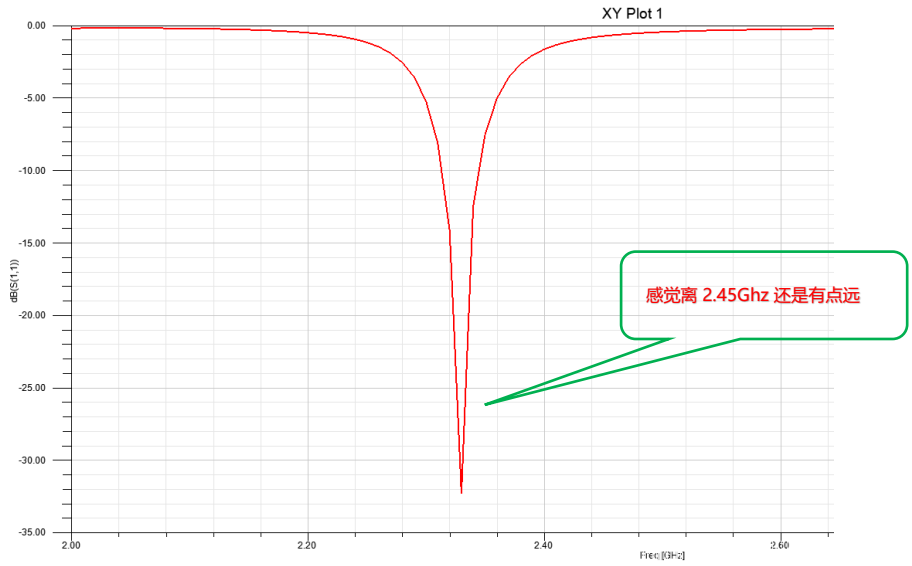


选择 Fast 设置扫频范围

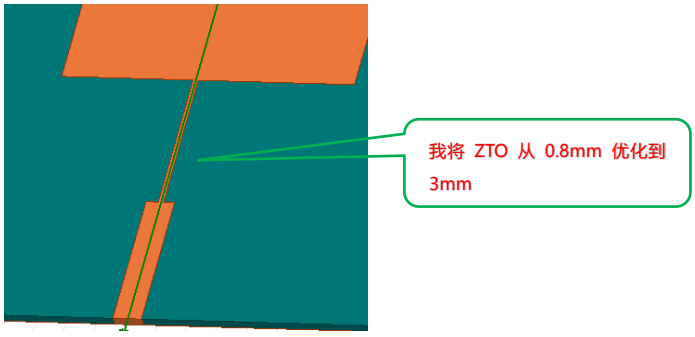


扫频范围默认 0.1Ghz，就是 100Mhz 递增扫描，我感觉扫频不够精细，所以改成 0.01Ghz(10Mhz)扫频，这样出来的曲线多一些。

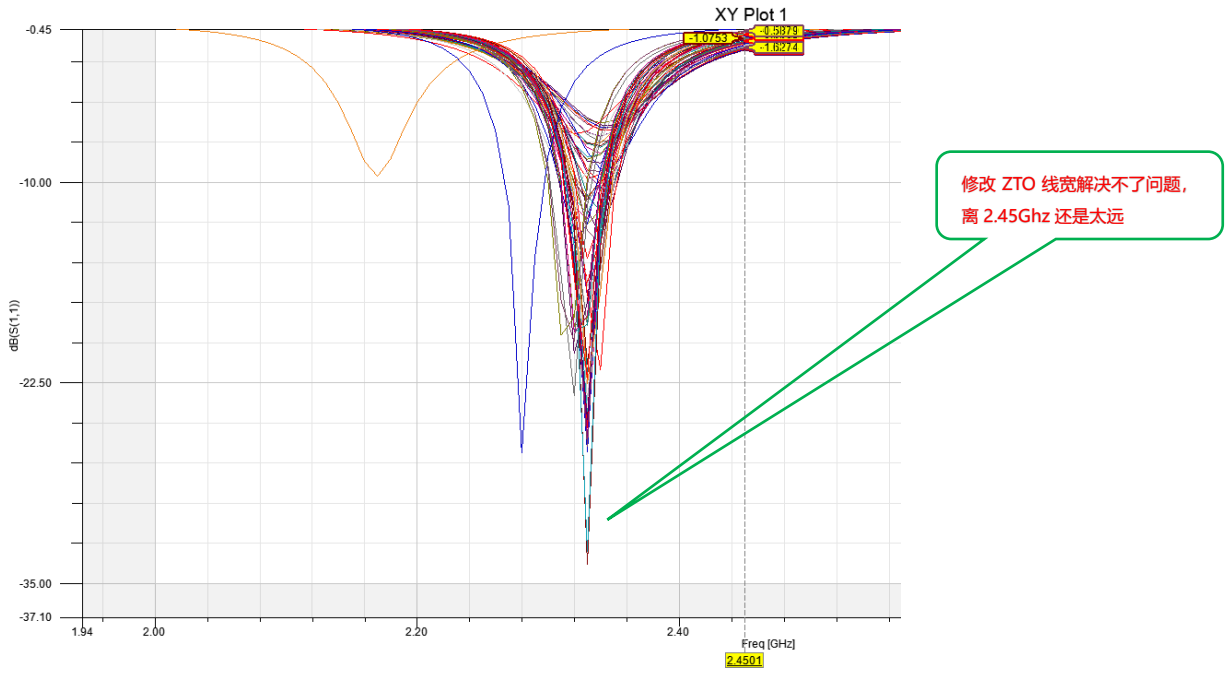


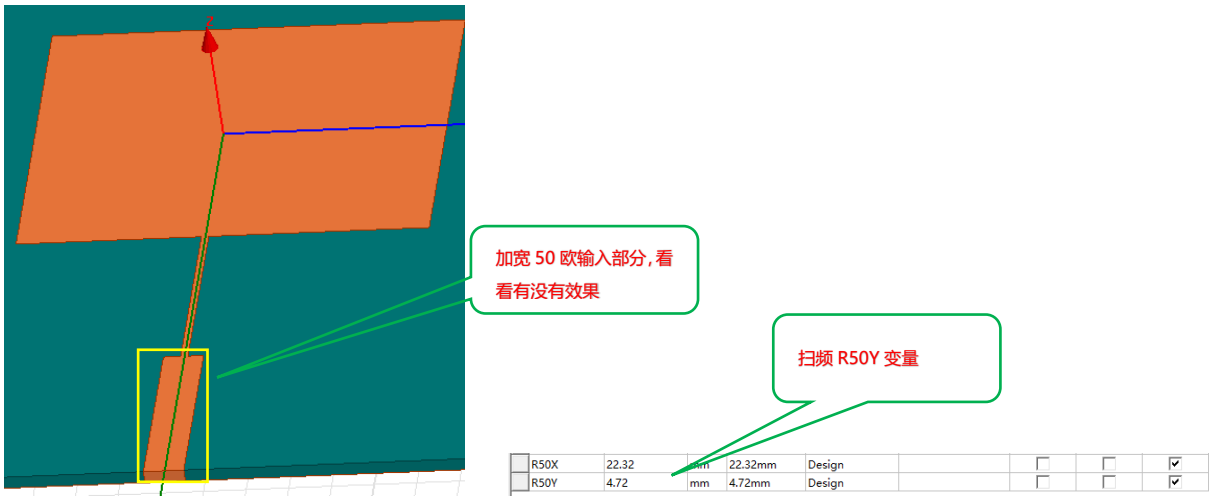


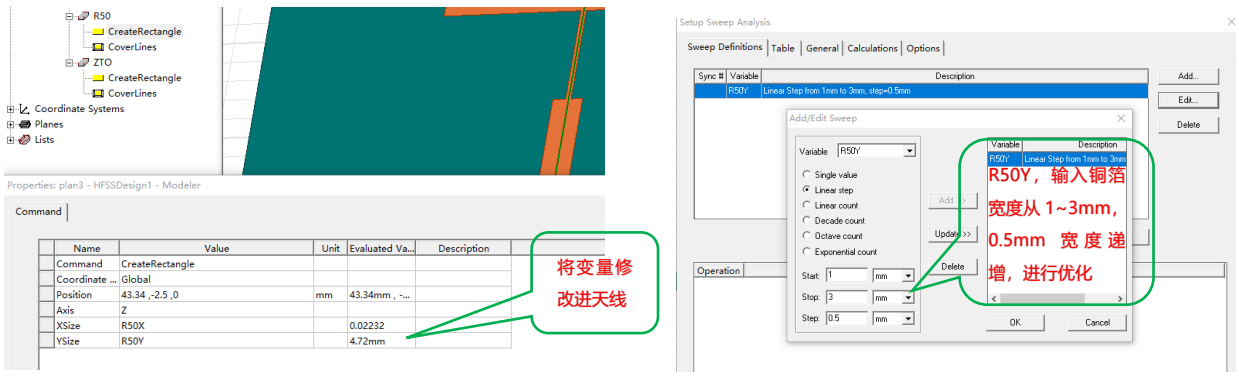
感觉离 2.45Ghz 还是有点远

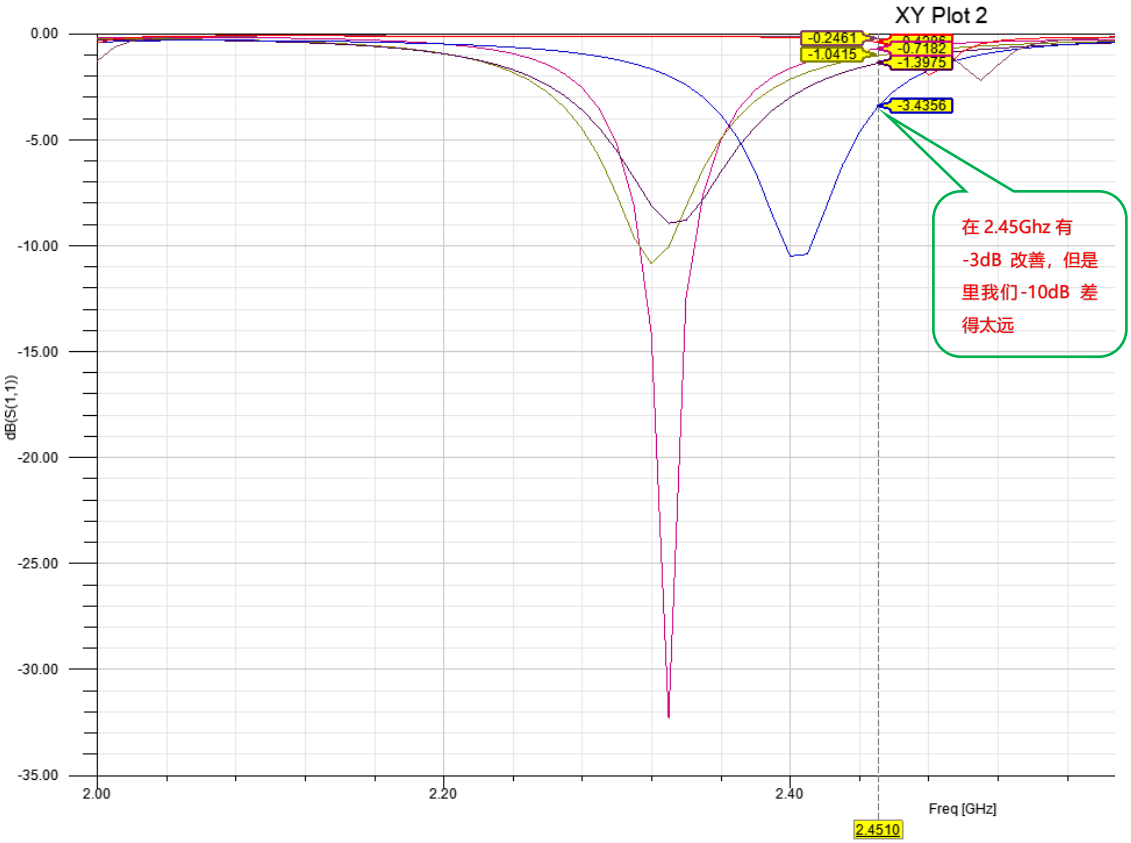


我将 ZTO 从 0.8mm 优化到 3mm

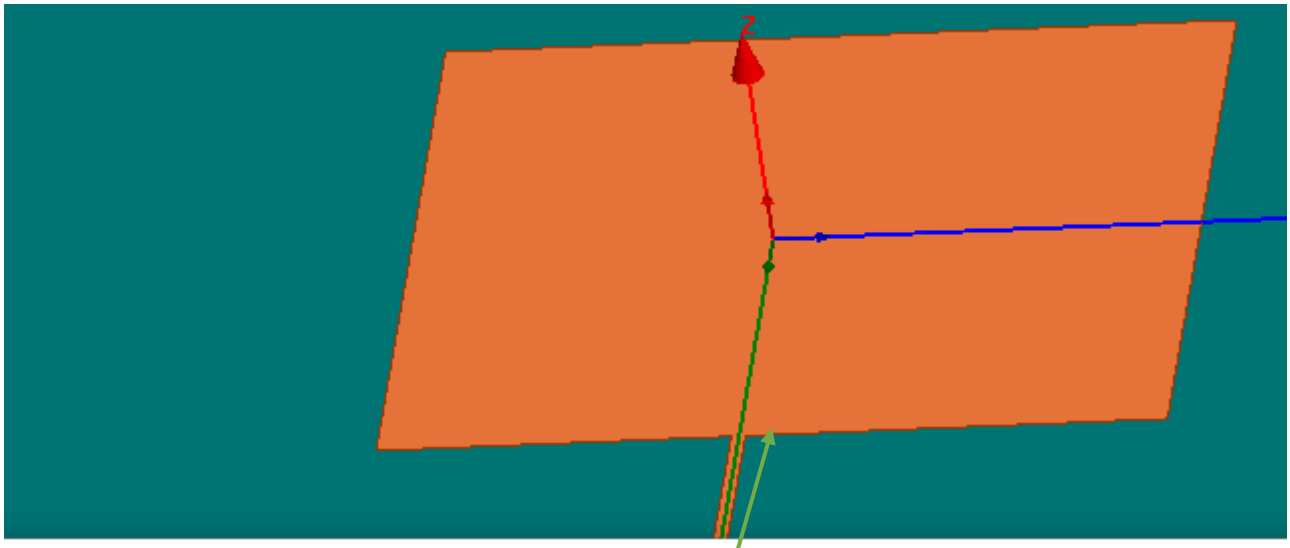

XY Plot 1
-0.45
-10.00
-22.50
-35.00
-37.10
dB(S(1,1))
1.94
2.00
2.20
2.40
Freq [GHz]
2.4501
修改 ZTO 线宽解决不了问题，
离 2.45Ghz 还是太远


加宽 50 欧输入部分，看
看有没有效果
扫频 R50Y 变量
R50X 22.32 mm 22.32mm Design
R50Y 4.72 mm 4.72mm Design


R50
CreateRectangle
CoverLines
ZTO
CreateRectangle
CoverLines
Coordinate Systems
Planes
Lists
Command
Name Value Unit Evaluated Va... Description
Command CreateRectangle
Coordinate ... Global
Position 43.34 ,-2.5 ,0 mm 43.34mm , -...
Axis Z
XSize R50X 0.02232
YSize R50Y 4.72mm
将变量修
改进天线
Setup Sweep Analysis
Sweep Definitions Table General Calculations Options
Add/Edit Sweep
R50Y，输入铜箔
宽度从 1~3mm，
0.5mm 宽度递
增，进行优化



看来修改阻抗变换线宽，和输入线宽没有明显改善，下面我们试试改善中心铜箔

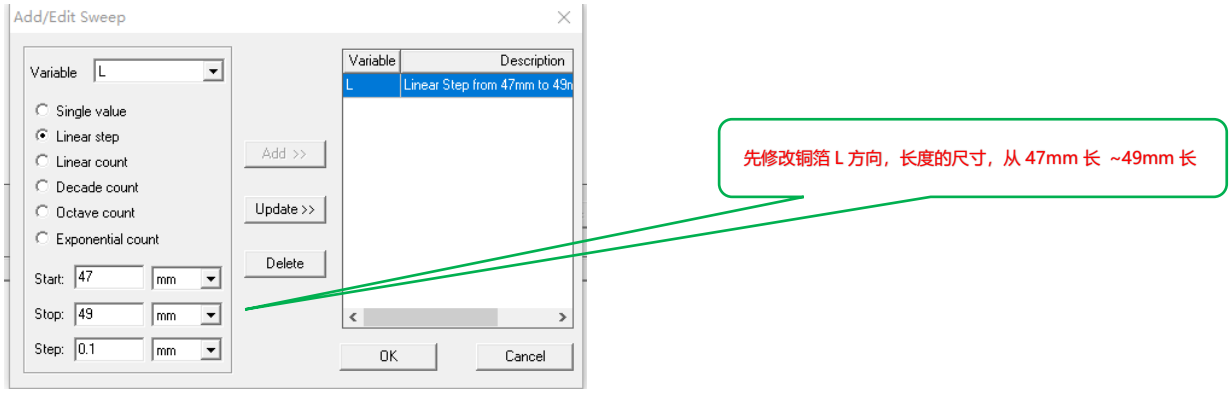


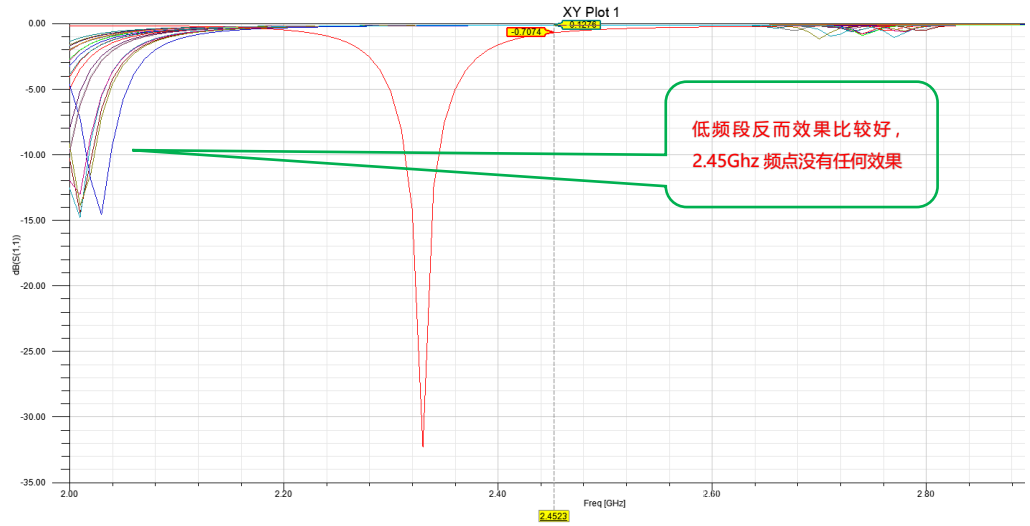
rties: plan3 - HFSSDesign1 - Modeler

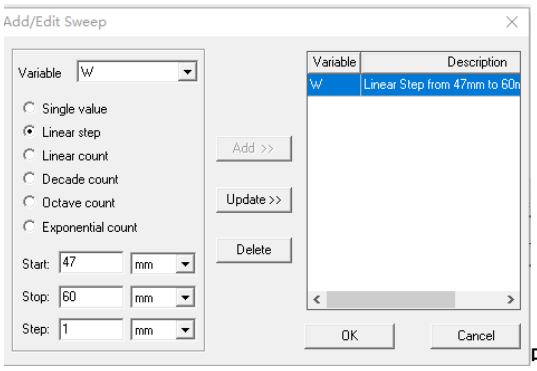
mand

	Name	Value	Unit	Evaluated Va...	Description
	Command	CreateRectangle			
	Coordinate ...	Global			
	Position	-20.5 ,-22.2 ,0	mm	-20.5mm , -2...	
	Axis	Z			
	XSize	L		40.5mm	
	YSize	W		48.4mm	

