

CircuitCAM v7.3

User Guide & Reference Manual

2014

<http://www.circuitcam.com>

Tianjin Decoter Co.,Ltd.

K1-5-102, No.6 Hi-Tech Develop 6th Road
XiQing District, Tainjin, 300384, P.R.China

Table of Contents

Table of Contents	2
Table of Figures	6
1. Introduction	16
Installation and Registration	19
1.1. System Requirements	19
1.2. Installation	19
1.3. Registration and Activation	20
1.3.1. Online Registration	21
1.3.2. Offline Registration	25
1.4. User Settings	30
1.5. About dialog box	31
2. Usage Guide	33
2.1. Import PCB Layout	33
2.1.1. Default Template File (default_insulate.cat)	33
2.1.2. Tutor example	35
2.1.3. Open sample circuit board	37
2.1.4. Import Production Data	38
2.1.5. Import Extended Gerber Data	50
2.1.6. Save Data	54
2.2. Prepare PCB Layout	54
2.2.1. Change Order of Layers	54
2.2.2. Move Layer	55
2.2.3. Rotate Layer	58
2.2.4. Move Board Outline to Separate Layer	61
2.2.5. Place Objects on Separate Layers	63
2.3. Edit SMT Stencils	67
2.3.1. GMC Tutorial	67
2.3.2. Package Library Tutorial	67
2.3.3. Create Package Tutorial	67
2.4. Generating NC Data	67
2.4.1. Create Milling Contour	67
2.4.2. Create Base Material and Fixture Holes	71
2.4.3. Create Rubout Area	71
2.4.4. Insulate Milling Tutorial	72
2.4.5. Customize Insulation and Rubout Job	72
2.4.6. Laser Fill Tutorial	77
2.4.7. Laser Insulate Tutorial	79
2.4.8. SMT Stencils Tutorial	84
2.5. Processing NC Data	84
2.5.1. Export Production Data	84

2.5.2. NC Mode Tutorial	85
2.5.3. Close CircuitCAM	85
2.6. Others	85
2.6.1. Create CircuitCAMTemplate File	85
2.6.2. Using Import Assignments	87
3. Reference Manual	91
3.1. Main User Interface	91
3.1.1. Menu	93
3.1.1.1 File	93
3.1.1.2 Edit	94
3.1.1.3 Select	97
3.1.1.4 Insert	99
3.1.1.5 Modify	101
3.1.1.6 Tool Path	104
3.1.1.7 Library	105
3.1.1.8 Analysis	107
3.1.1.9 Settings	107
3.1.1.10 View	109
3.1.1.11 Help	110
3.1.2. Toolbars	111
3.1.2.1 Standard	112
3.1.2.2 Zoom	113
3.1.2.3 Select	113
3.1.2.4 Insert	114
3.1.2.5 Layer	115
3.1.2.6 Grid / Unit	115
3.1.2.7 Relative Rotations	116
3.1.2.8 Flash Rotations	117
3.1.2.9 Full Screen	117
3.1.2.10 GMC	117
3.1.3. Status Bar	118
3.1.4. View arrangement	119
3.1.4.1 Dock View	119
3.1.4.2 Move View Outside CircuitCAM Window	119
3.1.4.3 Auto-hide View	119
3.1.4.4 Reset Layout	120
3.1.5. Message Log	120
3.1.6. General Settings	122
3.1.6.1 User Related	122
3.1.6.2 Document Related	129
3.1.6.3 Reset to Defaults	133
3.1.7. Customize	133

