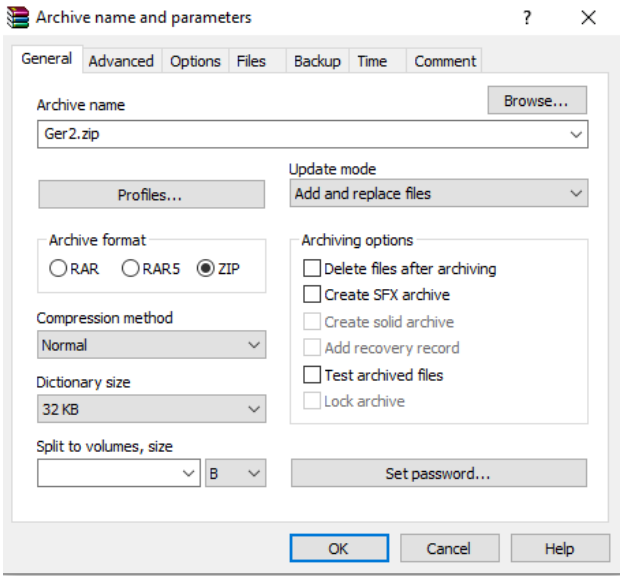
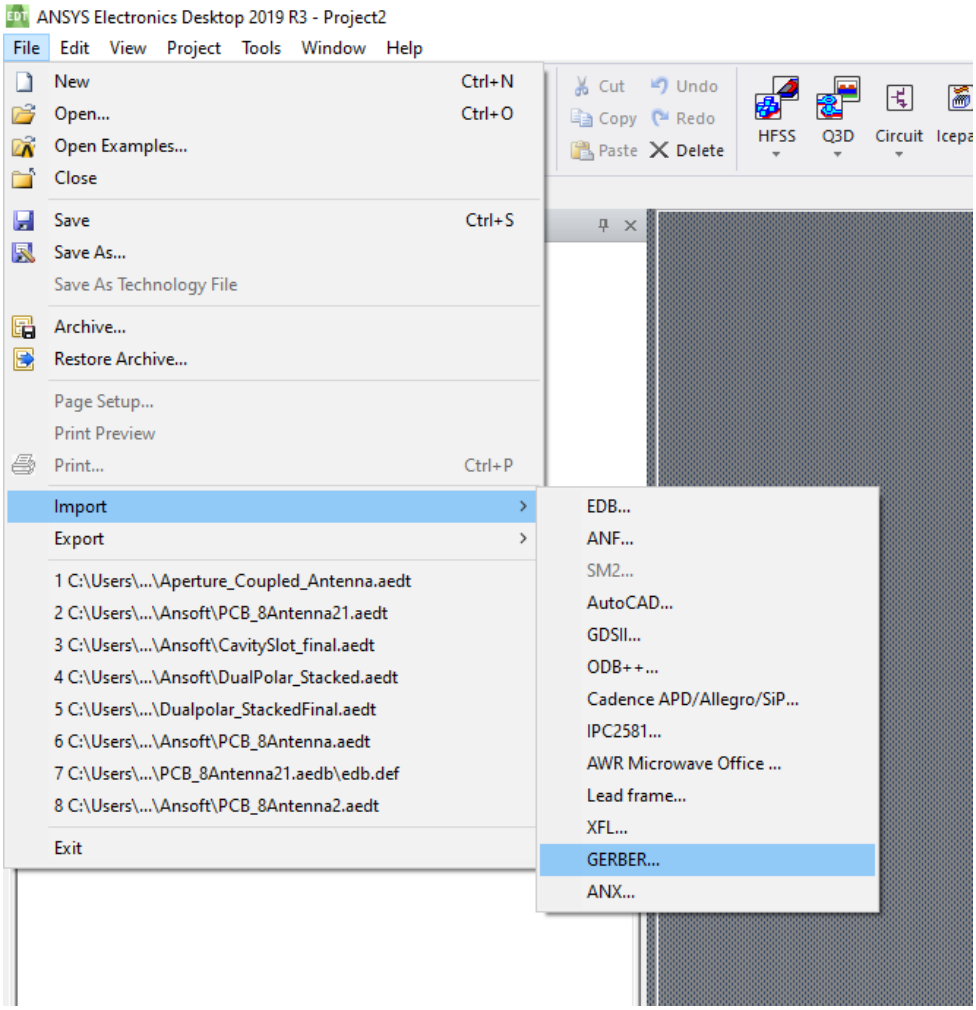
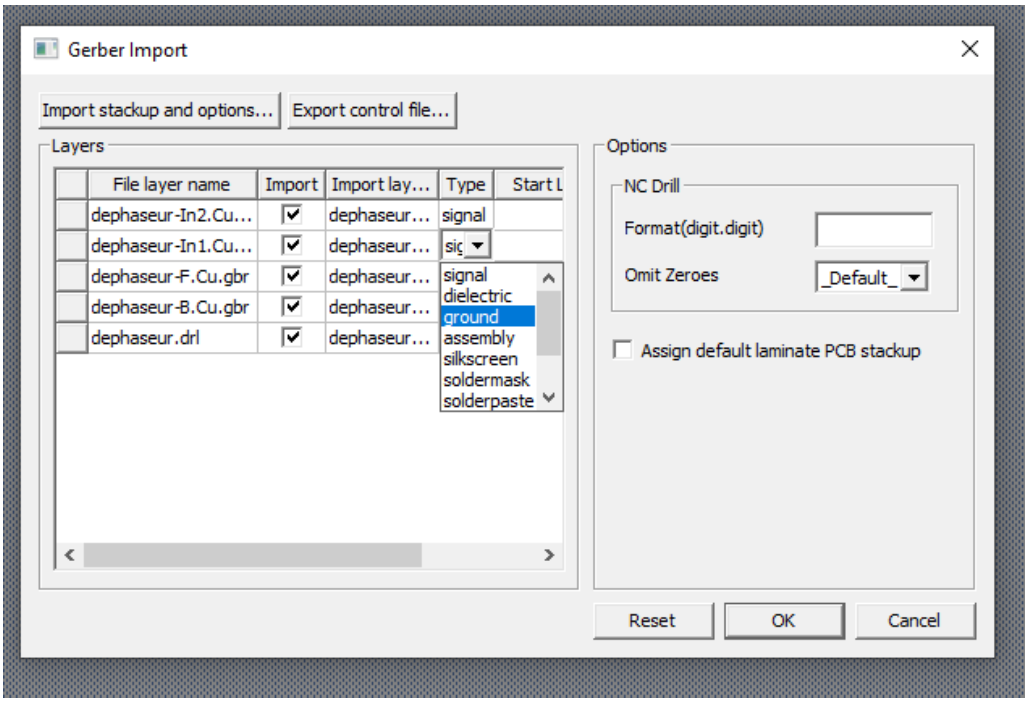
1. We will need, all copper layouts plus drill file with Gerber format. A zip file is required to contain all gerber files:



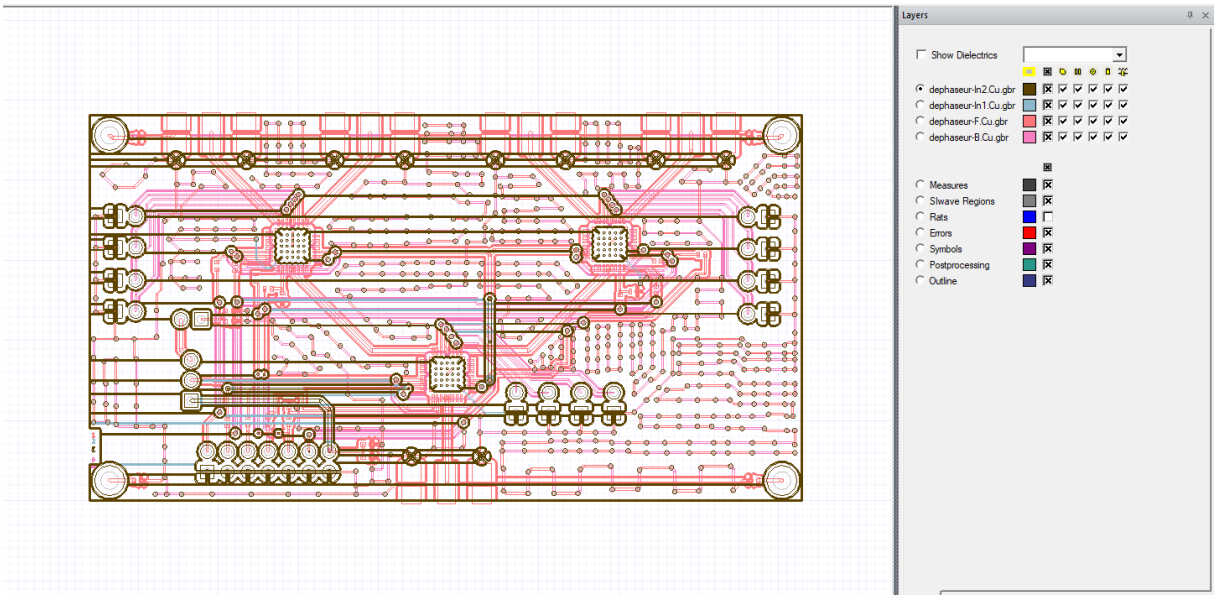
2. Import Zip of Gerbers



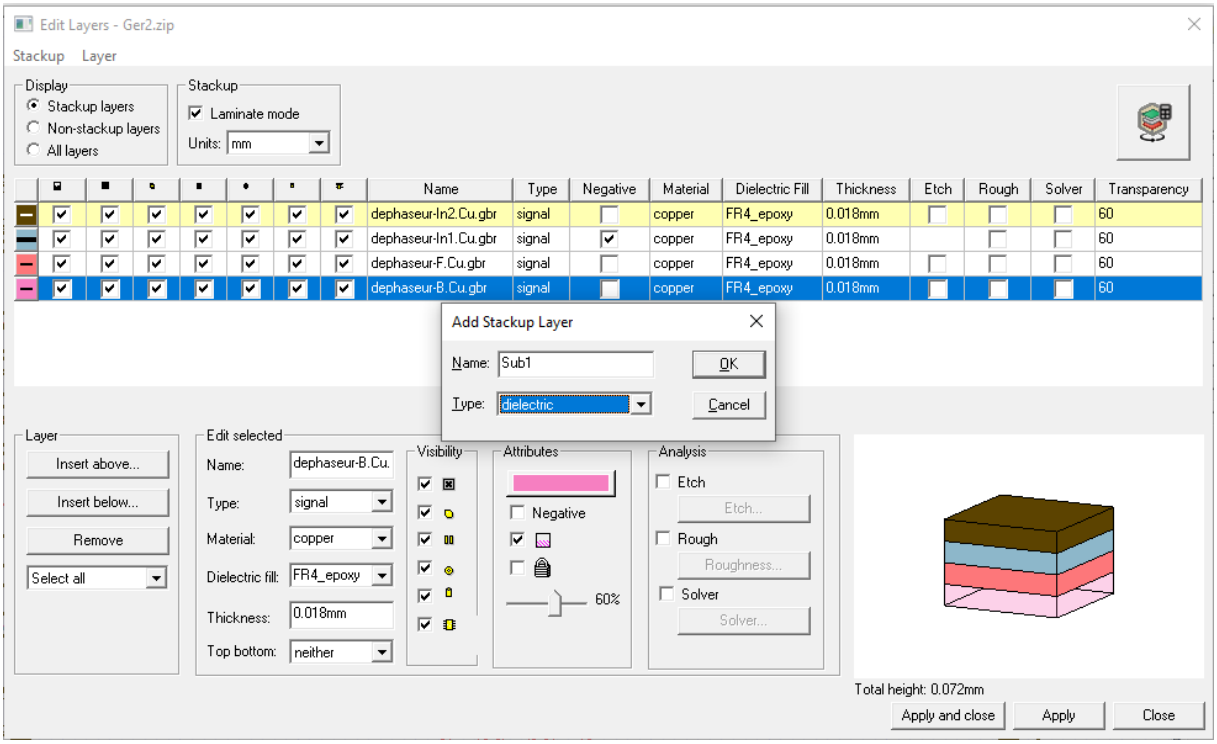
3. Choosing Ground layer. Power layer might be required in some cases:

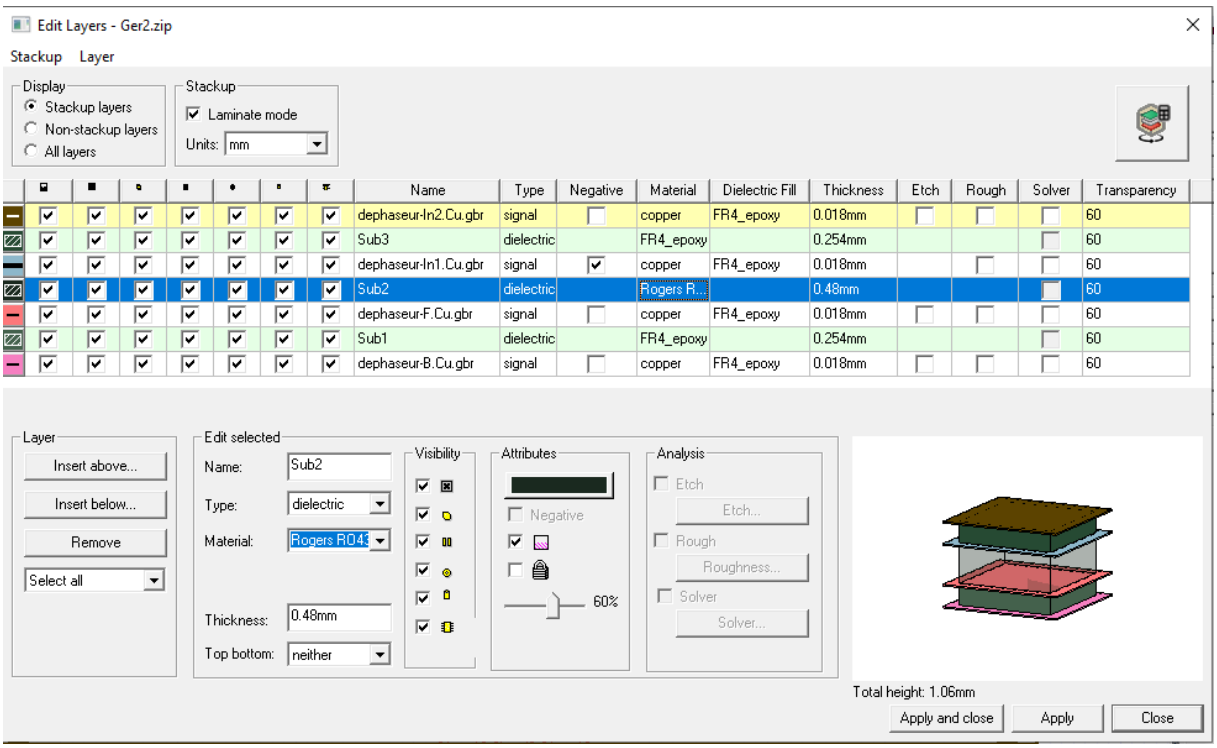


Imported PCB as following, containing only copper layers