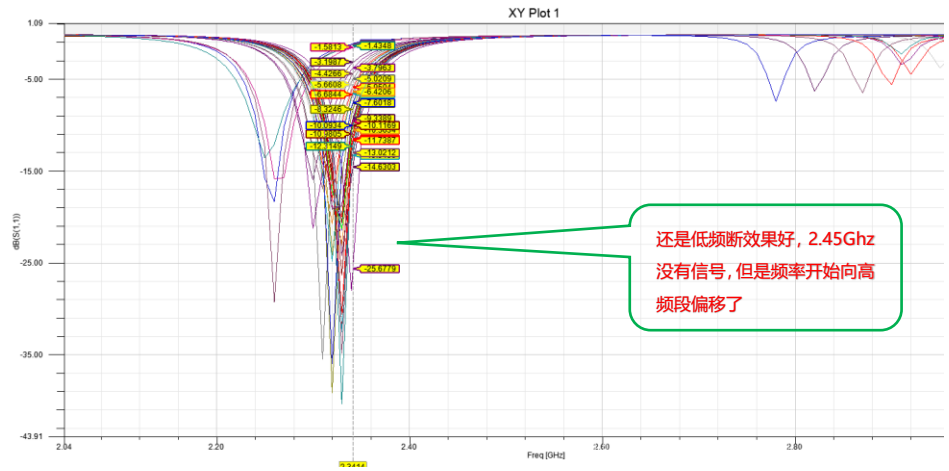
中心铜箔当前尺寸







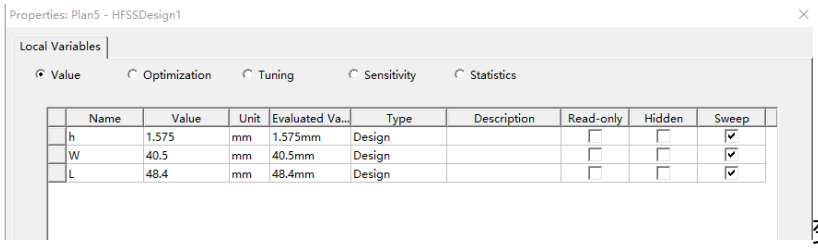
中心铜箔 Y 方向宽度 W 改成 47~60mm 宽度仿真



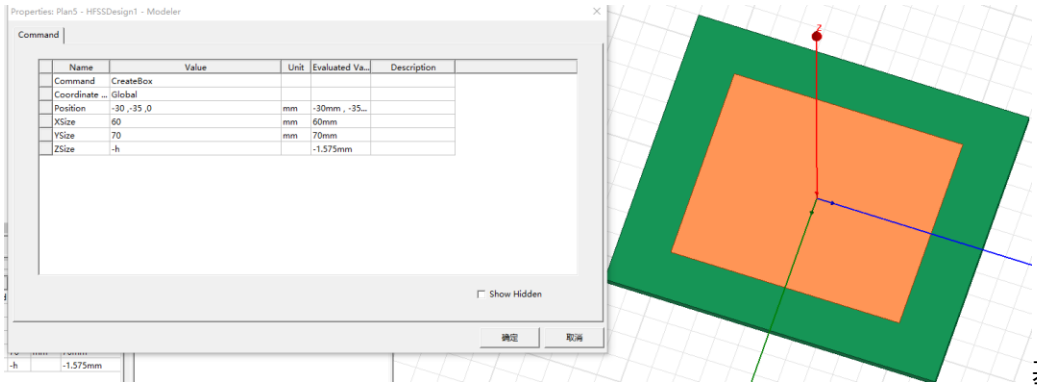
经过以上测试，自己多多修改中心铜箔的宽度和高度，还有介质厚度之类的。看能否到达 2.45Ghz 标准。我这里就不再多多实验了。

圆极化贴片天线

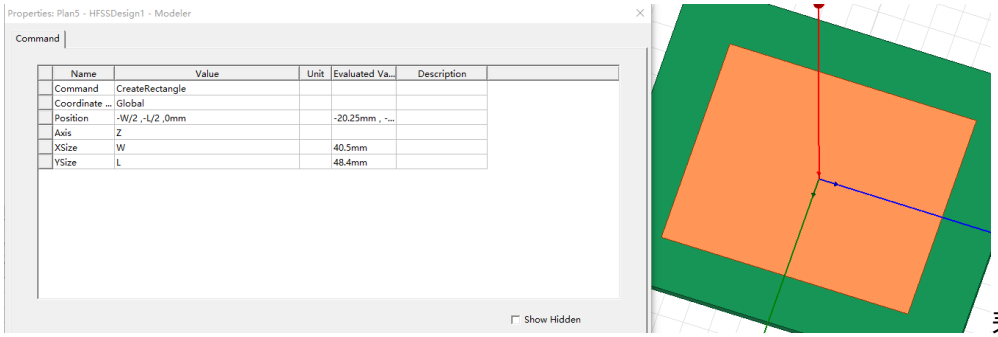
拷贝我之前的馈点天线工程



变量定义

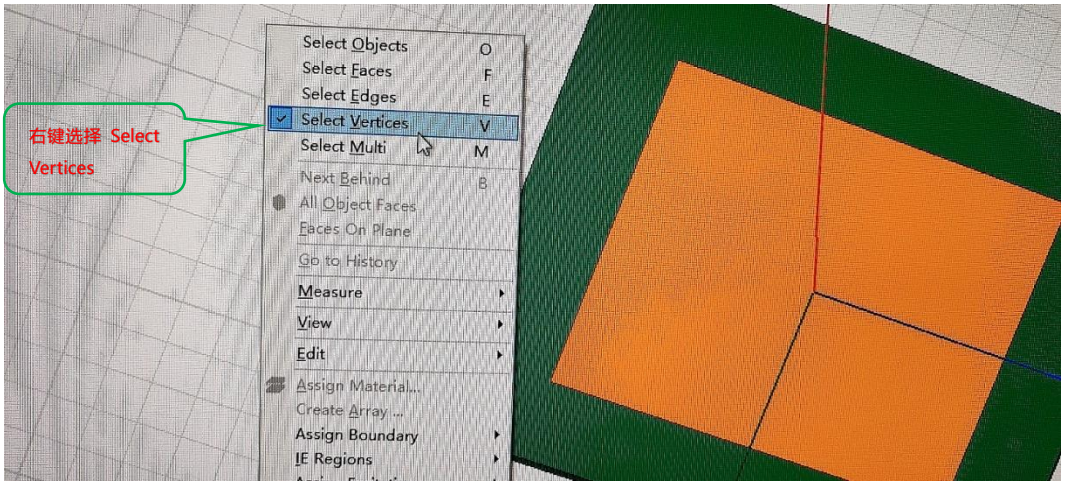


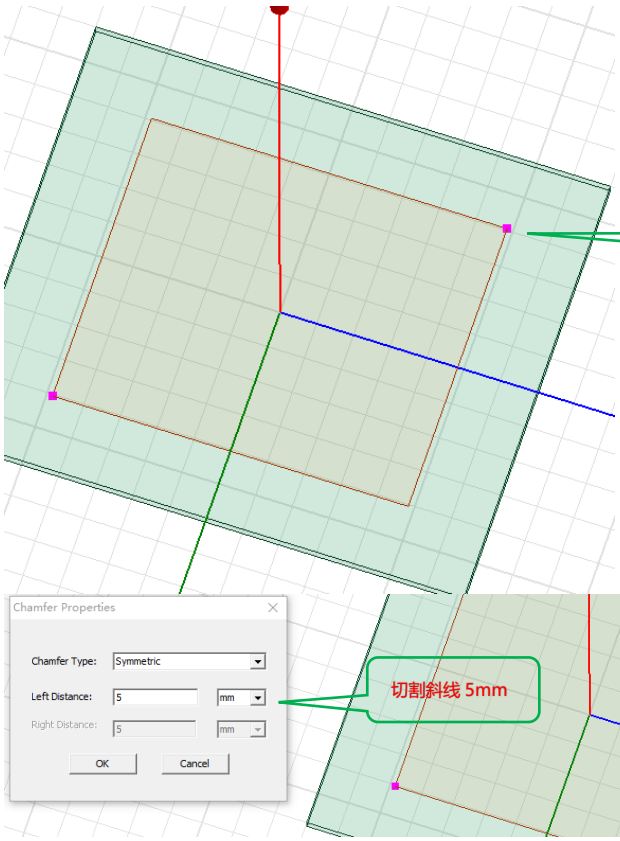
基材尺寸

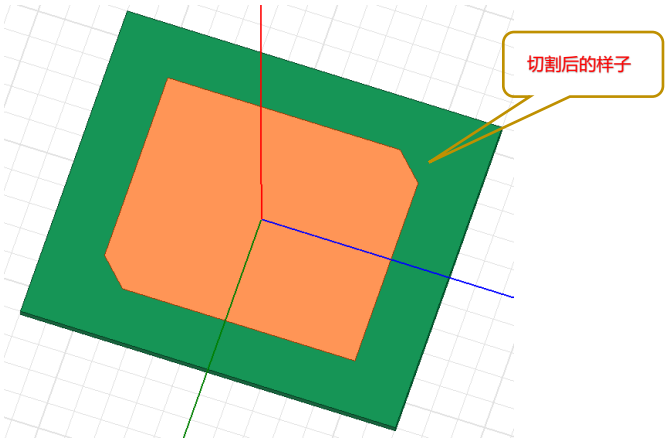


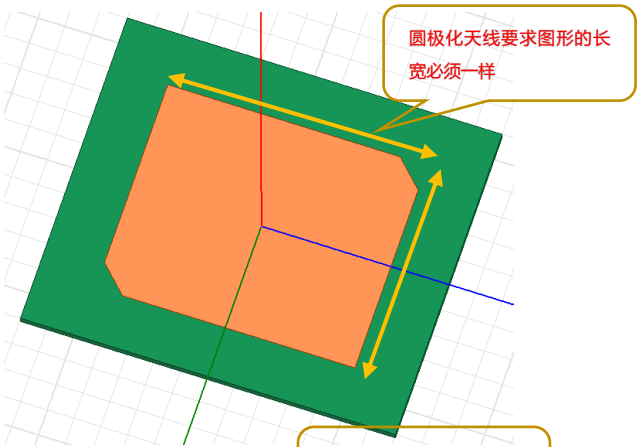
表面铜箔尺寸

开始切割圆弧







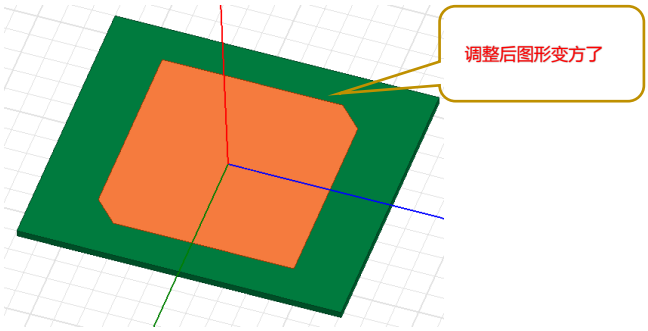


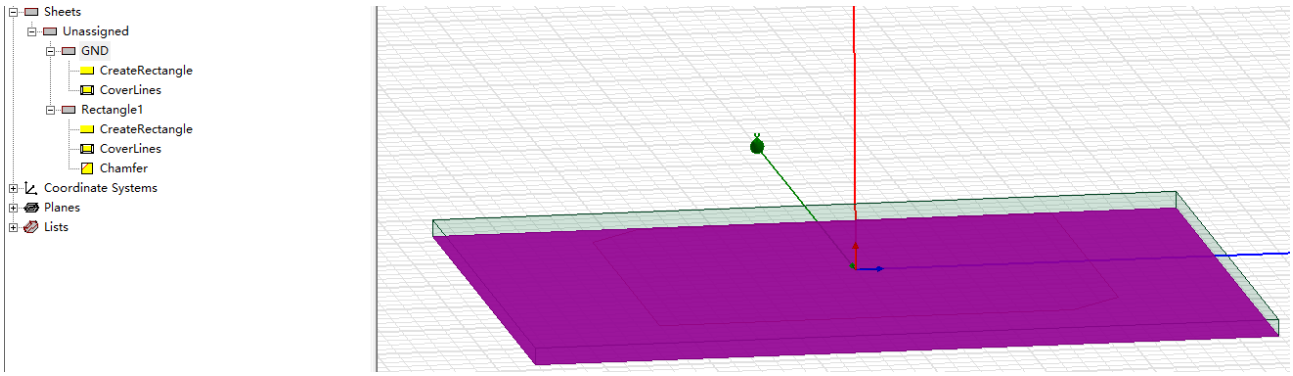
Name	Value	Unit	Evaluated Va...	Type
h	1.575	mm	1.575mm	Design
W	40.5	mm	40.5mm	Design
L	48.4	mm	48.4mm	Design

这是以前的长宽

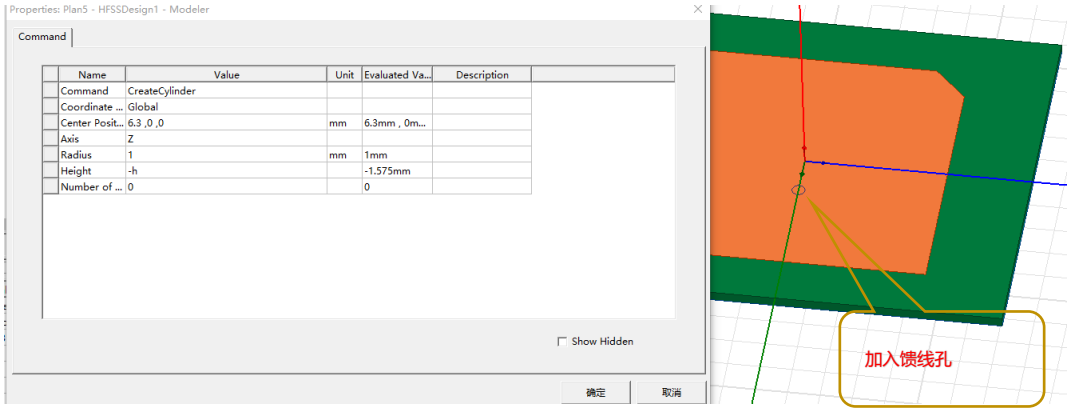
为了方便，我就将长宽取一个中间值

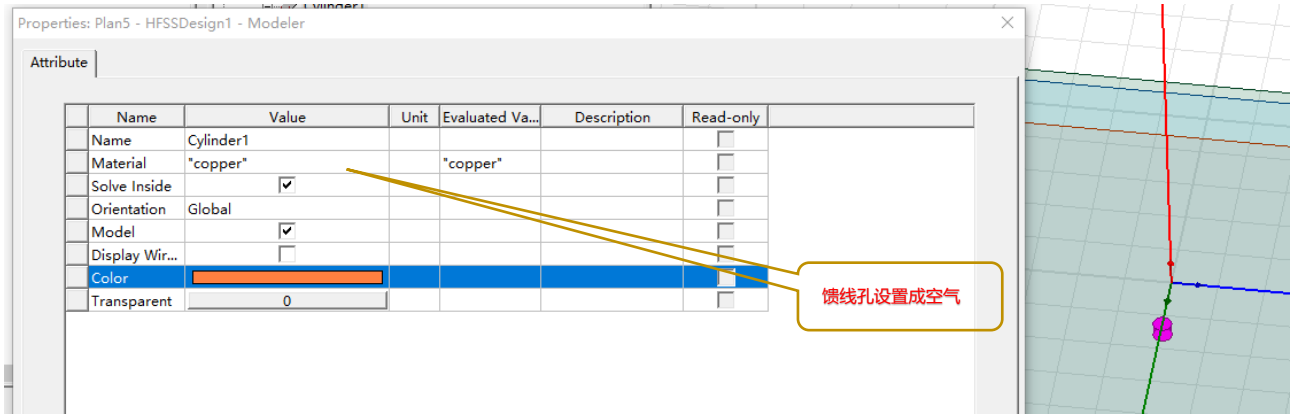
Name	Value	Unit	Evaluated Va...	Type
h	1.575	mm	1.575mm	Design
W	44	mm	44mm	Design
L	44	mm	44mm	Design