3.1.7.1 Toolbars	134
3.1.7.2 Commands	135
3.1.7.3 Keyboard	135
3.1.7.4 Options	136
3.1.8. Online Help	137
3.1.9. Preview / World View	138
3.1.10. Handling of Esc Key	139
3.2. CAM and CAT Document Handling	140
3.2.1. Open	141
3.2.2. Synchronize	142
3.2.3. New	143
3.2.4. Import	144
3.2.5. Saving your data	149
3.2.6. Export	150
3.2.7. Export Layout Instance	151
3.2.8. Example of Aperture List after Gerber File Export	153
3.2.9. Print	154
3.2.10. Read Out	159
3.2.11. Statistics	162
3.2.12. Batch and Background Processing	165
3.3. Graphic View	166
3.3.1. Zooming and Panning	168
3.3.2. Manipulation of Graphic Objects	172
3.3.3. Boolean Operations	184
3.3.4. Size Modification	191
3.3.5. Properties	194
3.3.6. Selection of Graphic Objects	199
3.3.7. Context Menu "Right Mouse Button"	201
3.3.8. Editing operations	202
3.3.9. Inserting Operations	206
3.3.10. Manipulation Operations	212
3.3.11. Text and Barcodes	213
3.3.12. Text	213
3.3.13. Barcode	215
3.4. List Views	216
3.4.1. Layers	218
4. Tools View	225
4.0.0.1 Tools List Properties	229
4.0.0.2 Tools Properties	233
4.0.0.3 Tool Edit Mode	237
4.0.1. Jobs and Quick Access	238
4.0.1.1 SMT Stencil Frame Generator	241

4.0.1.2 Export Job	244
4.0.1.3 Export Task	247
4.0.1.4 Export Transforms	252
4.0.1.5 Scripts	258
4.0.1.6 Script Tasks	265
4.0.1.7 Configurable Geometry Engine	268
4.0.2. Layouts	285
4.0.3. Libraries	288
4.0.4. Packages and Footprints	291
4.0.5. Create Package from Selection	292
4.0.6. Package Recognition	294
4.0.7. Footprints	295
4.0.8. Geometric Manipulation Centre	297
4.0.9. Predefined Shapes	300
4.0.10. Shape Modification	302
4.0.11. Size Modification	304
4.0.12. Glue Dots	305
4.0.13. Compare	307
4.0.14. Library	308
4.0.15. Data Formats	310
Aperture Files and Templates (translation files)	311
Aperture Template Editor	317
Barco-DPF	320
CAM	324
CAT	325
DXF	327
EESOF	327
Excellon	327
G-code	329
Gerber X	330
Gerber	334
HP-GL	335
ISO 10303-210	337
Laser Mill Drill (LMDLaser, Milling, Drilling)	337
ODB++	342
Sieb & Meyer	342
4.1. Tool Paths	344
4.1.1. Laser Scanner	344
4.1.2. Tool Compensation	344
4.1.3. Contour Routing	346
4.1.4. Insulate Milling	349
4.1.5. Striping & Stripping	355

4.1.6. NC Mode	355
4.2. Operations and Checks	365
4.2.1. Base Material with Fixture Hole	365
4.2.2. Copper Pouring	367
4.2.3. DataMatrix	369
4.2.4. Selecting Operations	370
4.2.5. PCB Design Rule Check (DRC)	371
4.2.6. Stencil Design Rule Check	373
5. Hotkeys	375
6. Epilogue	378
Glossary	378
Index	380

Table of Figures

Figure 1. The current installed variant of CircuitCAM is displayed in the title bar.	18
Figure 2. CircuitCAM Activation Wizard	20
Figure 3. CircuitCAM Activation Wizard, Online activation	22
Figure 4. CircuitCAM Activation Wizard, to setup connection	23
Figure 5. CircuitCAM Activation Wizard, connection to server and data transferring	24
Figure 6. CircuitCAM Activation wizard, completed activation with activated CircuitCAM version	25
Figure 7. CircuitCAM Activation Wizard, Offline registration, 1st step	26
Figure 8. CircuitCAM Activation Wizard, Offline registration, to save HTML file	27
Figure 9. CircuitCAM Activation Wizard, creation of HTML file	28
Figure 10. CircuitCAM registration page in browser window	29
Figure 11. CircuitCAM Activation Wizard, Offline activation, 2nd step	29
Figure 12. CircuitCAM Activation Wizard, to specify location of licence file	30
Figure 13. General Settings, User Information	31
Figure 14. PCB top and bottom faces	36
Figure 15. PCB component side on the left side (default color: red) and PCB solder side on the right (default color: green)	36
Figure 16. Select files for import dialogue window	38
Figure 17. Import dialogue window: 1-list of imported files; 2-Graphic, Text and Apertures panels; 3-Attribute panel	39
Figure 18. Select translation file for aperture list	40
Figure 19. Text panel with translation results	40
Figure 20. Gerber file content, circuit board component side	41
Figure 21. Component side is defined as TopLayer	42
Figure 22. Gerber file content, circuit board solder side (mirror presentation)	43
Figure 23. Solder side is defined as BottomLayer	44
Figure 24. Contents of tool list in Apertures panel	45
Figure 25. Selected translation file and tool list with parameters in Apertures panel	46

