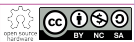


	<project_name>, <pcba_name>		
	Ennnn_ART	REV	X1
LAYER		EXTENSION	
PROJECT_ENGINEER	<prj_eng>	DATE	<art_pri_eng_date>
LAYOUT_ENGINEER	<art_engineer>	DATE	<art_engineer_date>
CHECKED	<art_checker>	DATE	<art_checked_date>

	<project name>, <pcba name>			
	DWG NUM	Ennnn_FAB	REV	X1
	PROJ ENG	<prj_eng>	DATE	<pcb prj eng date>
	DWG ENG	<pcb engineer>	DATE	<pcb engineer date>
	CHECKED	<pcb checker>	DATE	<pcb checked date>
	FILE	Ennnn_NAME_PCBA.kicad_pcb		
	Micro-USB connector 			
		6	7	



Assembly Top

	<project_name>, <pcba_name>		
	Ennnn_ART	REV	X1
LAYER		EXTENSION	
PROJECT_ENGINEER	<prj_eng>	DATE	<art_pri_eng_date>
LAYOUT_ENGINEER	<art_engineer>	DATE	<art_engineer_date>
CHECKED	<art_checker>	DATE	<art_checked_date>

C



<project_name>, <pcba_name>

DWG NUM	Ennnn_FAB	REV	X1	SHEET SIZE	USLegal
PROJ ENG	<prj_eng>	DATE	<pcb_prj_eng_date>	KICAD VER	KiCad E.D.A. 9.0.4
DWG ENG	<pcb_engineer>	DATE	<pcb_engineer_date>	SHEET	2 of 2
CHECKED	<pcb_checker>	DATE	<pcb_checked_date>		
FILE	Ennnn_NAME_PCBA.kicad_pcb				

D

6

7