

Data Format

Label of layer in Gatema	
name of pcb .*	
*.top	component side
*.bot	solder side
*.smt	mask component
*.smb	mask solder
*.plt	silk screen component
*.plb	silk screen solder
*.peeltop	Peelable solder-stop top
*.peelbot	Peelable solder-stop bottom
*.pastetop	SMD paste data top
*.pastebot	SMD paste data bot
*.in2, *.in3, *.in4, *.in5	nner layers
	4: TOP-IN2-IN3-BOT
	6: TOP-IN2-IN3-IN4-IN5-BOT
*.pth	PTH – unless stated otherwise – always resulting averages
*.1-2	blind via
*.2-7	buried via
*.mill_pth	PTH mill – unless indicated – diameter always finished size.
*.mill	NPTH mill, with NPT holes
*.score	V-scoring
*.dim	outline
*.txt	description file

Label of layer in Gatema

Make sure that your Gerber files DO NOT use zero size holes (size = 0.00mm or inches) and that your Excellon data will NOT have tools with zero size (size = 0.00mm or inches).