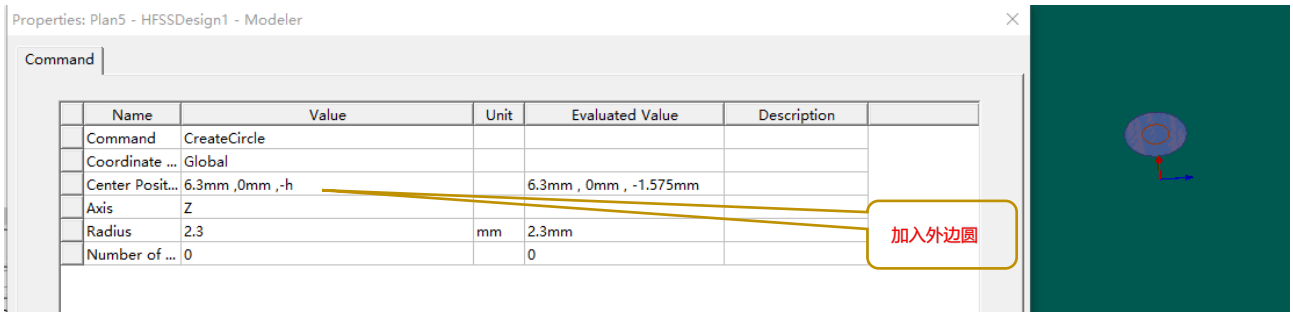


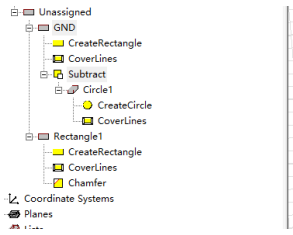
在 PCB 基材背面铺设 GND

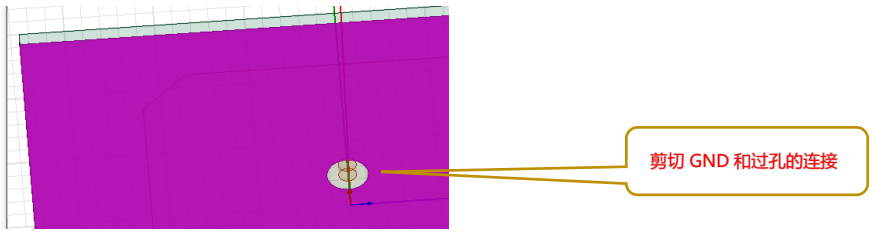




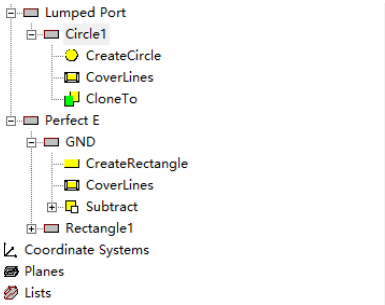
设置表层铜皮和基材背面的 GND 铜皮为理想电导体，记住一定是先设置了理想电导体，再按照后面步骤切割外圆。

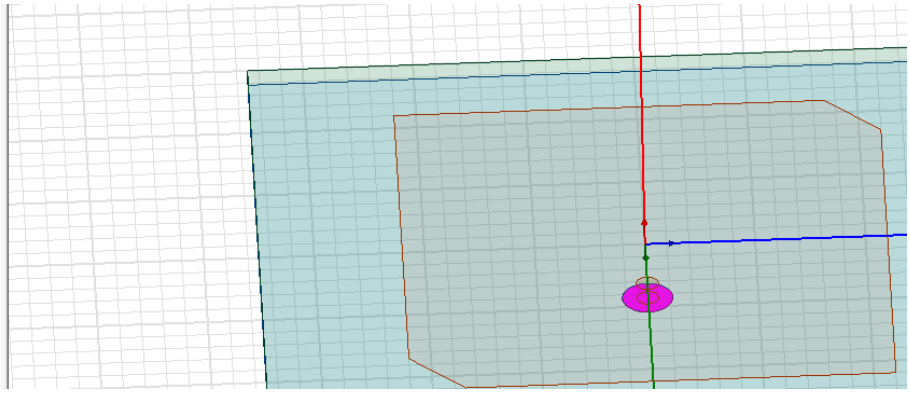




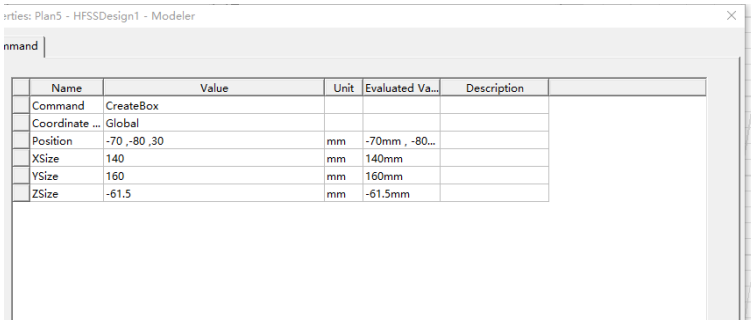


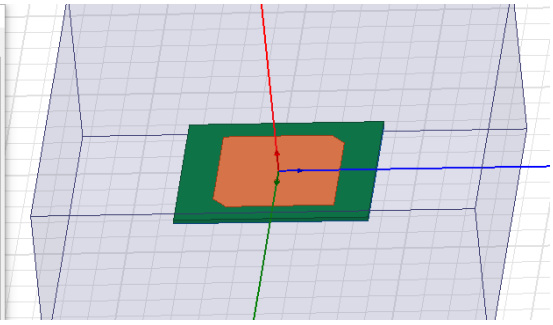
过孔因为加入了 copper 属性所以不需要设置电导体之类。





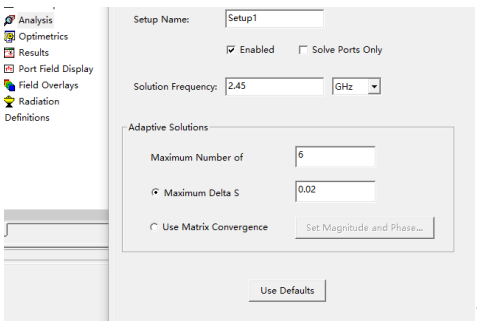
点击外圆设置激励端口。



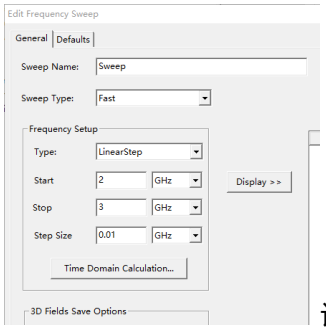


加入空气盒子。空气盒边界条件设置为 Radiation.....

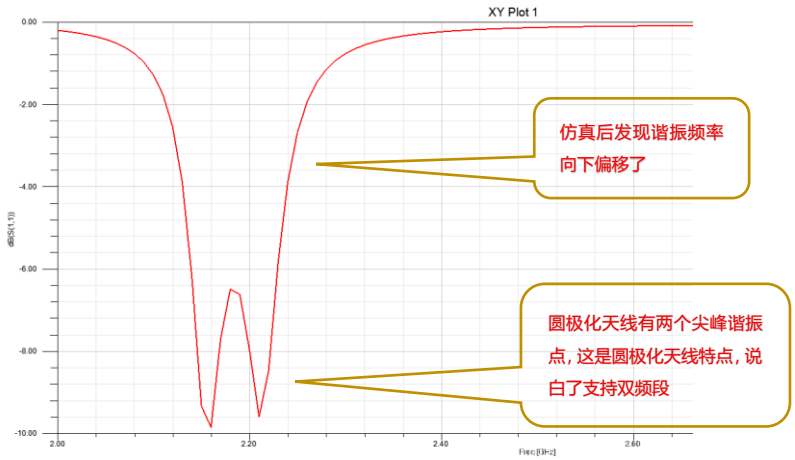
下面开始圆极化天线仿真



设置中心频率

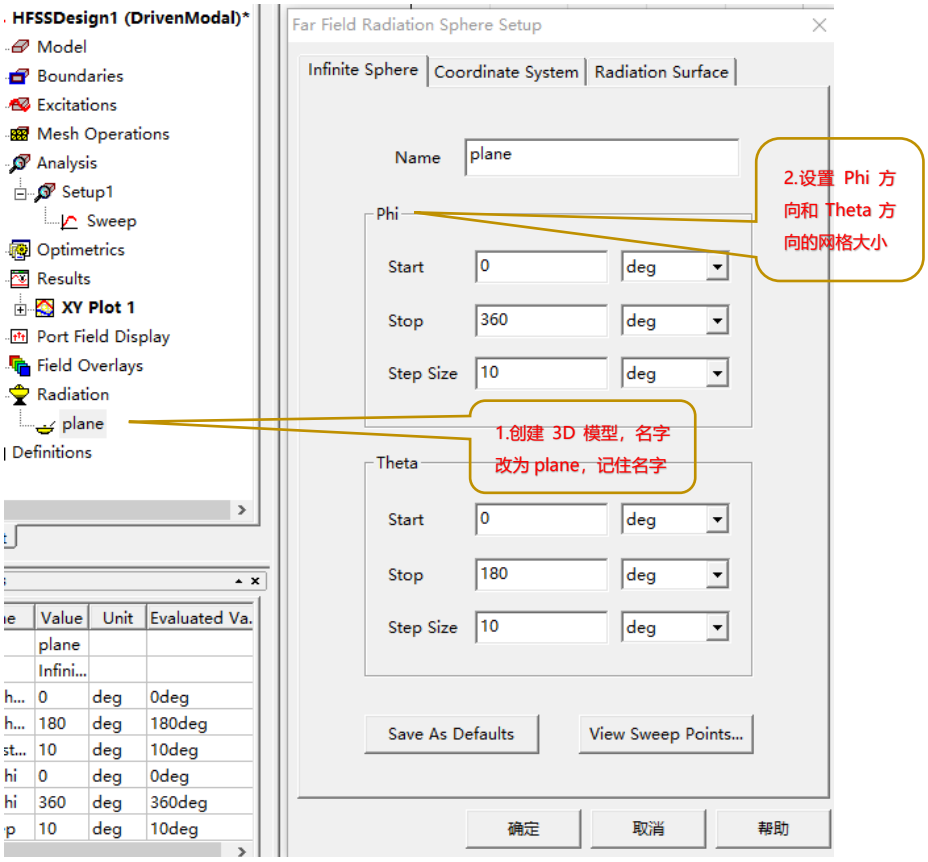


设置扫频



天线轴比数据生成

圆极化天线才看轴比，偶极子天线是不用看轴比的，表面贴片天线也不用看轴比。

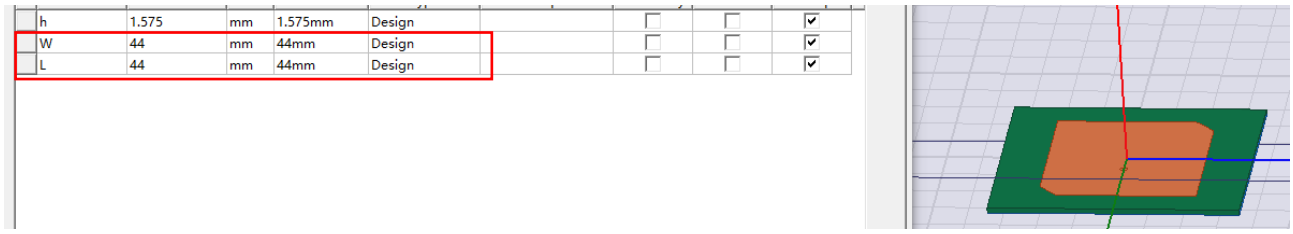


一个比较好理解的解释，Phi 是方位角，Theta 是俯仰角，横向是方向，纵向是俯仰.....

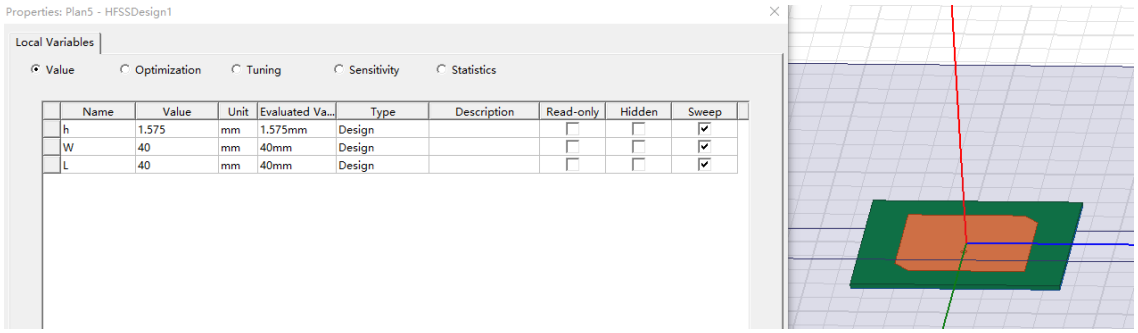
有了方向和俯仰角的设置，后面优化天线，我们就可以直接看轴比

下面我们要优化天线，前面有个章节说了，天线越长谐振频率越低，天线越短谐振频率越高。

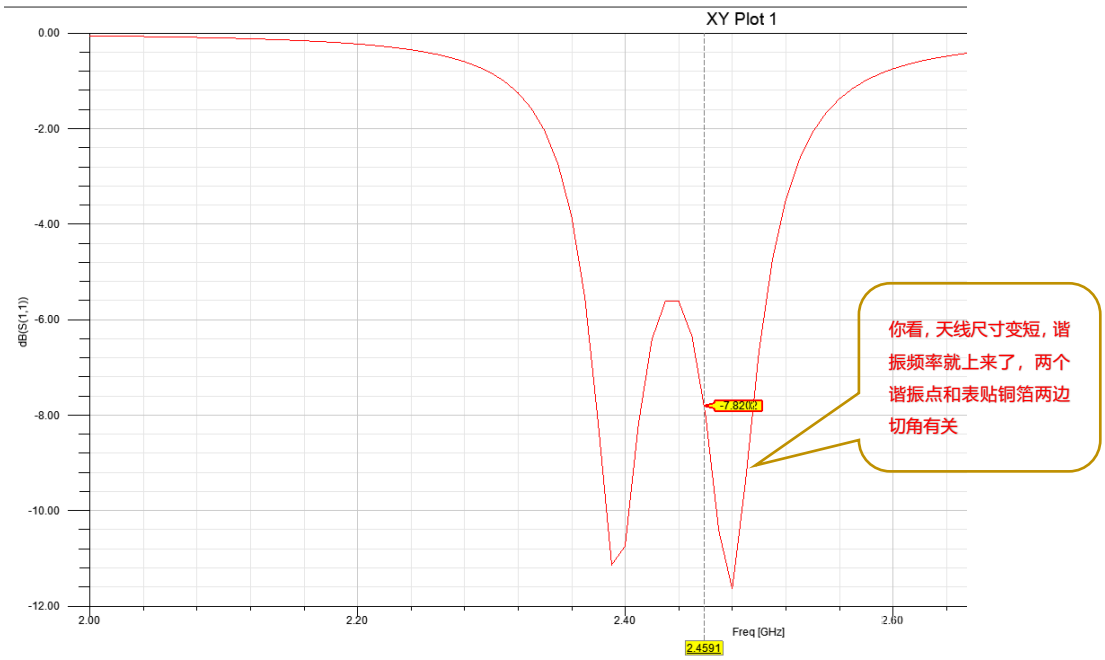
现在我们频点是低于 2.45Ghz 很多，我们要降低表贴铜箔的长宽尺寸

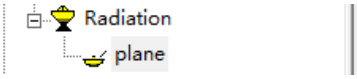


将长宽 44mm，改为 40mm

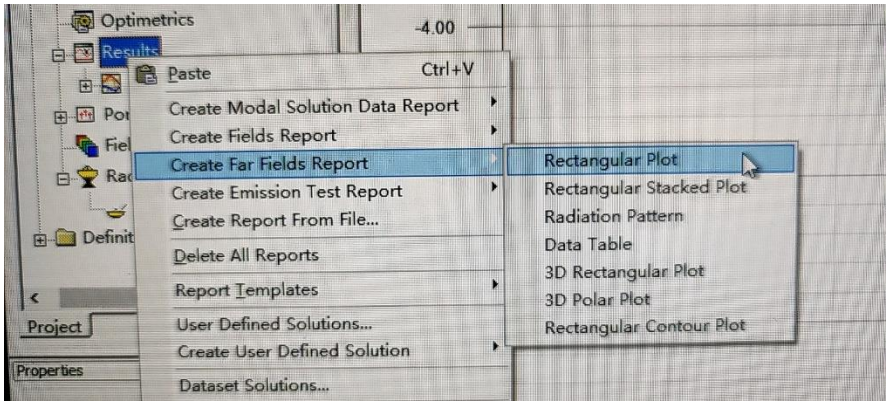


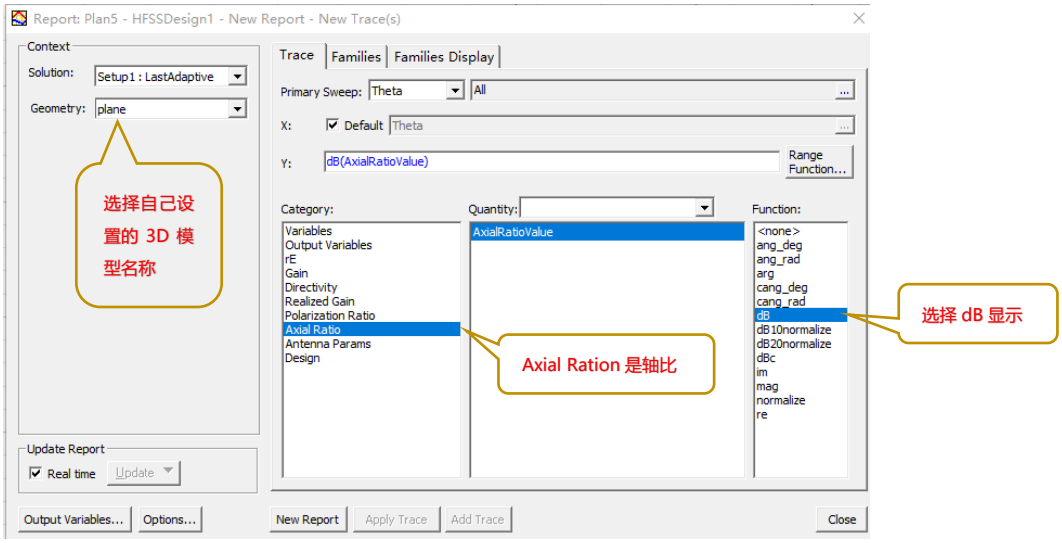
开始仿真

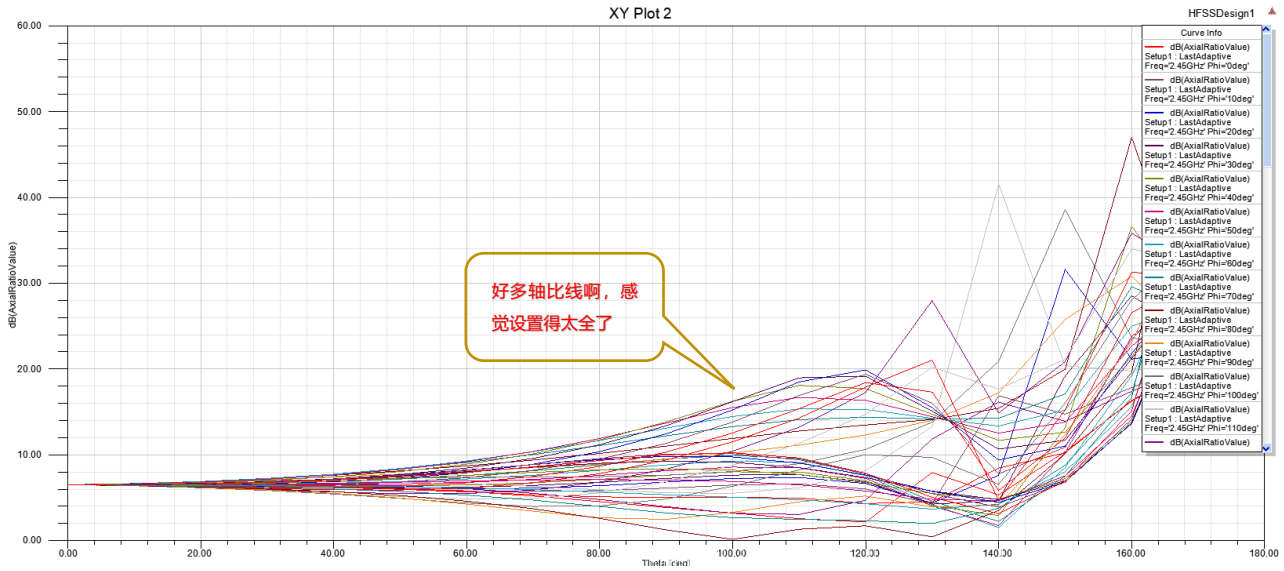


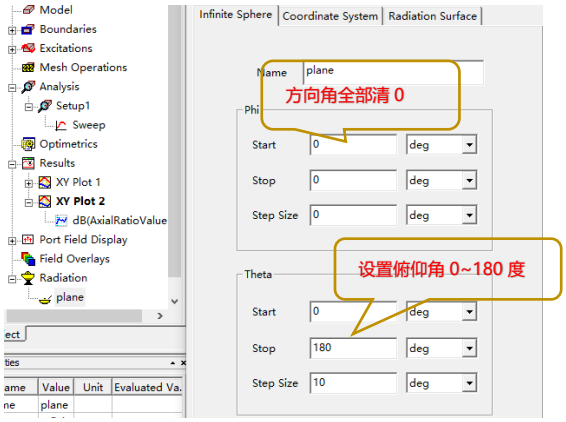


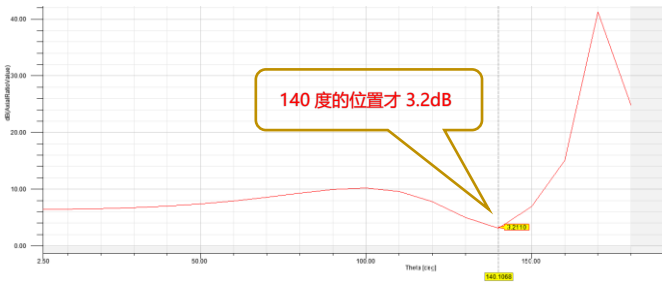
因为前面设置好了 3D 模型参数，Results 会多出 Create Far Fileds..参数