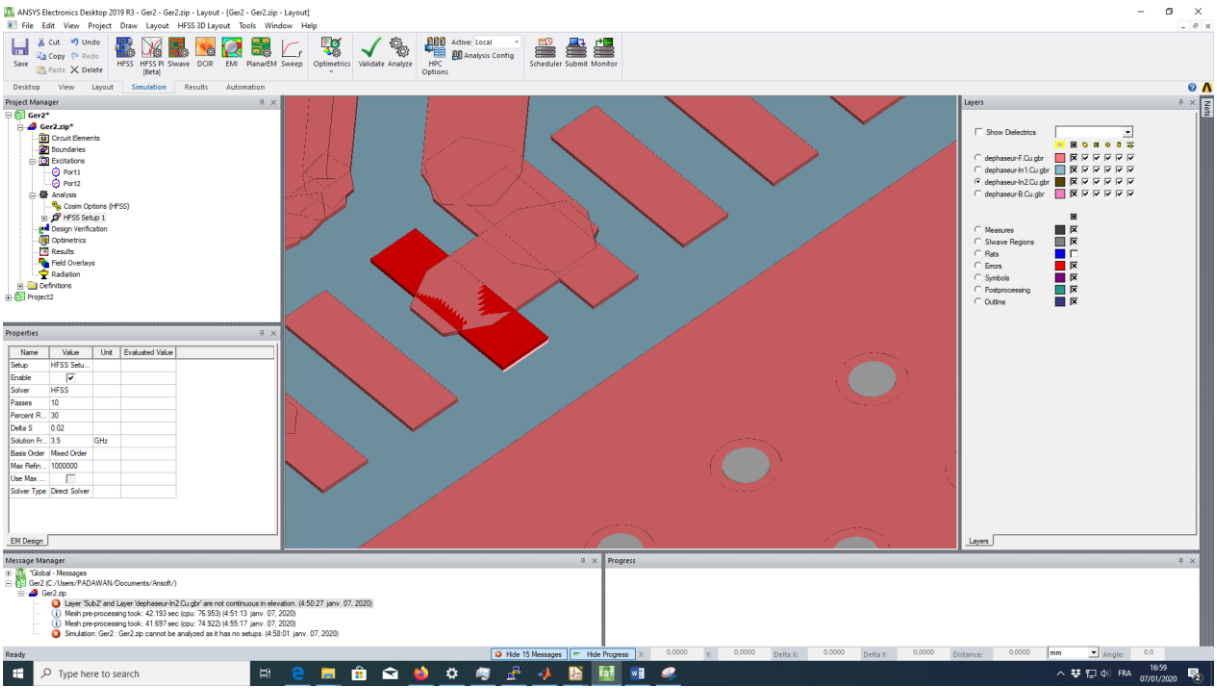


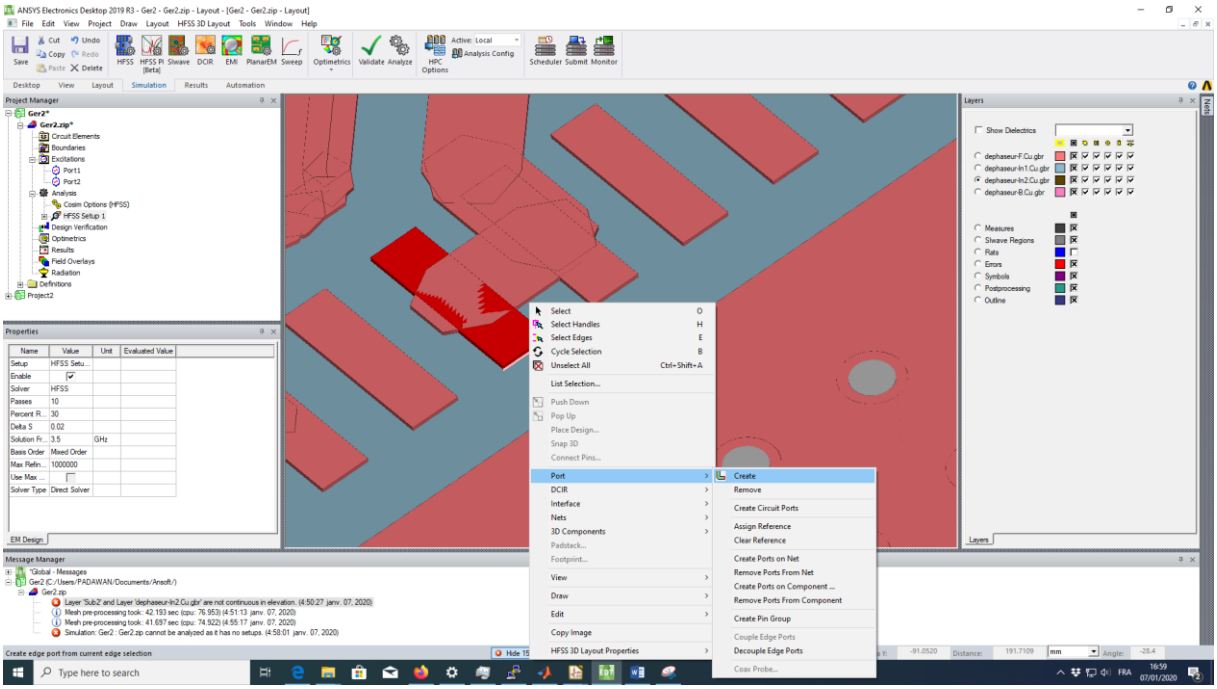
4. Add Dielectric layers and assign layers' width

