Notes:

- 1.) Hole locations are indicated in separate .drl file, included with the gerber package. That file takes precedence over this drawing.
- 2.) Board outline indicated in separate Edge.Cuts file, included with the gerber package. That file takes precedence over this drawing.
- 3.) Vendor to plate holes and exposed copper pads
- 4.) Total board height: 1.6mm +/- 10 %

		K	33.500	mm	→		
		K	23.500	mm	1		
	1			٦		 	
		0					
			700				
61.000 mm	51.500 mm		-0				
	<u></u>		• € (
¥					9		

Layer (Material) Type	Layer (File) Name	Thickness (mm)	Dielectric Constant	Notes
Silkscreen	F.Silk	0.00762		
Soldermask	F.Mask	0.0127	3.8	
Copper	Top.HF	0.035		RF signals are on the top layer
2-2313 (Prepreg)		0.11	4.29	
Copper	In1.LF	0.035		Ref. for 100 Ω RX diff. W: 0.1524mm, C=0.11mm
FR4 (Core)		1.13	3.96	
Copper	In2.PWR	0.035		Ref. for 50Ω TX, Track W: 0.64mm, C=0.15mm
2-2313 (Prepreg)		0.11	4.29	
Copper	B.GND	0.035		Few GPIO traces are here, mostly GND
Soldermask	B.Mask	0.0127	3.8	
Silkscreen	B.Silk	0.00762		

Total: 1530 um

PCB Specifications

1. Material: FR-4 (TG-150)
2. Soldermask color: Black/Blue

3. Silkscreen color: White 4. Surface Finish: ENIG

5. Impedance Control: Yes

Tented with soldermask 6. VIAs:

7. Stackup: See stackup table below for layer specific info

drawn by: Orkhan AmirAslan (azerimaker)

The Things Industries

Sheet:

File: generic_node.kicad_pcb

Title:	Generic	Node	SE

Date: 2020-10-12 Rev: 1.1 KiCad E.D.A. kicad (5.1.8)-1 ld: 1/1