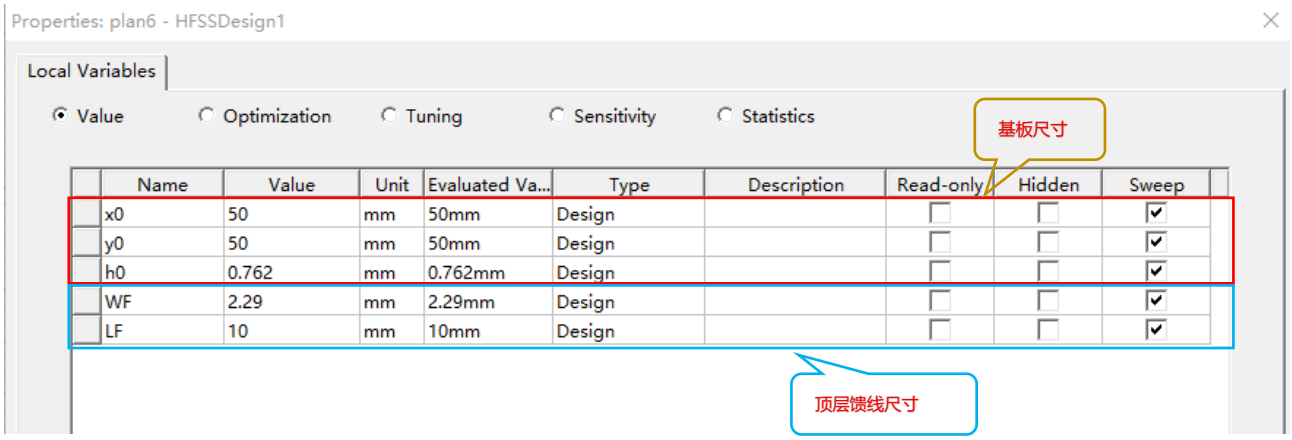
轴比要 < 3dB，才是最好的。

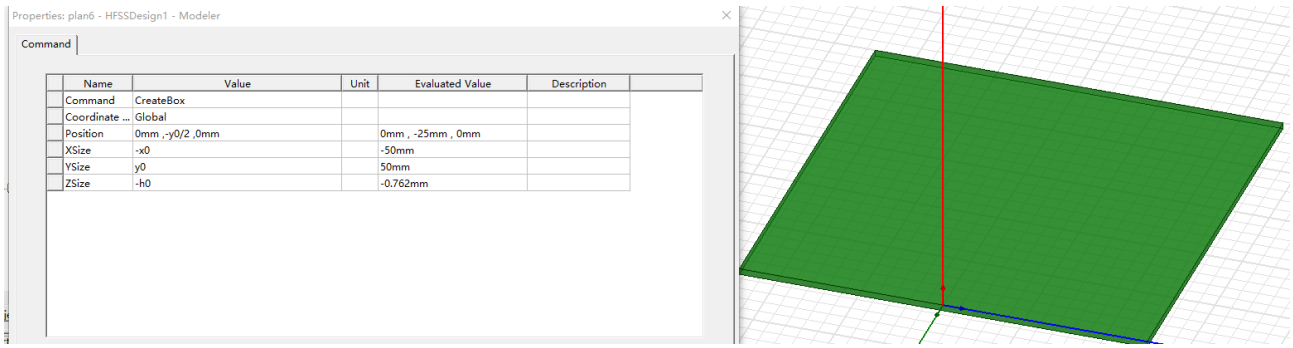
八木天线仿真

八木天线的基础天线是偶极子天线

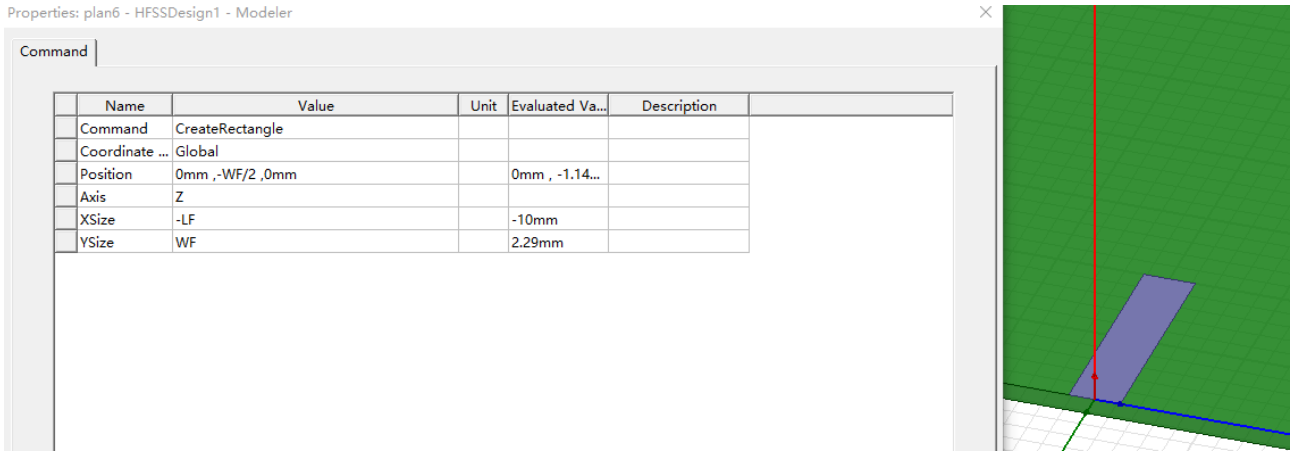
基础偶极子天线仿真

3Ghz 频段进行仿真





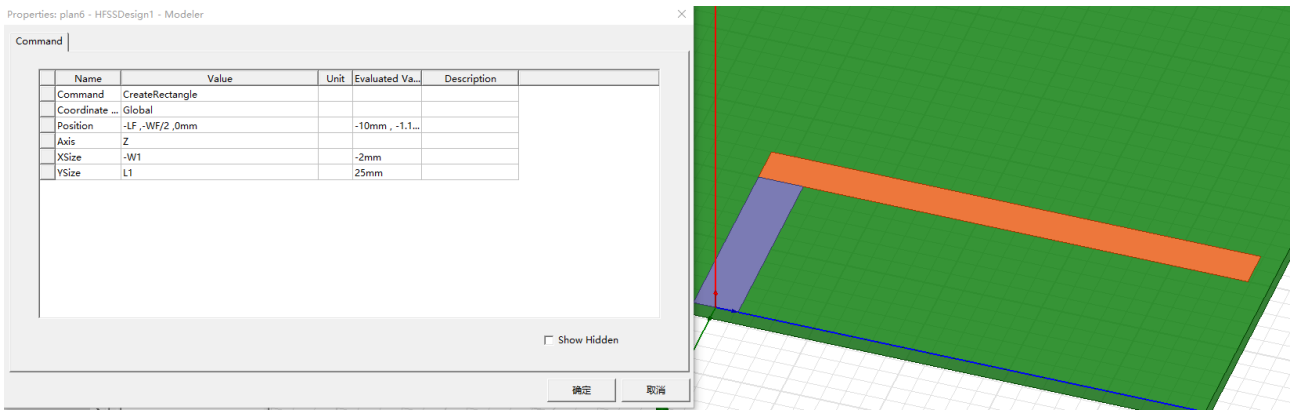
基材 PCB 板定义



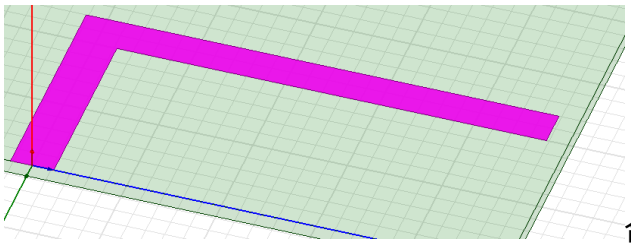
顶层馈线定义

	W1	2	mm	2mm	Design		☐	☐	☑
	L1	25	mm	25mm	Design		☐	☐	☑

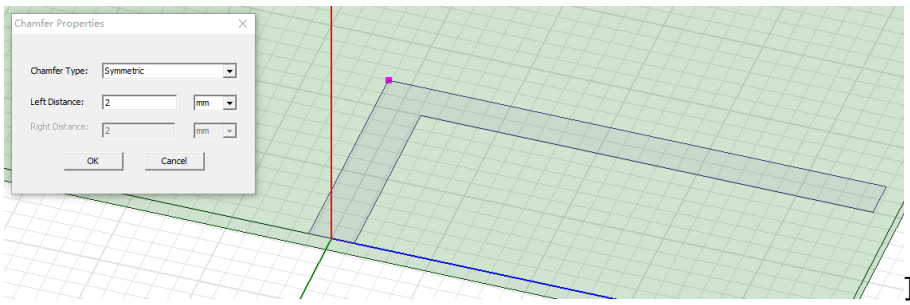
顶层天线尺寸



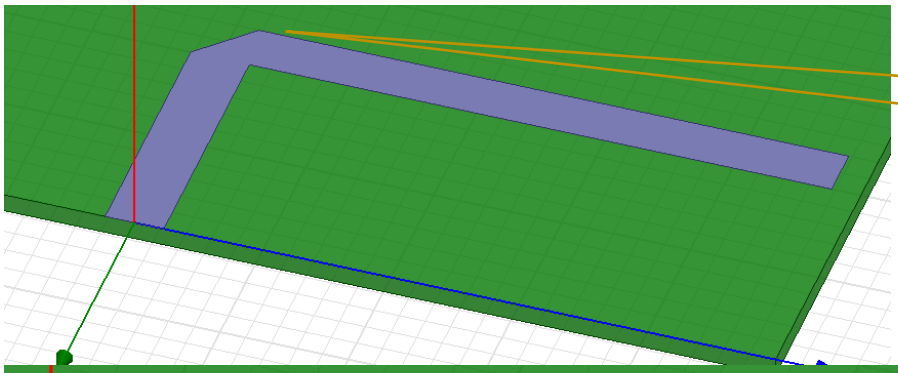
顶层天线尺寸



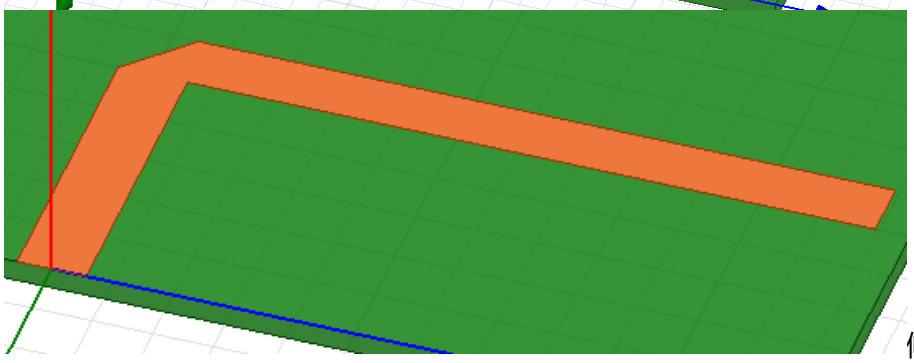
合并顶层天线



顶层天线切角 2mm

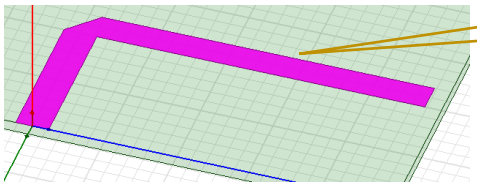


这个切角长度 = 天线宽度的 1.6 倍



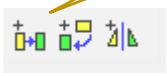
修改下天线颜色

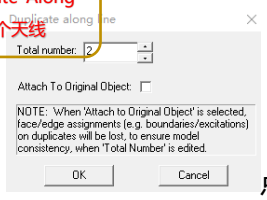
平移画好的天线



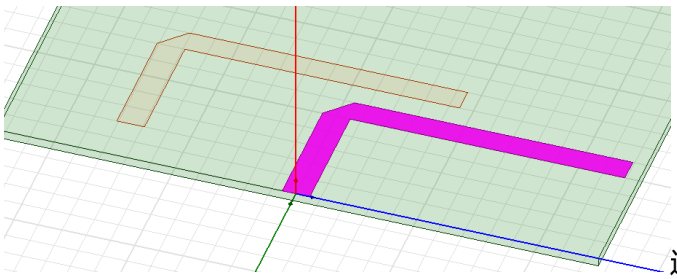
选中天线

选中 Duplicate Along Line 平移整个天线



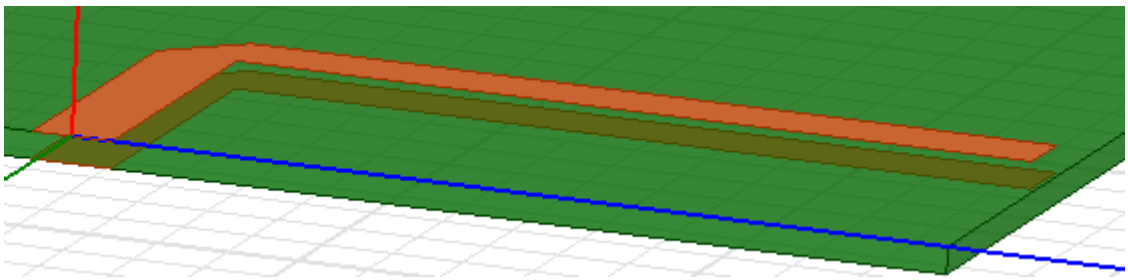


点击 OK

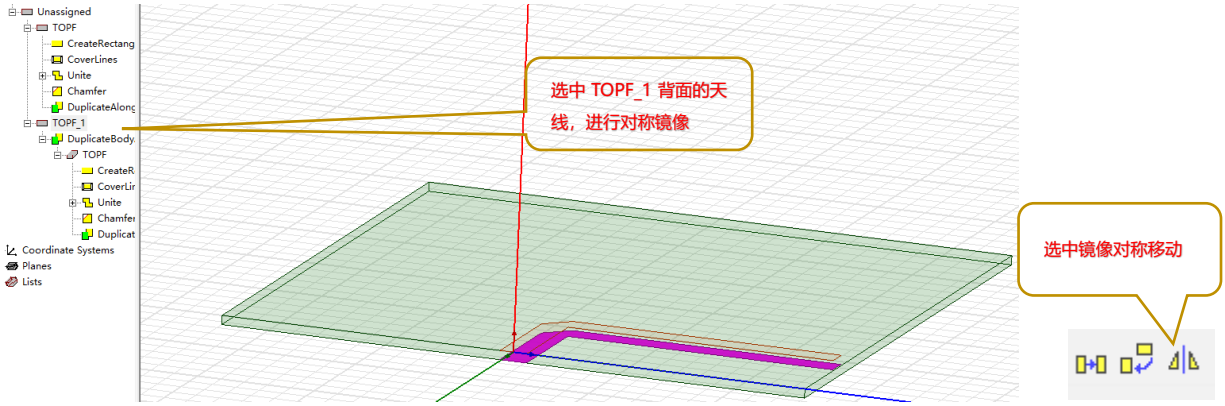


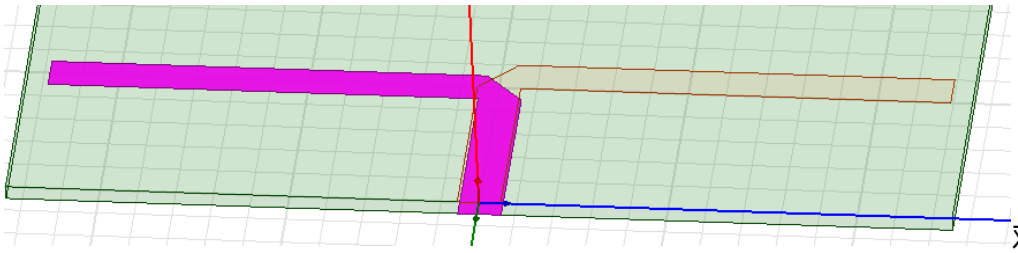
这就是平移天线的效果。这只是演示

我们要把天线平移到背面，对称

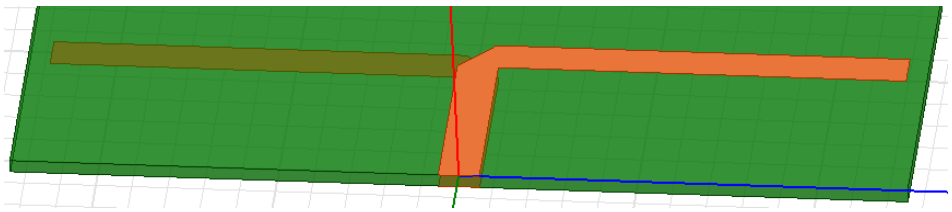


天线被平移到背面

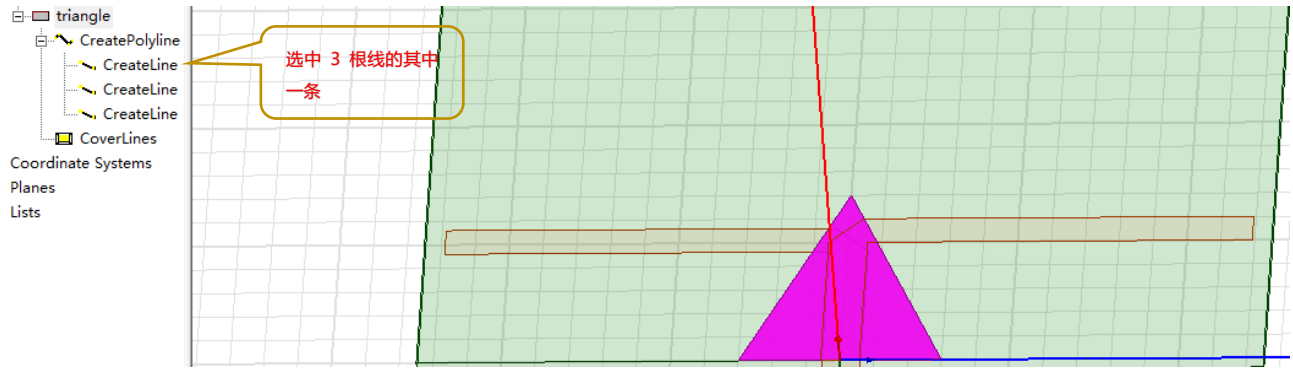




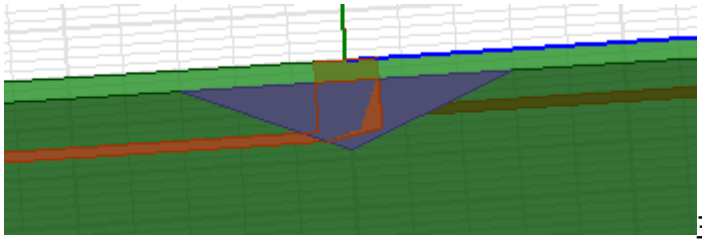
对称成功。



下面加入 GND 地参考铜

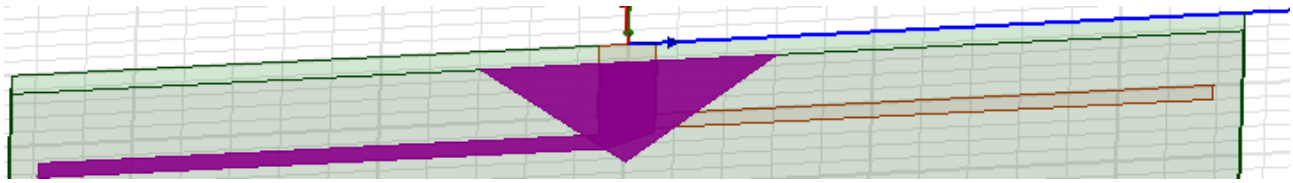


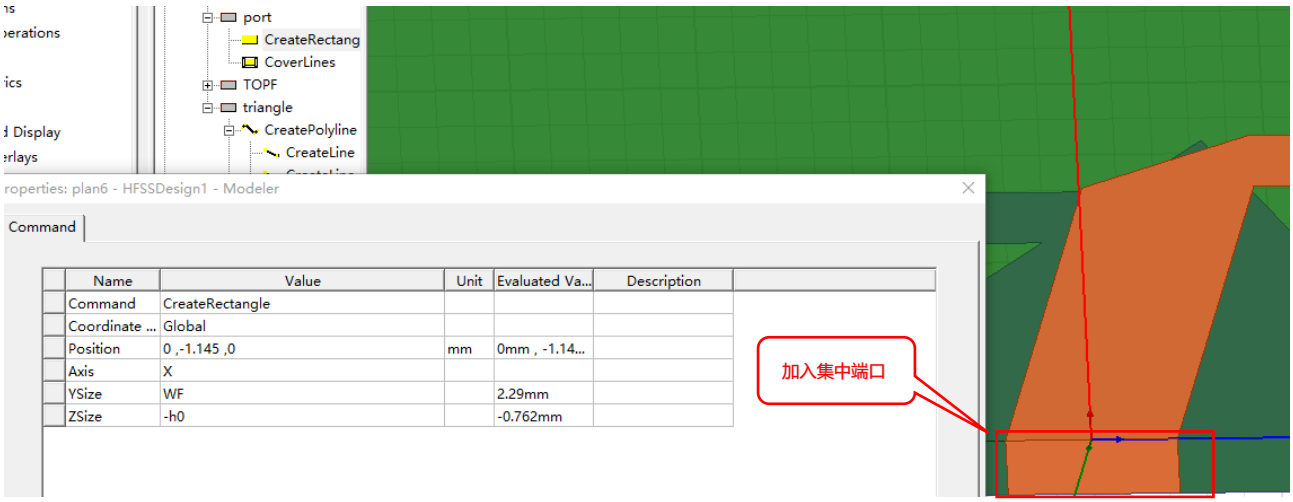
因为三角形 GND 是在 PCB 基板背面

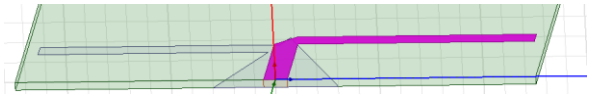


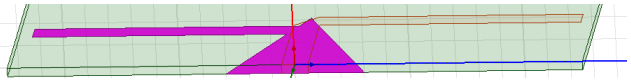
三条线段改完之后，三角形在 PCB 背面了

将地线三角形，与 PCB 背面天线进行合并



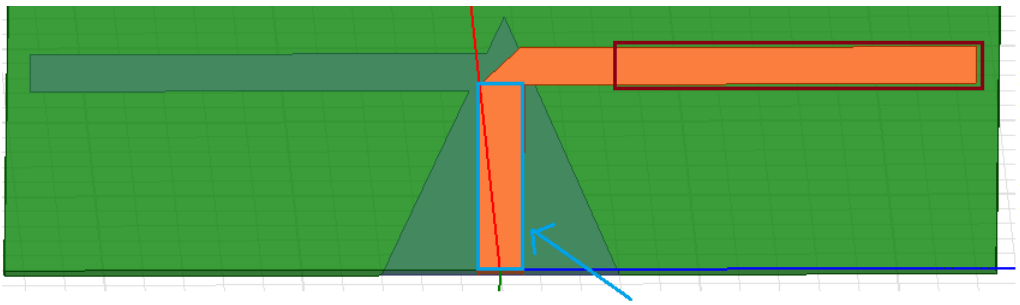




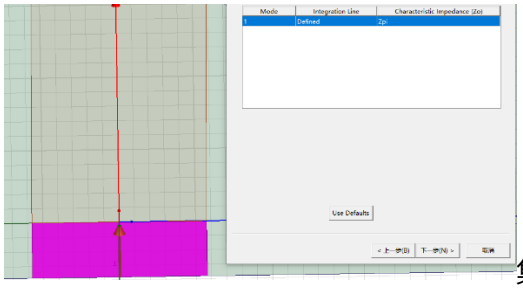


将顶层和底层铜箔设置成理想电导体

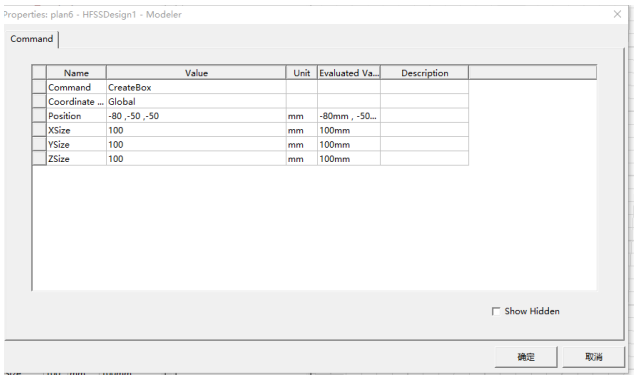
微带线的宽度会不会对阻抗造成影响?

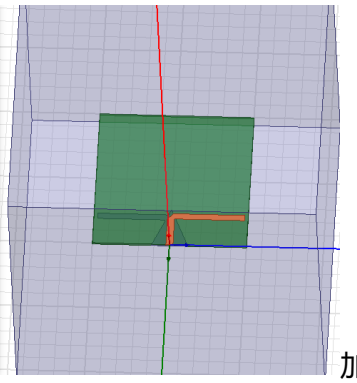


总结: 就是导线背面有地线做参考的才是微带线，如果导线背面没有地线做参考，就不是微带线。

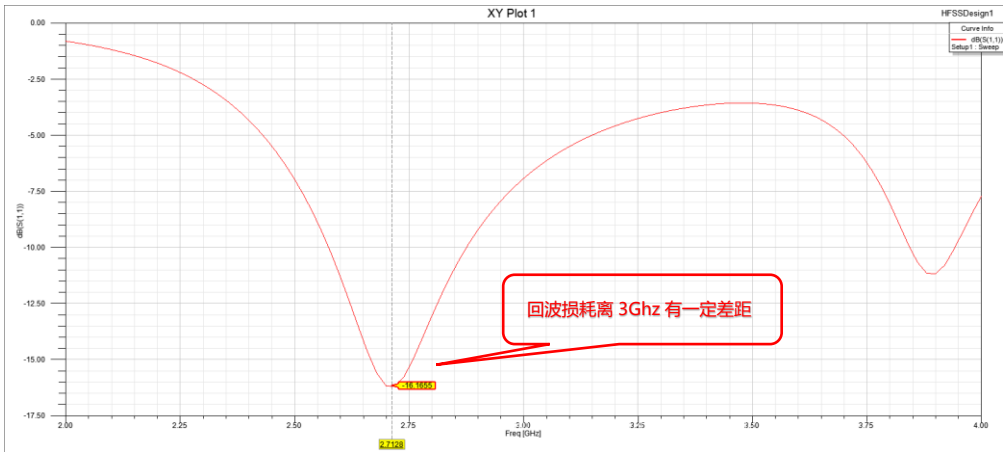


集中端口设计

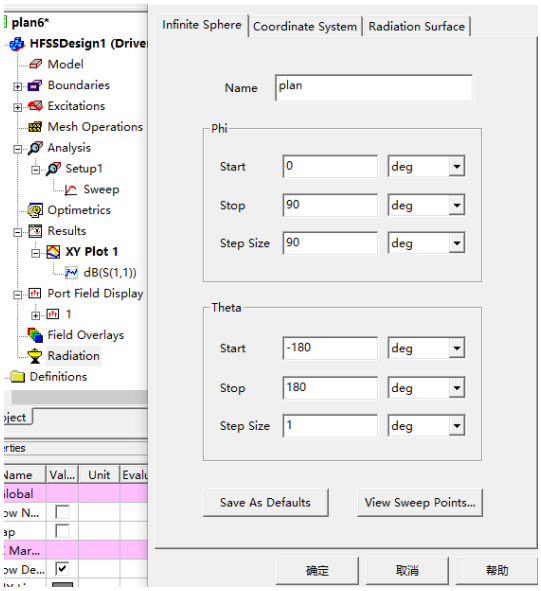


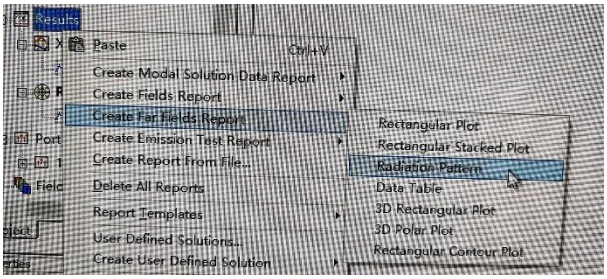