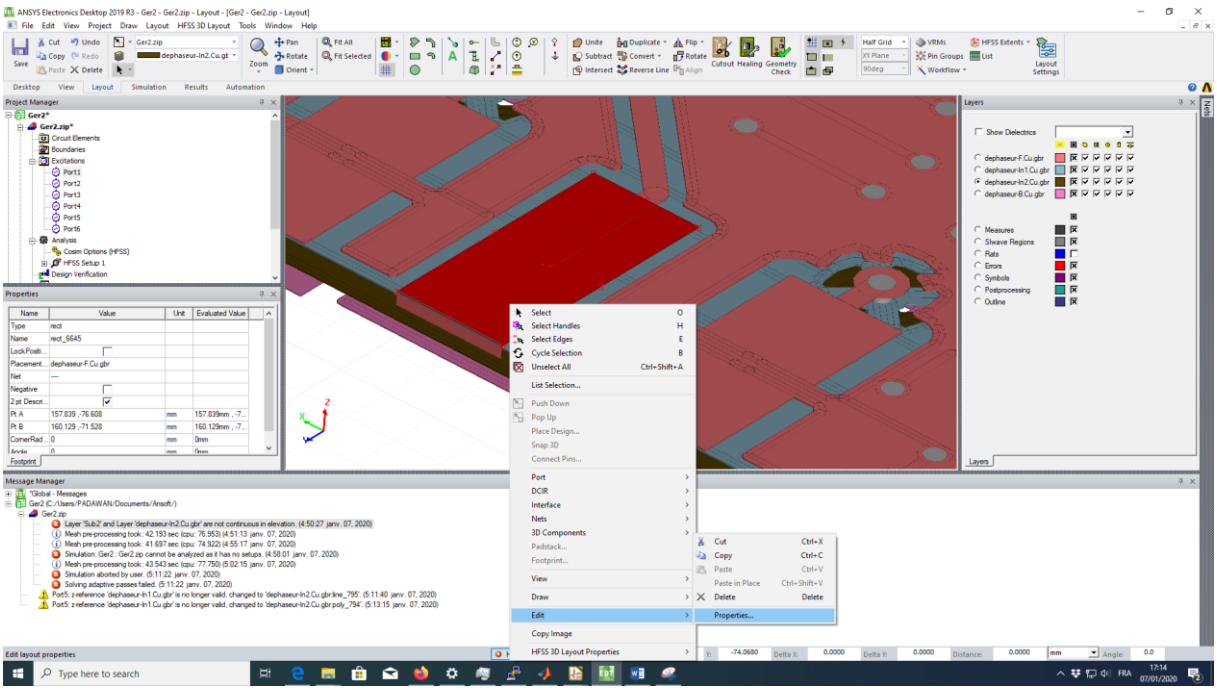


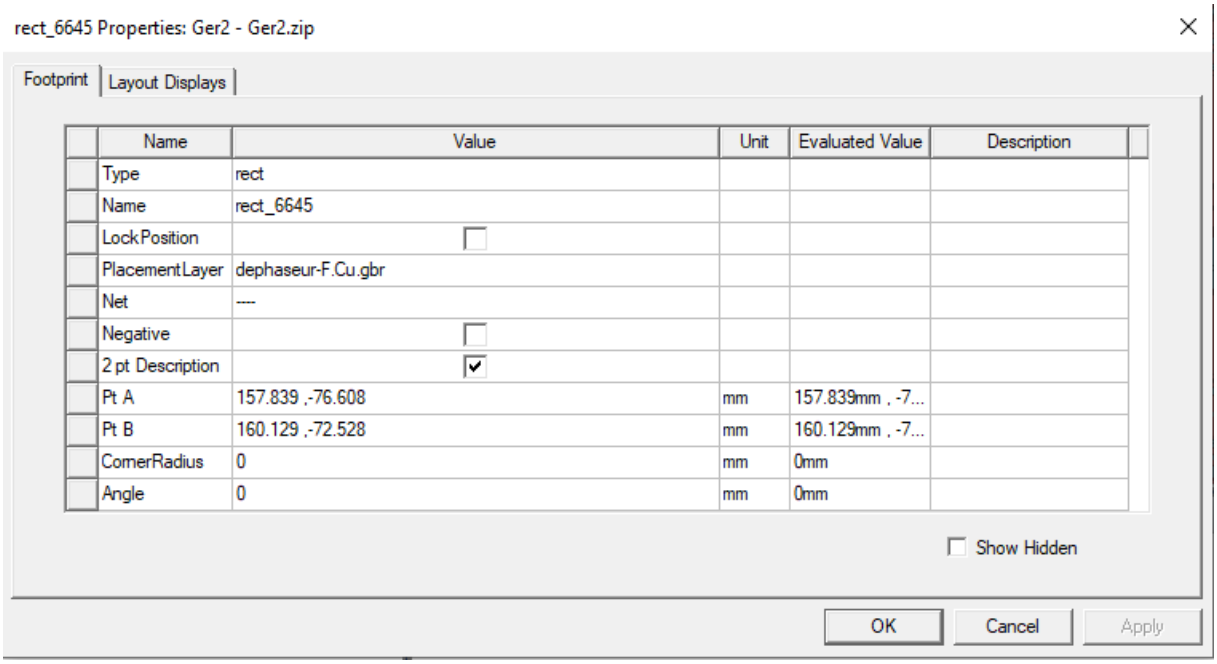


5. Assign EDGE ports and modify geometry


ANSYS Electronics Desktop 2019 R3 - Ger2 - Ger2.zip - Layout - [Ger2 - Ger2.zip - Layout]
File Edit View Project Draw Layout HFSS 3D Layout Tools Window Help
Project Manager
Properties
Layers
Message Manager
Progress
Type here to search

