

22.11.2018

**FILENAME :** kleineplatinever11\_pan

**gerber:** extended metric .3 absolute  
**drill:** excellon metric .3 ascii

**Layer\_stack** file extension

top side .top  
 bottom side .bot

solder mask\_top .smt  
 solder mask\_bottom .smb

paste mask\_top .pmt don't plot !

silk screen\_top .sst

drill\_check .dri

milling program gerber .mil  
 panel .pan

drill program .nc  
 drill report .rpt

tolerances plated drills +0.1 / -0.05 and unplated drills +0.1 unless otherwise specified

attention, please aspire to get solder resist between the (SMD-)Pads.

<b>PCB:</b>	kleineplatinever11_pan			<b>File:</b>	0
<b>Specification</b>					
material:	<b>FR4</b>	layer:	<b>2</b>	thickness:	<b>1.55mm</b>
space/trace :	<b>.2/.2</b>	Cu_thick (final):	<b>35um</b>	smd:	<b>top</b>
surface :	<b>HAL leadfree</b>			via tenting:	<b>no</b>
solder mask :	<b>top+bot</b>	<b>green</b>		silk screen:	<b>top white</b>
electr. Test	<b>yes</b>				
measurements pcb:	<b>80.6 x 50.6 mm<sup>2</sup> +-0.1</b>				
drills/Qty single board:	<b>186</b>	different tools	<b>7</b>		
smallest drill :	<b>0.4</b>	largest drill :	<b>3.8</b>		
milling/scoring panel	<b>161.2 x 111.2 mm<sup>2</sup> +-0.1</b>		<b>2 x 2 =</b>	<b>4 pcs.</b>	