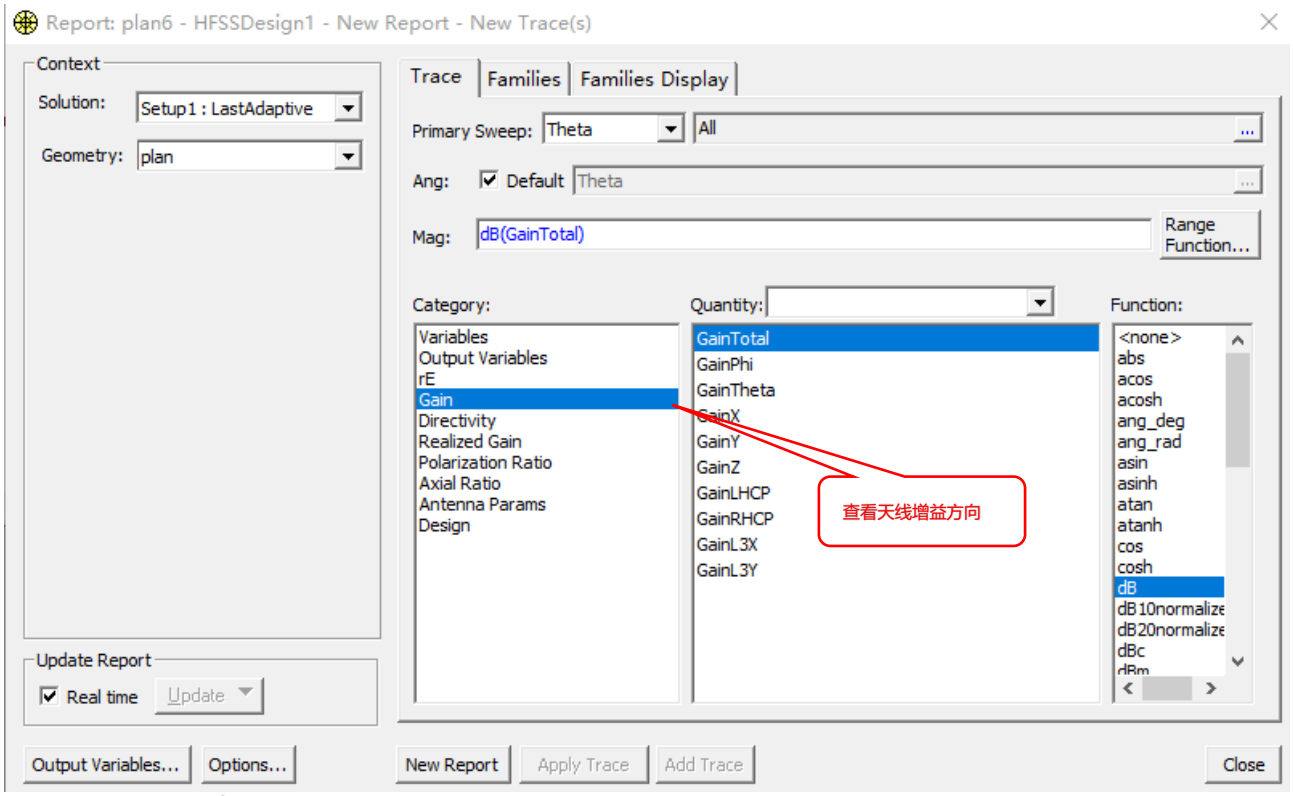
加入空气盒，开始仿真

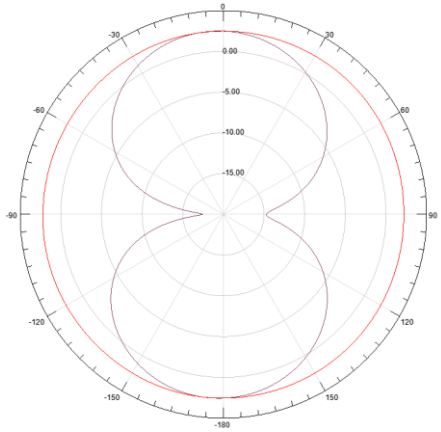


查看天线增益

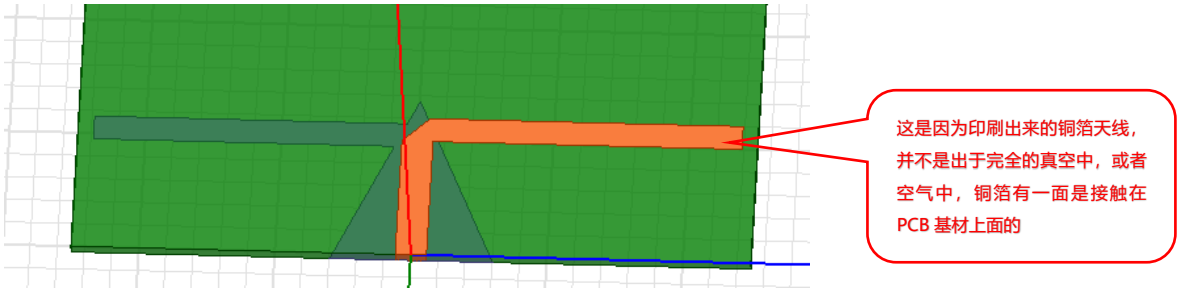

plan6*
HFSSDesign1 (Drive
Model
Boundaries
Excitations
Mesh Operations
Analysis
Setup1
Sweep
Optimetrics
Results
XY Plot 1
dB(S(1,1))
Port Field Display
1
Field Overlays
Radiation
Definitions
Infinite Sphere
Coordinate System
Radiation Surface
Name
plan
Phi
Start
0
deg
Stop
90
deg
Step Size
90
deg
Theta
Start
-180
deg
Stop
180
deg
Step Size
1
deg
Save As Defaults
View Sweep Points...
确定
取消
帮助


Results
Paste
Ctrl+V
Create Modal Solution Data Report
Create Fields Report
Create Far Fields Report
Create Emission Test Report
Create Report From File...
Delete All Reports
Report Templates
User Defined Solutions...
Create User Defined Solution
Rectangular Plot
Rectangular Stacked Plot
Radiation Pattern
Data Table
3D Rectangular Plot
3D Polar Plot
Rectangular Contour Plot

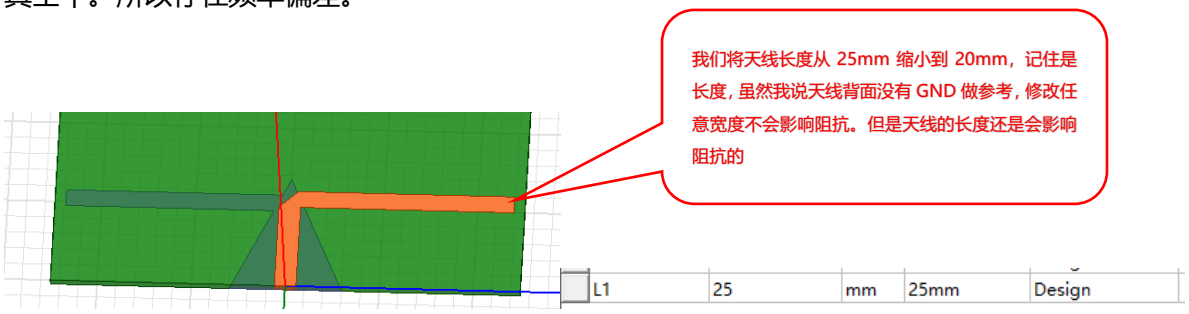

Report: plan6 - HFSSDesign1 - New Report - New Trace(s)
Context
Solution:
Setup1 : LastAdaptive
Geometry:
plan
Trace
Families
Families Display
Primary Sweep:
Theta
All
Ang:
Default
Theta
Mag:
dB(GainTotal)
Range Function...
Category:
Quantity:
Function:
Variables
Output Variables
rE
Gain
Directivity
Realized Gain
Polarization Ratio
Axial Ratio
Antenna Params
Design
GainTotal
GainPhi
GainTheta
GainX
GainY
GainZ
GainLHCP
GainRHCP
GainL3X
GainL3Y
查看天线增益方向
<none>
abs
acos
acosh
ang_deg
ang_rad
asin
asinh
atan
atanh
cos
cosh
dB
dB10normalize
dB20normalize
dBc
Update Report
Real time
Update
Output Variables...
Options...
New Report
Apply Trace
Add Trace
Close

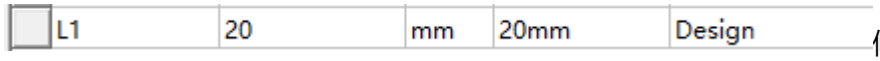

0
30
60
90
120
150
180
-30
-60
-90
-120
-150
0.00
-5.00
-10.00
-15.00

为什么偶极子天线仿真出来比 3Ghz 频段低?