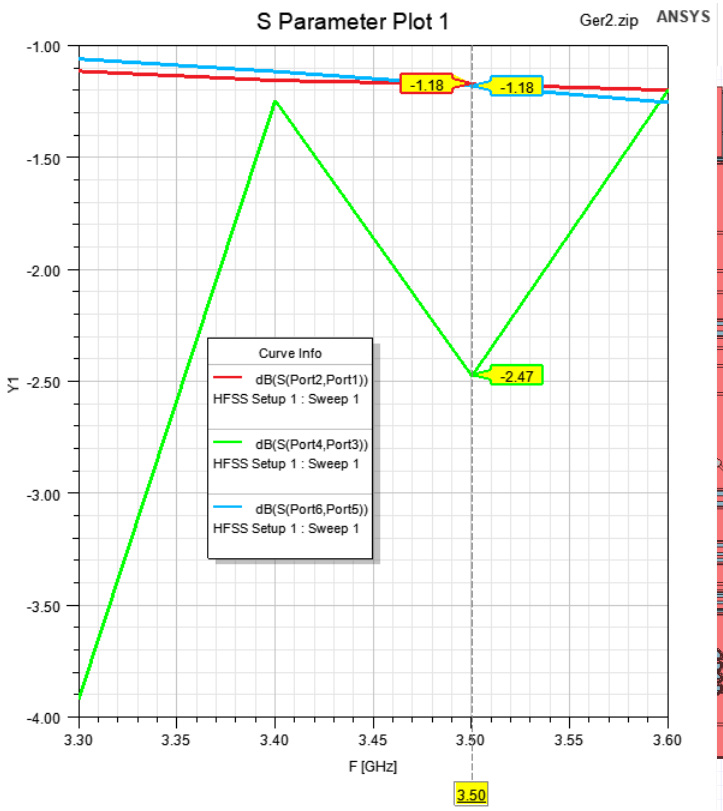

ANSYS Electronics Desktop 2019 R3 - Ger2 - Ger2.zip - Layout - [Ger2 - Ger2.zip - Layout]
Select
Select Handles
Select Edges
Cycle Selection
Unselect All
List Selection...
Push Down
Pop Up
Place Design...
Snap 3D
Connect Pins...
Port
DCIR
Interface
Nets
3D Components
Padstack...
Footprint...
View
Draw
Edit
Copy Image
HFSS 3D Layout Properties
Create
Remove
Create Circuit Ports
Assign Reference
Clear Reference
Create Ports on Net
Remove Ports From Net
Create Ports on Component ...
Remove Ports From Component
Create Pin Group
Couple Edge Ports
Decouple Edge Ports
Coax Probe...
Create edge port from current edge selection
Type here to search

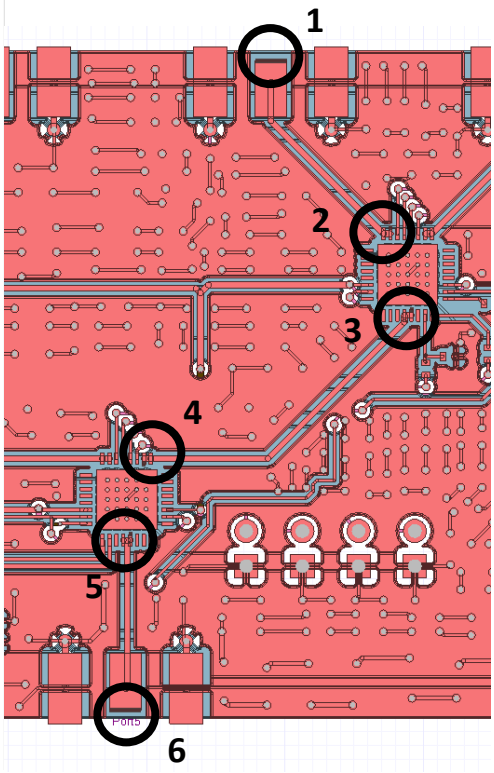




6. Analyze the design:

❖ Transmission





For every phase-shifter passed, there is 7.1dB loss. So, For this design, from port 6 to port 1:

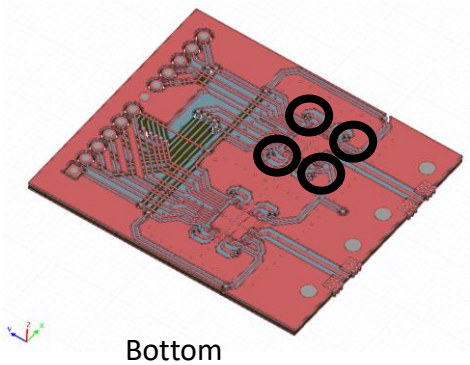
$Loss_{TOT}$ = 2*7.1 + 2*1.2 + 2.5 = 19.1 dB @3.5GHz

With 4 parallel ports:

