① Microstrip Antenna for 2.4 GHZ :-

* Open ADS

* Create New Workspace – Create New folder – select all libraries

Click layout icon * New Layout – give cell name – OK (select 1st mm)

design

* frequency : 2.4 GHz

$$\lambda = \frac{c}{f\sqrt{\varepsilon_r}} = \frac{3\times10^8}{2.4\times10^9\sqrt{4.3}} = 0.06028\,m$$

$\therefore \lambda = 60.28$ mm

⇒ dimensions of radiator } $\frac{\lambda}{2} \times \frac{\lambda}{2}$

dielectric medium } "FR4"

⇒ Feed → length $= \frac{\lambda}{4}$; width = 3 mm

* Click rectangular icon & put it in black window opened.

Click box : in properties : (draw random box)

lower left x : -15

lower left y : 0

width : 30

height : 30

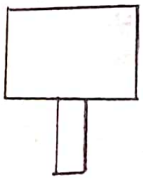
* Again draw random rectangular box to create feed.

Click that box : properties : lower left x : -1.5

lower left y : -15

width : 3

height : 15



* Give ctrl S and save this.

* Click polygon icon  and keep it in Centre of feed box. 

* Click substrate editor  (Multicolor Icon) next to EM icon

In New substrate window, (OK)

substrate model opens :

* click yellow color layer (bottom layer)

· Thickness :~ 0.035 mm

* Click above blue layer :- (dielectric layer)

Material : FR_4_core

Thickness : 1.6 mm

* select Conductor layer :- other yellow color small layer

Thickness :- 0.035 mm

* Give ctrl S.

* Go EM → simulation set-up →

frequency plan : Type: Linear → start freq: 2 GHZ

Fstop: 3 GHZ

select All generator frequencies ↗ * Give Simulate → Graph will be opened

* place the marker on the graph

check frequency & S_{11} (dB)

* . click smith icon, smith opened.

place marker on smith line randomly & note impedance value

click * Far field icon → solution set-up → Click nearby value to 2.4 GHZ → plot properties → Far Fields → Antenna Parameters → Observe parameters

* Far Field cuts → enable → display cut & data → evaluate HPBW by locating the Half power points on gain points. (red color).

Exp: 2 **MICROSTRIP LINE**

* Create New schematic window → OK

* library → TLines – Microstrip
  ⇒ click [MSUB] and drag & drop it
  ⇒ click [MLIN], drag & drop it.

* Change library to Simulation S_parameters
  ⇒ click [S-PARAMETERS], drag & drop
  ⇒ click TermG (two No's), drag & drop

* Connect MLIN to TermG blocks

For S-PARAMETERS:-

Start = 2 GHZ
Stop = 3 GHZ
Step = 0.1 GHZ

For MSUB:

H = 1.6 mm
Er = 4.3
Cond = 5e+9
TanD = 0.0025
T = 0.035 mm

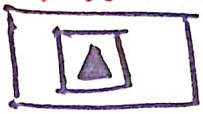
* click Tools → linecalc → startLinecalc

Substrate Parameters:
Er = 4.3
height = 1.6 mm
T = 0.035 mm
Con = 5e9
tanD = 0.0025

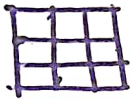
Component parameters:

Freq : 2.4 GHZ

electrical { Zo : 50 ohm
E-Eff : 90 deg

* Click synthesize 

(in physical) * Copy, paste W & L values in MLIN

* click simulate, select & drop  icon

⇒ plot traces & Attributes window opens

⇒ Add { S(1,1) → dB → OK → graph shows
S(2,1) → dB → OK
S(1,2) → dB → OK
S(2,2) → dB → OK

⇒ Place marker on 2.4 GHZ on all graphs

⇒ Observe all dB (magnitudes)

⇒ Add S(1,2) → phase → OK
Add S(2,1) → phase → OK
place marker at 2.4 GHZ
observe phase

Exp ③ :- TEE PLANES

1. Open EMPRO software.
2. New project (select) ↑ → Generic FEM, design in mm
   ⇒ Type Minimum frequency : 8 GHZ
   Maximum frequency : 12 GHZ
   ⇒ Click (OK)
3. Workspace opens
4. Create → geometry → select Box 

Primary waveguide specifications for H-plane
- Width : 80 mm
- Depth : 24 mm
- Height : 10 mm

Note:
give Name to box
↳ PWG

5. To make hollow of box:
   ⇒ select shell and put shell option to both phases of Box → 
   ⇒ shell thickness : 1 mm
6. Assign materials :-
   ⇒ materials → right click → select default library ⇒ choose ☐ Al
   ⇒ drag ☐ Al and drop it to Box
7. To form secondary waveguide :-
   ⇒ click box  → W : 24 mm
   D : 35 mm
   H : 10 mm

Note :
Name : SWG

   ⇒ put shell option to box like we did before.
   ⇒ select secondary waveguide → modify → transform → translate : y : 29.5 mm
8. Create new  box : W – 24mm
   D – 35 mm
   H – 10 mm
9. select newly created box → modify → transform → translate : y = 28.5 mm

10. Select newly created box → (Ctrl) (Control) → Select primary waveguide → Select boolean → Subtract

11. Circuit Components/ports → Click New waveguide port
↑ click

⇒ click Face and select small edge of H-plane

click this for port creation

⇒ go to boundary extensions in ports editor and uncheck ☐ Auto-extend to simulation domain boundaries

⇒ go to properties → waveguide port → 50 ohm def: → Volt source

⇒ go to Impedance Lines:

i) Select face from endpoint 1 and put it to edge bottom of H-plane

click

ii) select face from endpoint 2 and put it to top edge of H-plane

like this, give faces to all other openings.

12. Before FEM simulation → Click icon

→ Change adaptive to Linear and done

13. save file and click simulation

14. Results → sensor type: surface sensor → Domain → right click

→ View default

⇒ Display mode: change Vector field

⇒ click Materials

→ click sequence and click ▷ play button

⇒ go to results window and search s-parameters

S-parameters → Domain – frequency → right click → View default

⇒ Graph shows.

⇒ select 2 & 3 in Matrix selector.

⇒ Marker → point Marker → place mark on any point on shown graph.