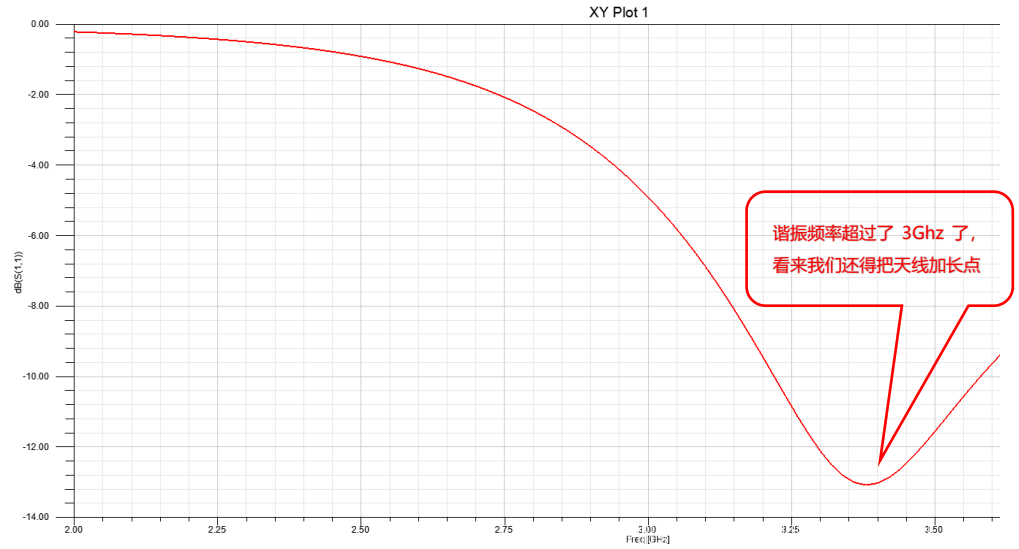


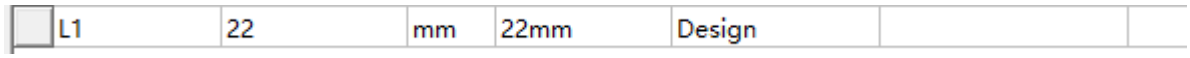
所以我们算出来的 100(mm)毫米波长是针对于真空中 3Ghz 的波长。所以在 PCB 上，是无法做到真空的计算效果的。不管是 TOP 层天线还是 GND 天线，总有一面是挨着 PCB 基材的，不是完全处于空气或者真空中。所以存在频率偏差。



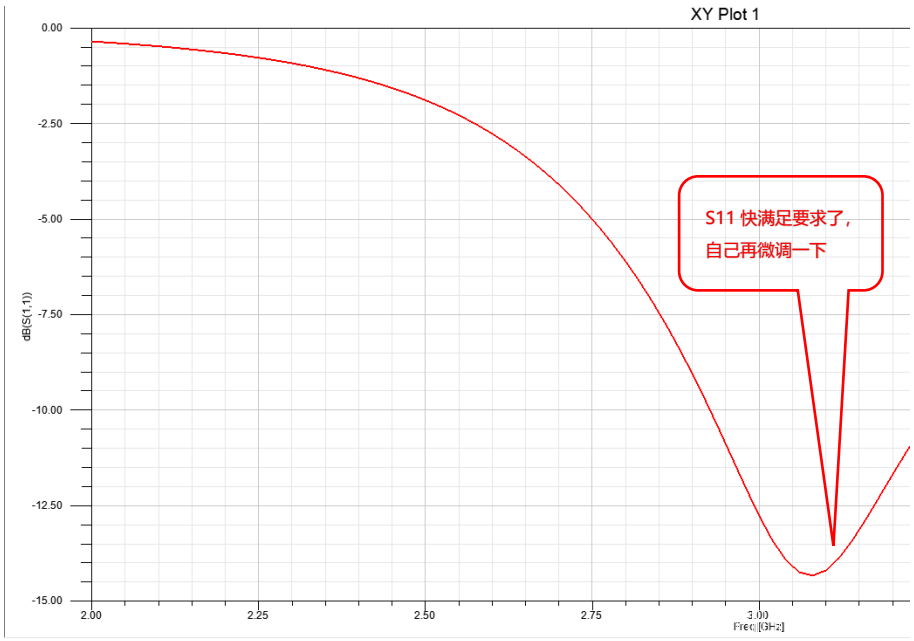


修改成 20mm

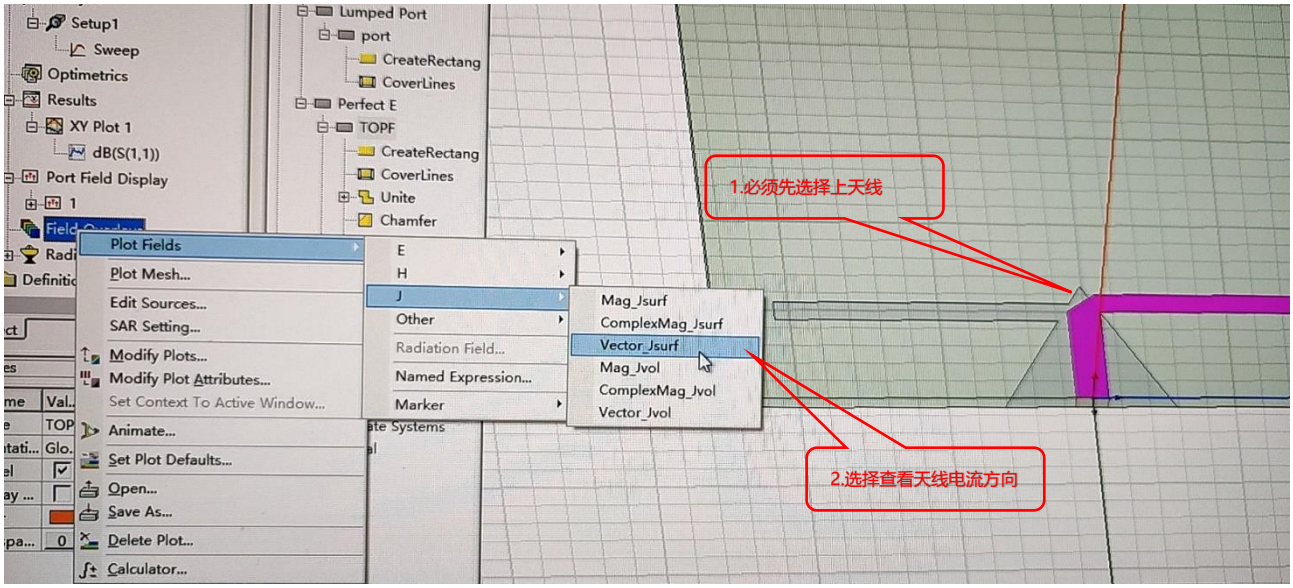


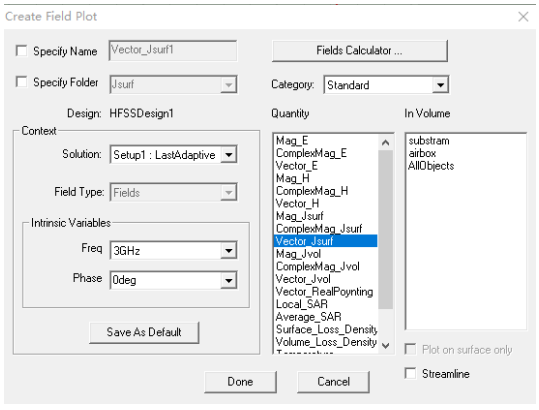


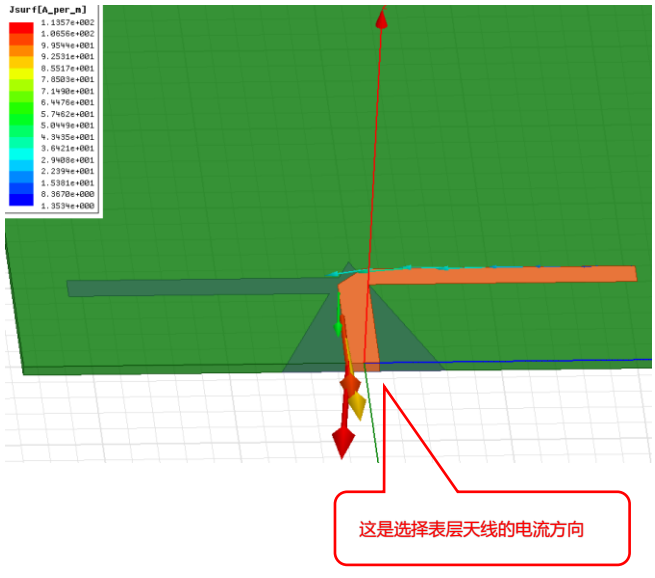
改成 22mm 长度试试

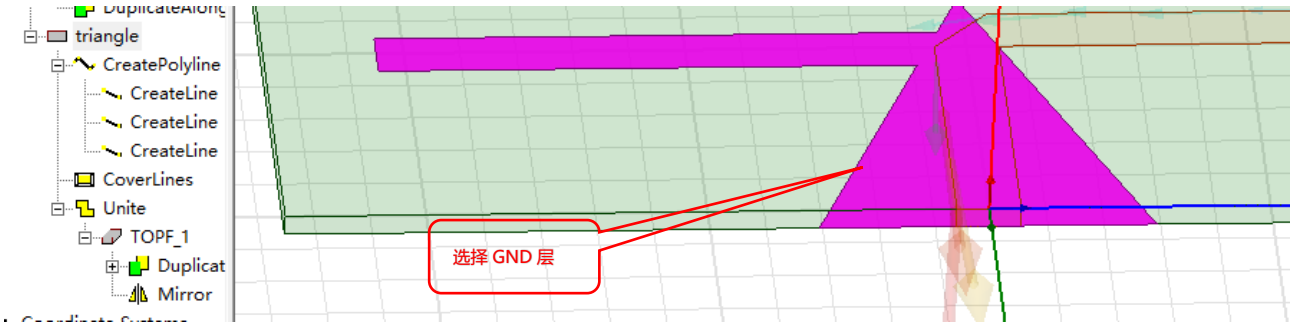

XY Plot 1
dB(S(1,1))
0.00
-2.50
-5.00
-7.50
-10.00
-12.50
-15.00
2.00
2.25
2.50
2.75
3.00
S11 快满足要求了，
自己再微调一下

天线电流方向查询

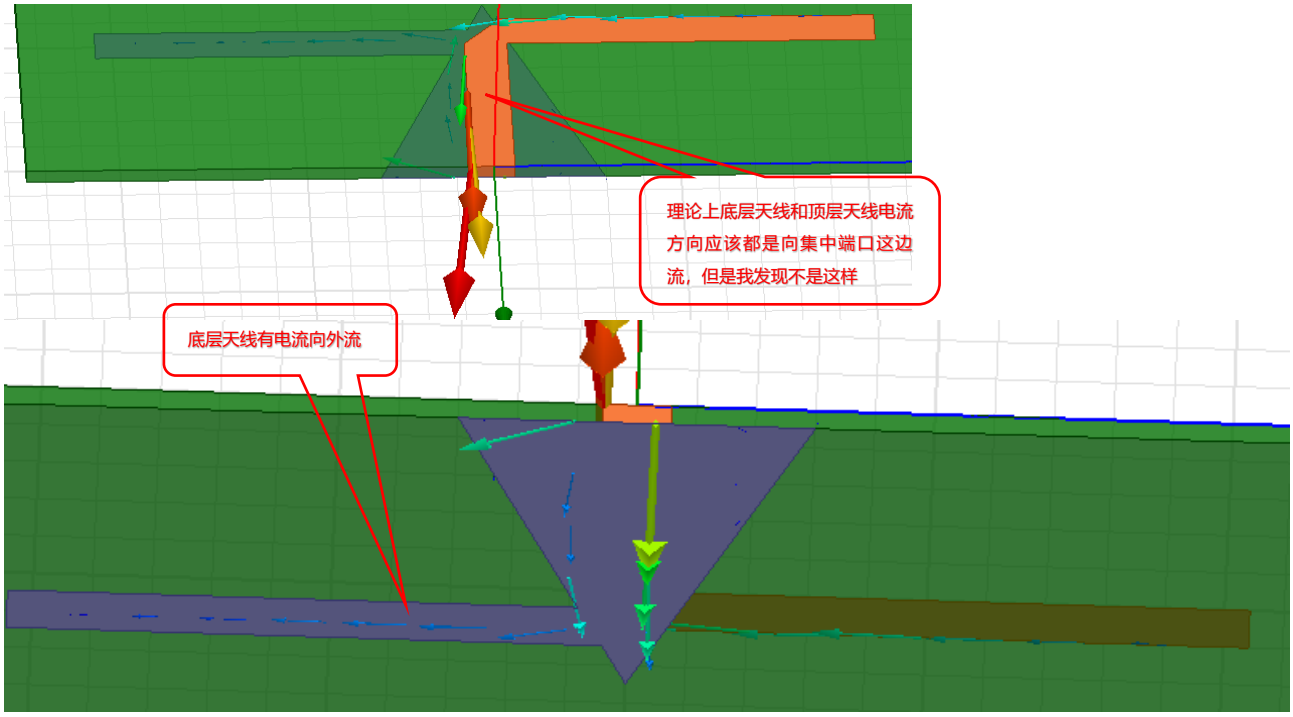

Setup1
Sweep
Optimetrics
Results
XY Plot 1
dB(S(1,1))
Port Field Display
1
Lumped Port
port
CreateRectang
CoverLines
Perfect E
TOPF
CreateRectang
CoverLines
Unite
Chamfer
Plot Fields
Plot Mesh...
Edit Sources...
SAR Setting...
Modify Plots...
Modify Plot Attributes...
Set Context To Active Window...
Animate...
Set Plot Defaults...
Open...
Save As...
Delete Plot...
Calculator...
E
H
J
Other
Radiation Field...
Named Expression...
Marker
Mag_Jsurf
ComplexMag_Jsurf
Vector_Jsurf
Mag_Jvol
ComplexMag_Jvol
Vector_Jvol
1.必须先选择上天线
2.选择查看天线电流方向

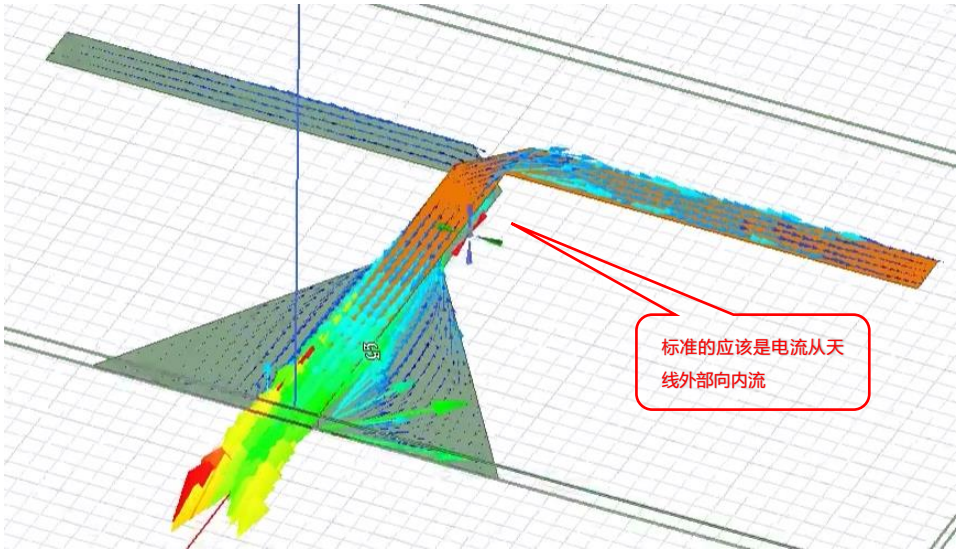

Create Field Plot
Specify Name
Vector_Jsurf1
Fields Calculator ...
Specify Folder
Jsurf
Category:
Standard
Design:
HFSSDesign1
Quantity
In Volume
Context
Solution:
Setup1 : LastAdaptive
Field Type:
Fields
Intrinsic Variables
Freq
3GHz
Phase
0deg
Save As Default
Mag_E
ComplexMag_E
Vector_E
Mag_H
ComplexMag_H
Vector_H
Mag_Jsurf
ComplexMag_Jsurf
Vector_Jsurf
Mag_Jvol
ComplexMag_Jvol
Vector_Jvol
Vector_RealPoynting
Local_SAR
Average_SAR
Surface_Loss_Density
Volume_Loss_Density
substram
airbox
AllObjects
Plot on surface only
Streamline
Done
Cancel