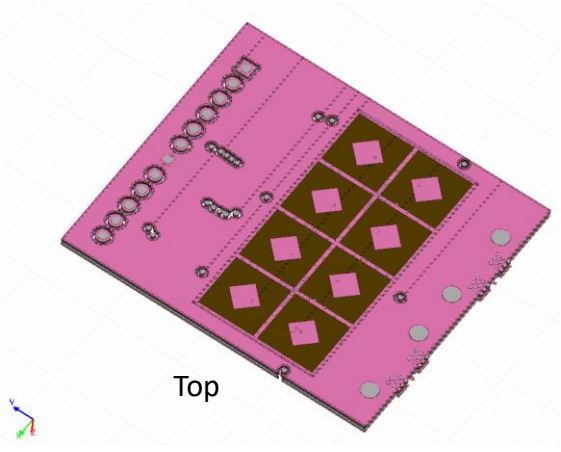
$Loss_{In\text{-}Out}$ = $Loss_{TOT}$ + 6dB = 13.1dB @3.5GHz

- Is the TM lines' width optimal?

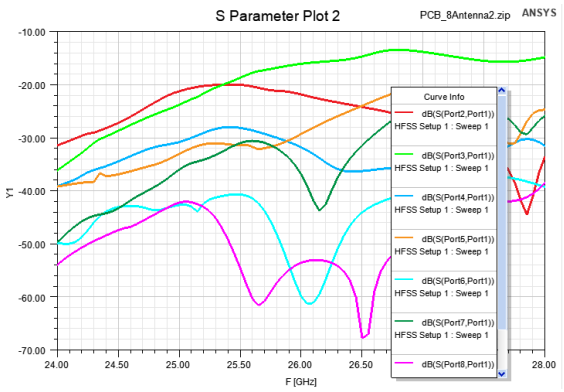
❖ Radiation

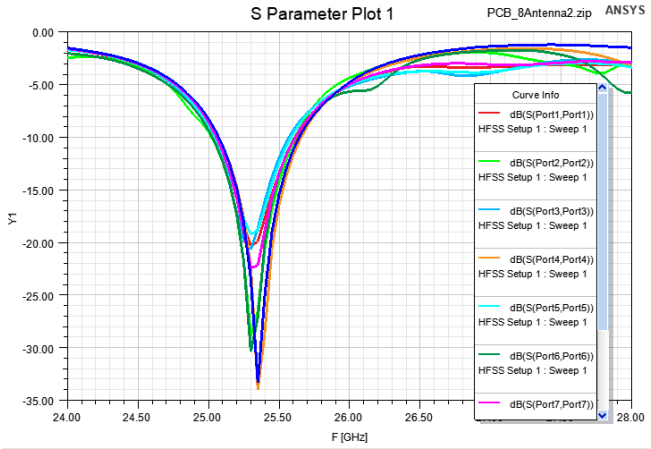


Bottom



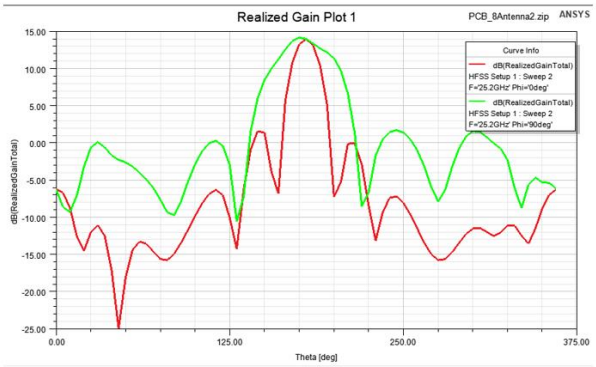
Top

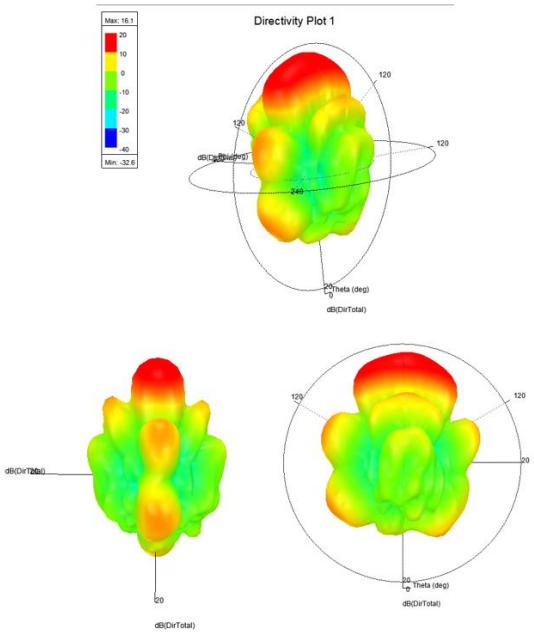




4x2 array PCB

- Array Gain and Pattern





- Is frequency band large enough?