

	<b>Eagle Layerbuild</b>	<b>Id.- Nr.: XX - YY - 000</b> Rev.: B Datum: 18.03.2005
--	-----------------------------	--



**Important Information!**

## LAYERBUILD LIST FOR EAGLE FILES IN PCB-POOL®

<u>Layer:</u>	<u>Eagle Names:</u>
Component side (CS)	top(1) pads(17) vias(18)
Solderside (SS)	bottom(16) pads(17) vias(18)
Soldermask (CS)	tstop(29)
Soldermask (SS)	bstop(30)
Position Print (CS)	tplace(21) tnames(25)
Position Print (SS)	bplace(22) bnames(26)
Contour and additional cut out's	dimension(20)
Drills (pth)	drills(44)
Drills (npth)	holes(45)
Innerlayer 1	Route2(2)
Innerlayer 2	Route3 (3)

The setting for covering vias can be found under „edit – design rules“ and can be adjusted under „mask – limit“

Please note: If additional or different Layers are used, please specify this on your order.

© Beta Layout GmbH Feldstraße 2 65326 Aarbergen Tel. +49 (0)6120 907010 Fax. +49 (0)6120 907014 © hw	File: e_layerlist_eagle.doc ----- Hotline: +353 61 701170 Internet: <a href="http://www.pcb-pool.com">www.pcb-pool.com</a> Info: <a href="mailto:info@beta-layout.com">info@beta-layout.com</a>	<b>Page 1 of 1</b>  DIN EN ISO 9002:1994 Zertifikat Nr. 01 100 004044
---	---	---