Jsurf[A_per_m]
这是选择表层天线的电流方向

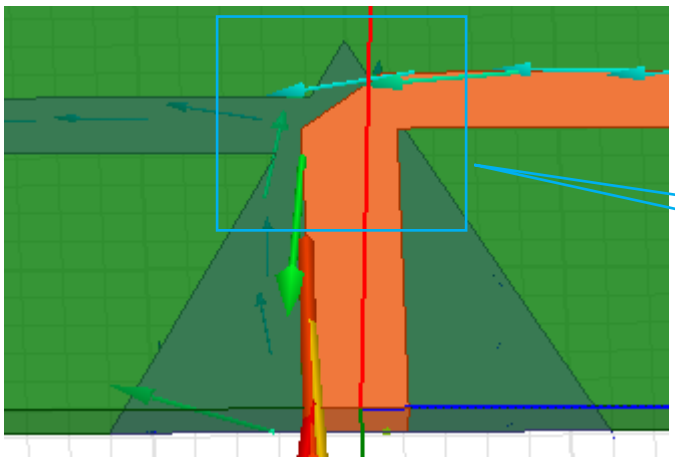


与前面同样的操作方式

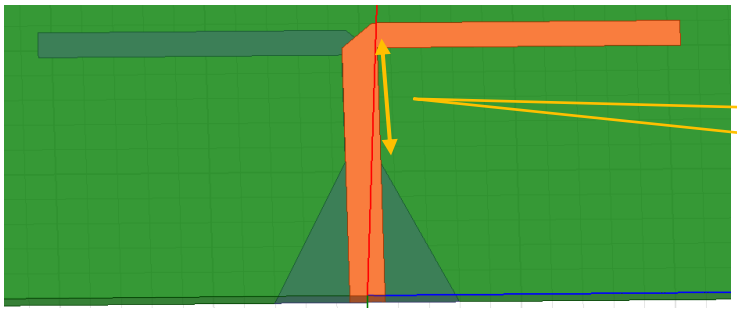




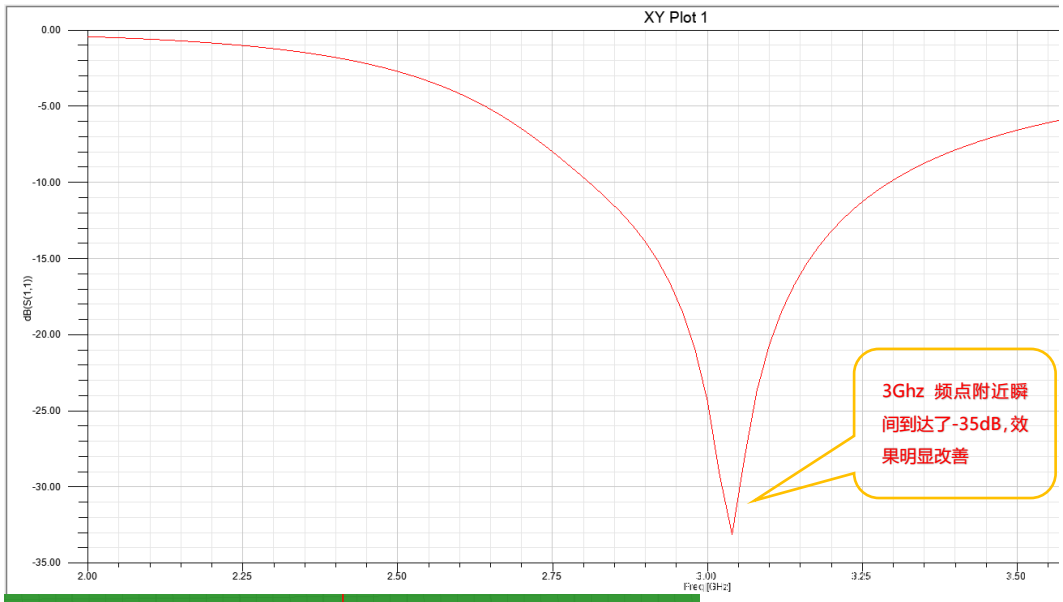
所以我的偶极子对称天线设计有问题，而且问题出在 GND 问题上。



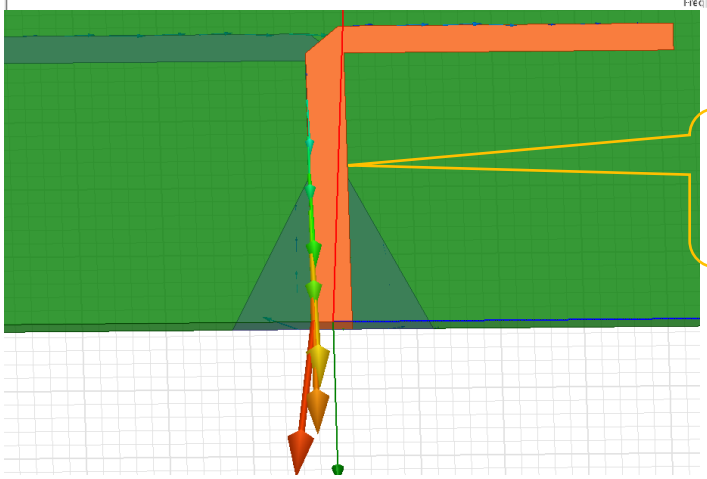
我的 GND 底层铜将天线交叉头的位置覆盖住了



在原来的基础上增加了10mm 长度，让 GND 避开交叉点

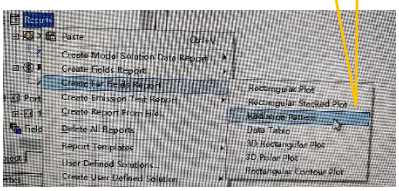


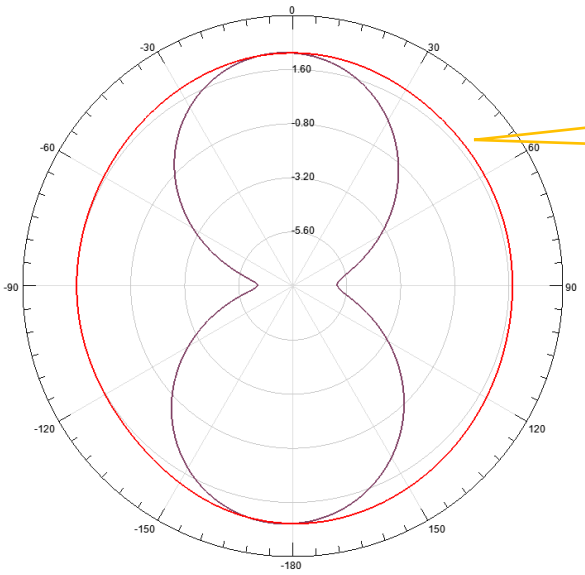
3Ghz 频点附近瞬间到达了-35dB，效果明显改善



不管顶层天线电流，还是底层天线电流，都是向内部流动，只有少许的向外部流动，问题得到改善

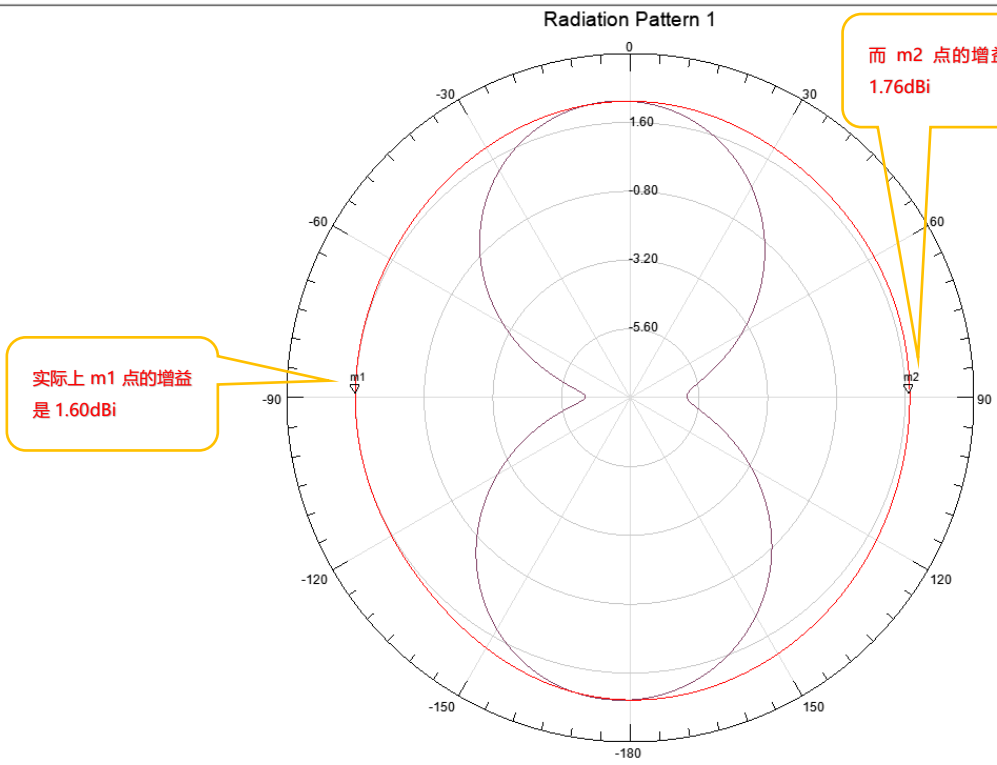
看看天线方向图





理论上偶极子天线应该是个辐射圆形，但是实际上，红色这个线没有这么圆

Name	Theta	Ang	Mag
m1	-89.0000	-89.0000	1.6020
m2	89.0000	89.0000	1.7623

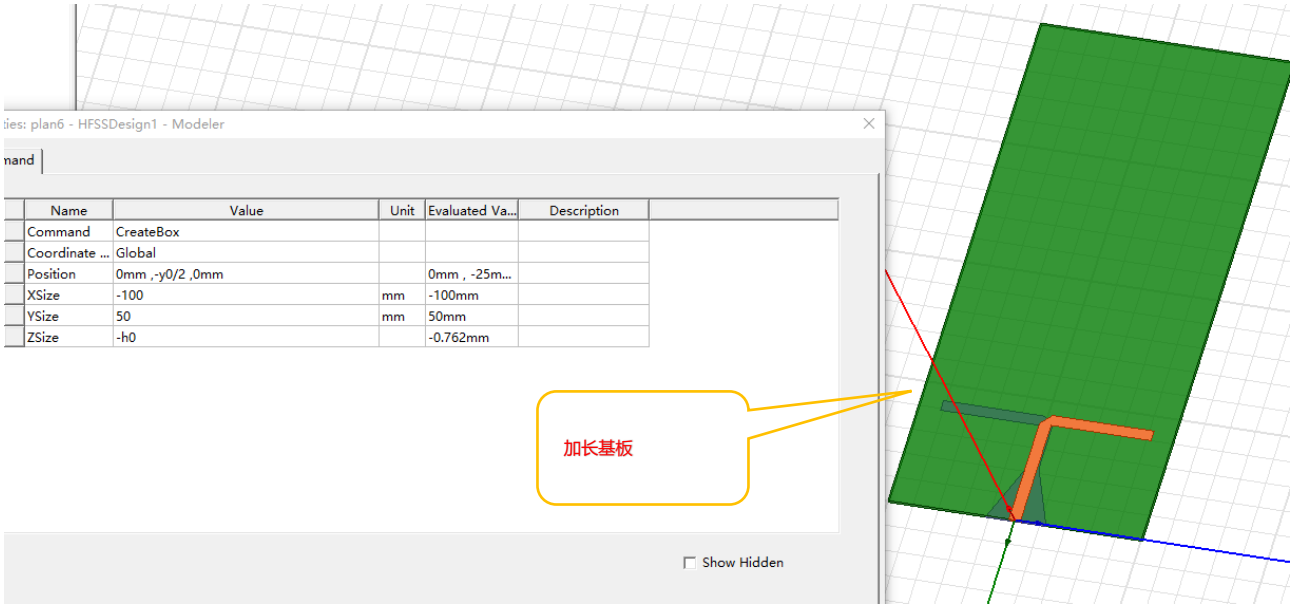


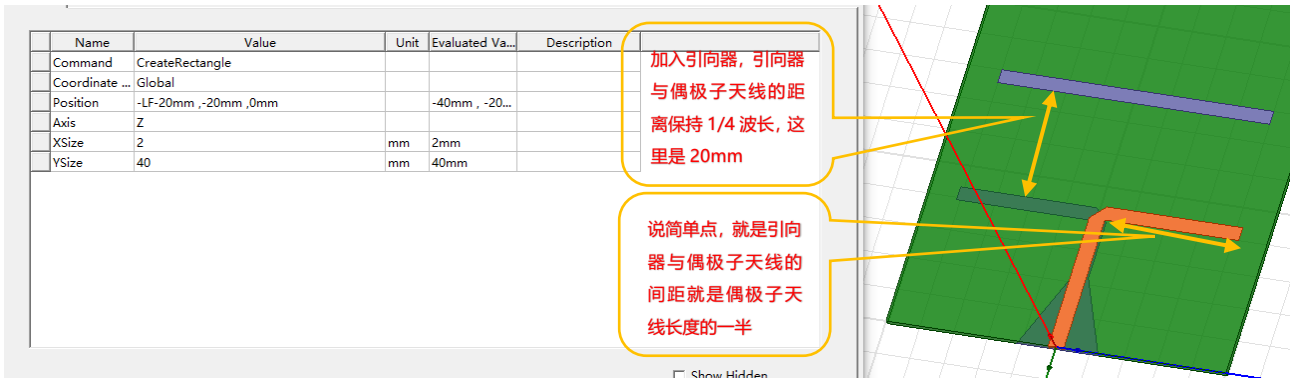
而 m2 点的增益是 1.76dBi

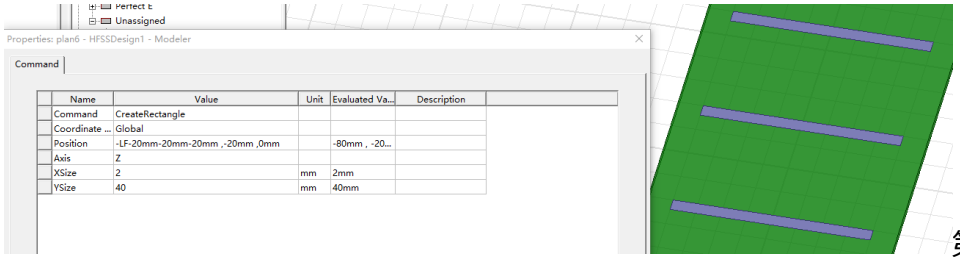
实际上 m1 点的增益是 1.60dBi

很明显，两边增益不一致，怎么会是圆呢？当然不是圆。

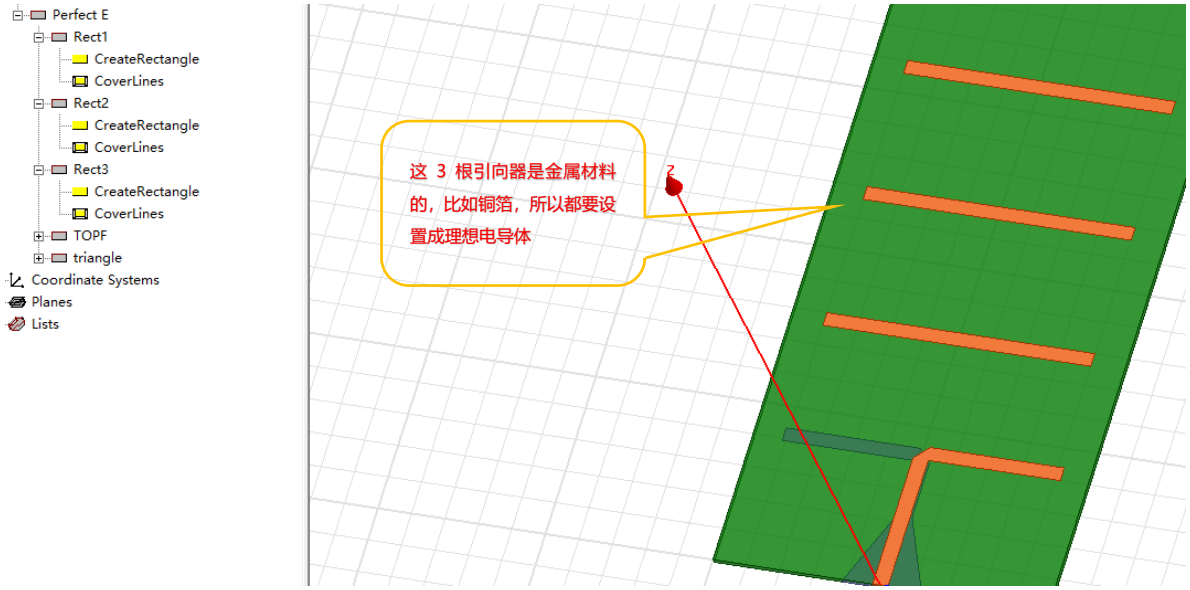
改进偶极子天线为八木天线



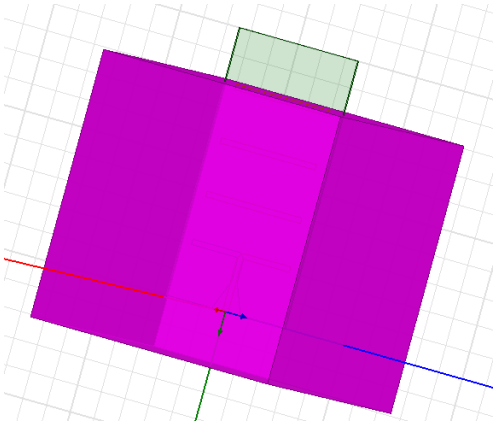


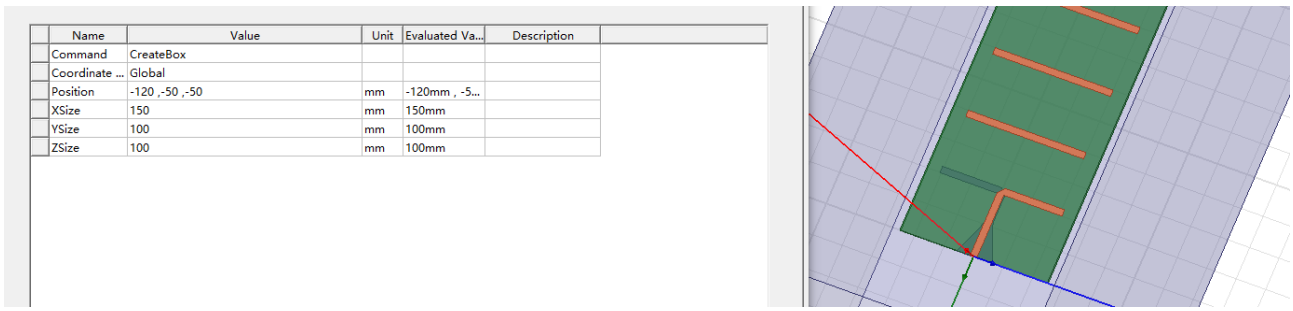


第 3 根也是



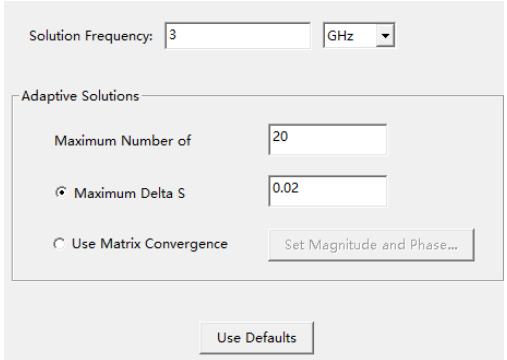
空气盒子修改

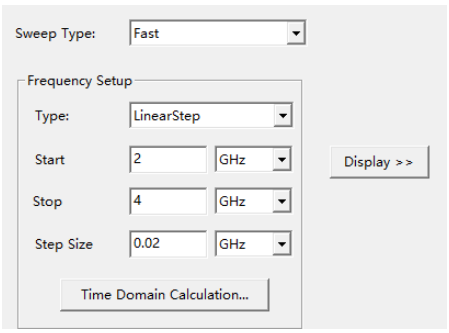




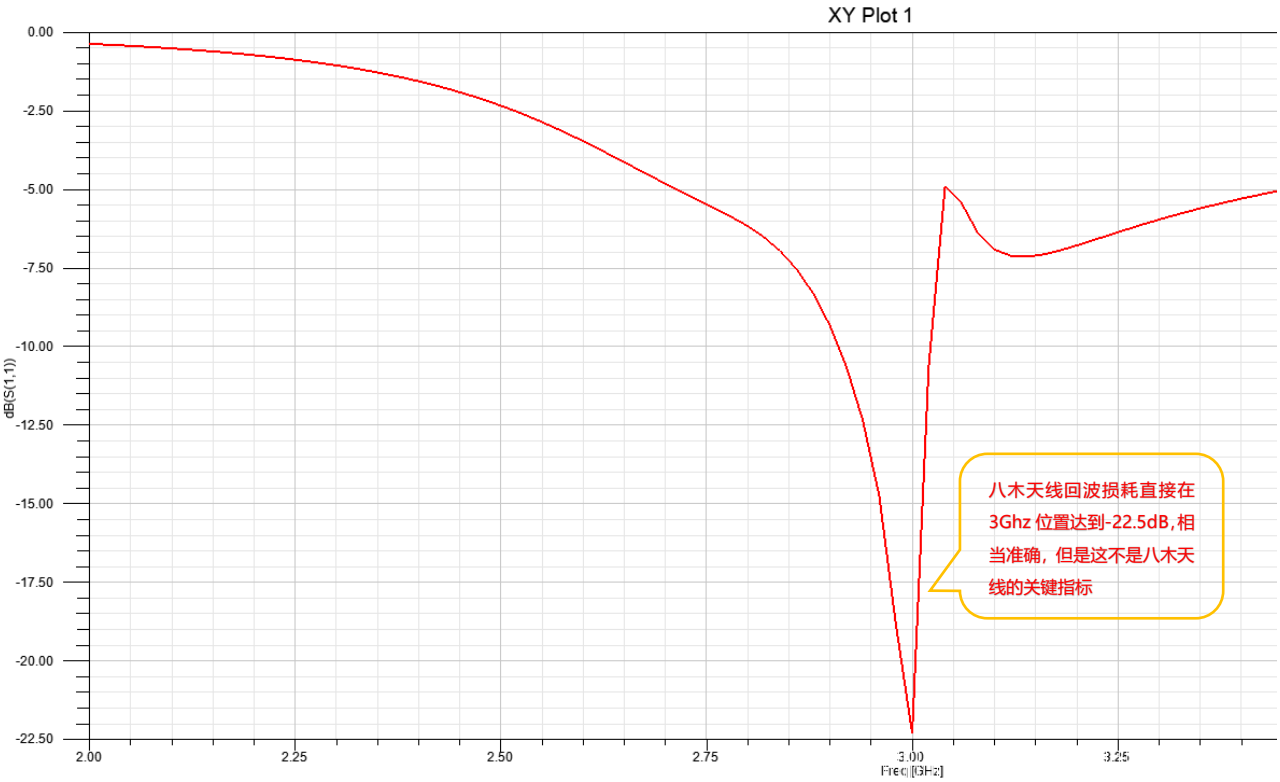
Name	Value	Unit	Evaluated Va...	Description
Command	CreateBox			
Coordinate ...	Global			
Position	-120 ,-50 ,-50	mm	-120mm , -5...	
XSize	150	mm	150mm	
YSize	100	mm	100mm	
ZSize	100	mm	100mm	

开始仿真，

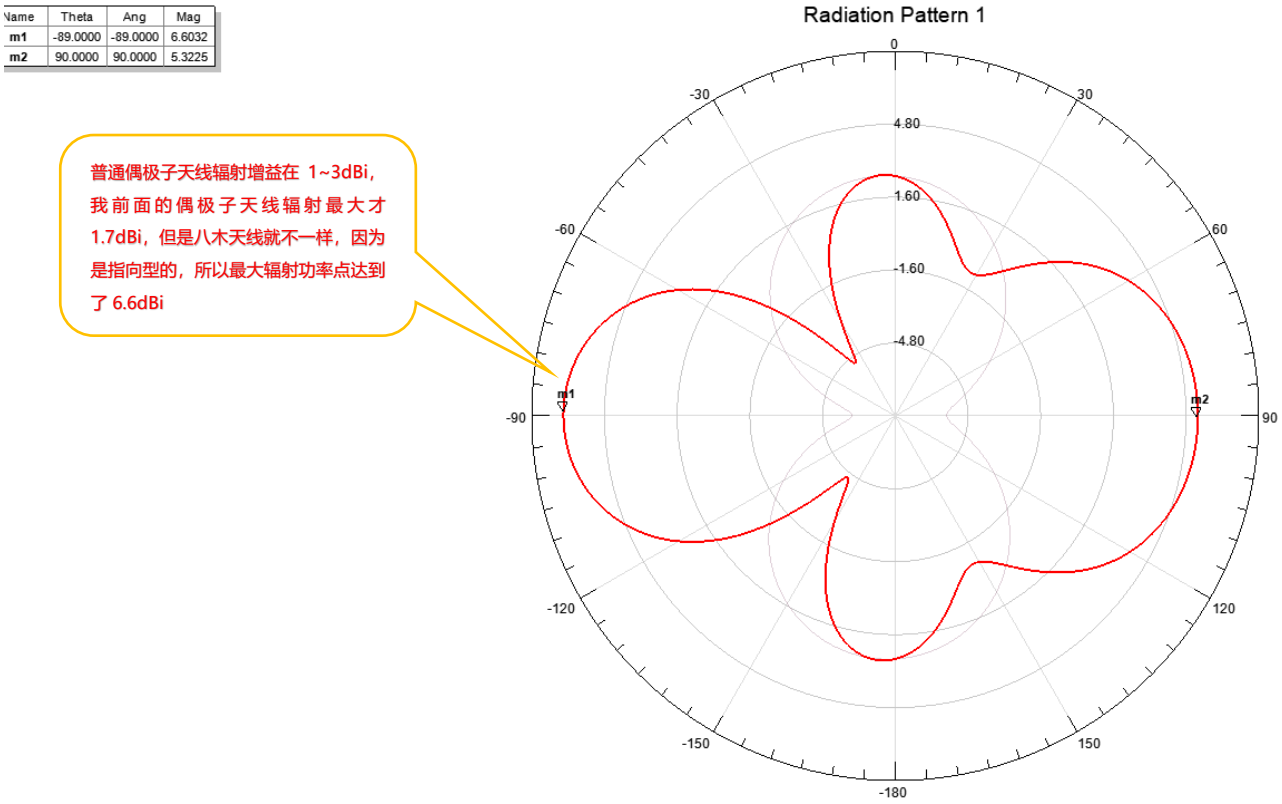


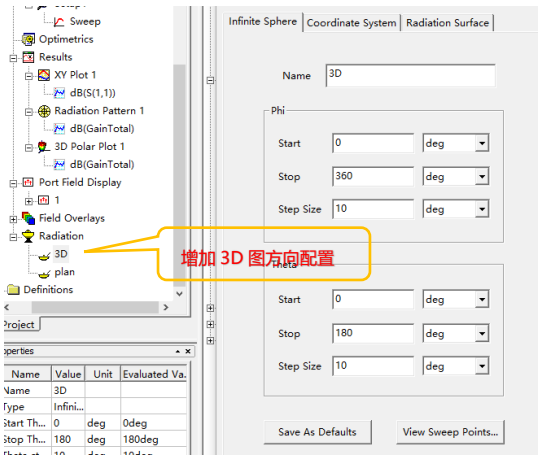


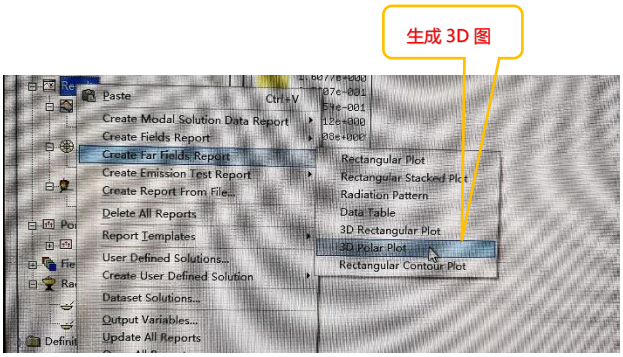
我们还是仿真的 3Ghz

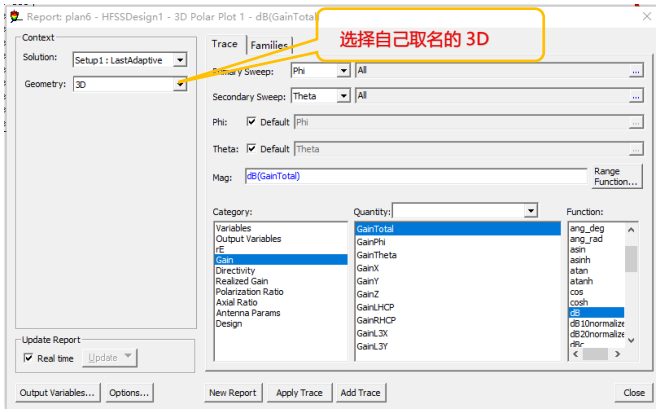


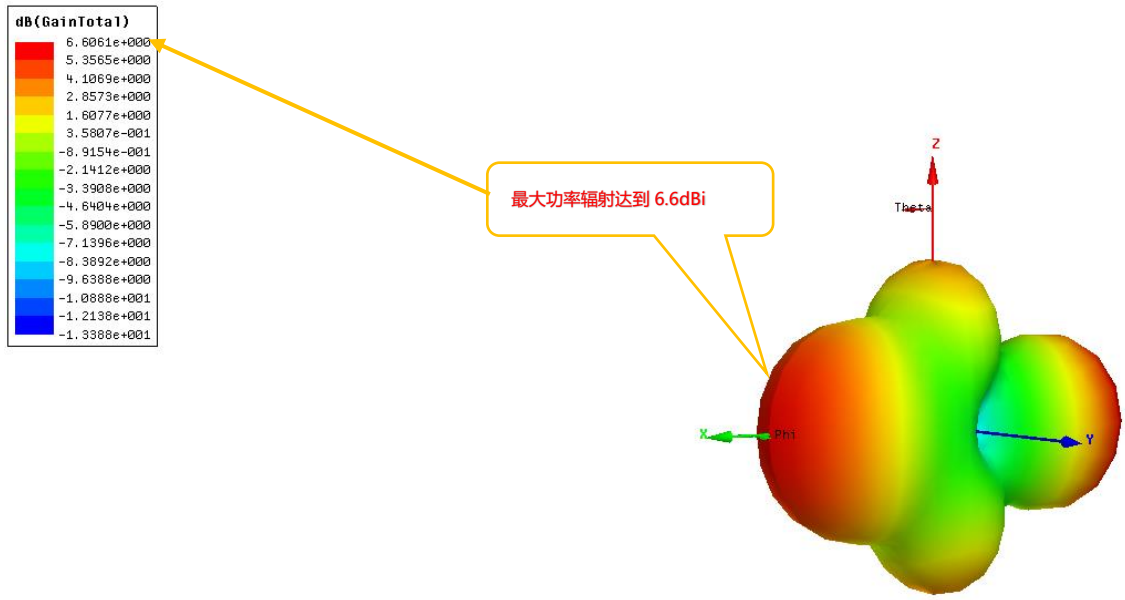
八木天线的关键指标是天线辐射的方向图，看看功率主要向哪个方向辐射。





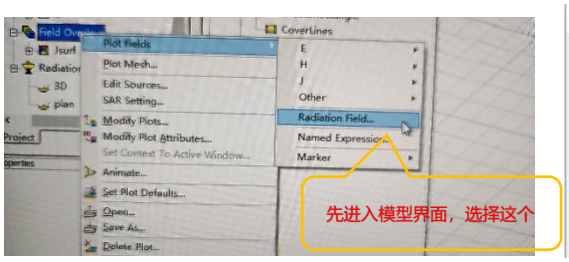


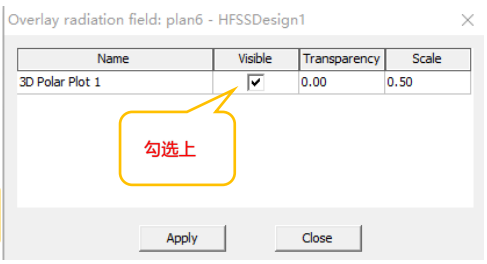


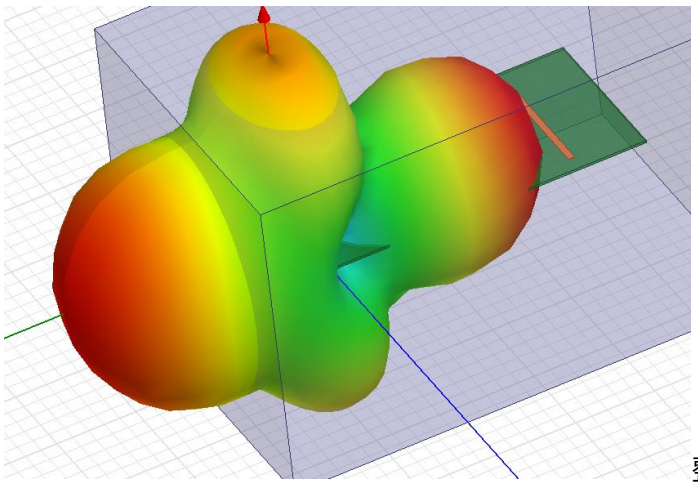


所以八木天线的结构，就把全向的偶极子天线做成了指向性天线。

下面我们看看辐射方向到底处于 PCB 板上面位置

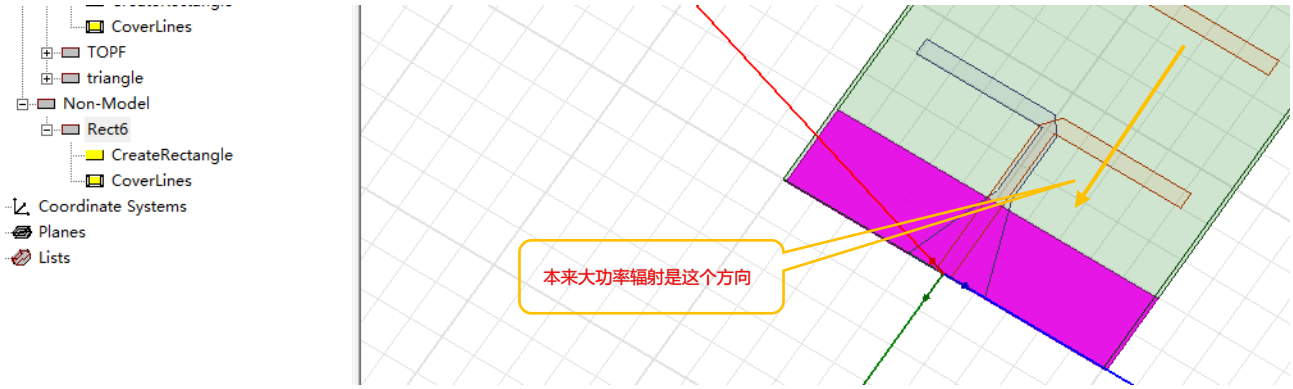


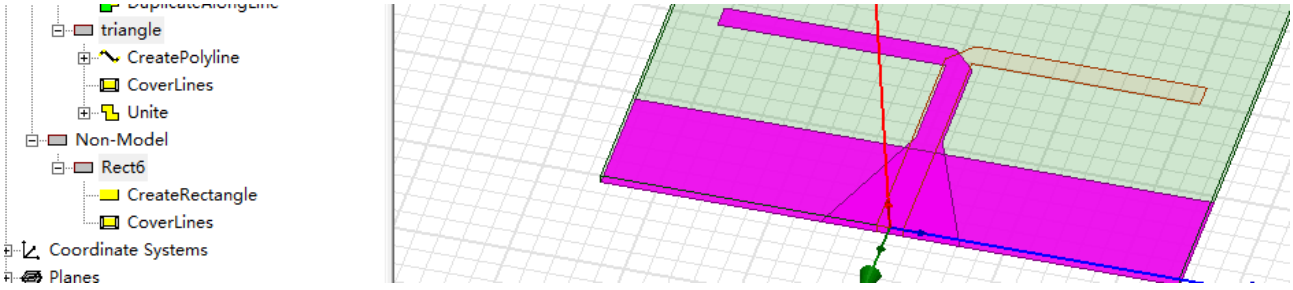




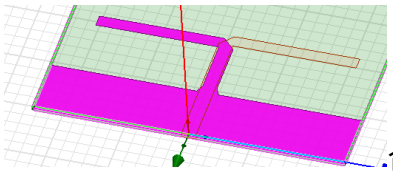
整个辐射方向是向 PCB 后方向辐射的。

我在 PCB 背面加入长条的 GND 金属铜箔，将辐射抵挡回去





将背面长条的 GND 铜箔，与三角形 GND 相合并



合并后的样子。设置为理想电导体。自行测试，这里就不多讲了。