Figure 26. Graphic view content of selected Excellon file	47
Figure 27. Select DrillPlated layer	47
Figure 28. Overlapping holes in Graphic view	48
Figure 29. Graphic view with correct format parameters	49
Figure 30. CircuitCAM displays imported product data	50
Figure 31. Import dialogue window with Extended Gerber files list	51
Figure 32. Import dialogue window with graphic view of drill file content	52
Figure 33. Import dialogue window with prepared Extended Gerber data for import	53
Figure 34. CircuitCAM window with imported Extended Gerber production data	54
Figure 35. Objects of active DrillPlated layer are not visible in Graphic view	54
Figure 36. Dragging of DrillPlated layer over TopLayer	55
Figure 37. Components of DrillPlated layer are displayed in Graphic view (light blue color)	55
Figure 38. Layer / Template is changed to ToturTools.TXT for changing aperture tool size.	56
Figure 39. Circuit board graphic view	57
Figure 40. Moving layer	57
Figure 41. Layer / Template is changed to ToturTools.TXT for changing aperture tool size.	59
Figure 42. Circuit board graphic view	59
Figure 43. Select hole (set anchor point)	60
Figure 44. Active layer in Layers view and Select Layer button in Select tool bar	60
Figure 45. Rotate layer 270° around anchor point	61
Figure 46. Rotated layer	61
Figure 47. All layer, excluding TopLayer, are reset to Not selectable mode	61
Figure 48. Graphic view with selected circuit board outline	62
Figure 49. Move to Layer context sensitive menu	62
Figure 50. TopLayer the only visible layer in Layers list and component side in graphic view	63
Figure 51. Select text objects on TopLayer	64
Figure 52. Text objects on TextTop layer are presented in Graphic view	64
Figure 53. Bottom layer components in Graphic view	65
Figure 54. Text objects on TextBottom layer are presented in Graphic view	65
Figure 55. Circuit board presentation in Graphic view	66
Figure 56. Fastening holes on DrillUnplated layer (dark blue color in Graphic view)	66
Figure 57. Selected circuit board outline in Graphic view and highlighted BoardOutline layer in Layers panel	67
Figure 58. Contour routing dialogue window	68
Figure 59. Milling contour in Graphic view	69
Figure 60. Contour selected in Graphic view	69
Figure 61. Setting anchor point	70
Figure 62. Positions of anchor point for breakout tabs	70
Figure 63. Contour with breakout tabs	70

Figure 64. Sequence number of selected path in Status bar	71
Figure 65. Defined rubout rectangle area in Graphic view	72
Figure 66. Jobs view, Run selected job	73
Figure 67. Run job dialogue window	73
Figure 68. Tools dialogue window	74
Figure 69. Layers dialogue window	75
Figure 70. Insulation and rubout job result in Graphic view	76
Figure 71. Zoomed window in Graphic view	76
Figure 72. TopInsulate_Primary layer insulation channels of true width and filled (yellow color)	76
Figure 73. Laser Fill run job dialogue	77
Figure 74. Running the Laser Fill script with default parameters	78
Figure 75. Running the Laser Fill script; with Y-Parallel Lines	79
Figure 76. Running the Laser Fill script; with Concentric Lines	79
Figure 77. Laser Insulate run job dialogue	80
Figure 78. Laser Insulate Script with default values	81
Figure 79. Laser Insulate Script with inner isolation flag on	82
Figure 80. Laser Insulation Script with a defined layer fill	83
Figure 81. Laser Insulate Script with a defined layer rubout	84
Figure 82. CircuitCAM message window with destination file path	85
Figure 83. General settings window, Directories panel	86
Figure 84. General Settings window, Import Assignment panel	87
Figure 85. Assignment for aperture list	88
Figure 86. Import assignment DrillPlated	90
Figure 87. CircuitCAM frame window	91
Figure 88. CircuitCAM Menu bar	93
Figure 89. Selection size limit dialogue window	98
Figure 90. Layout Properties dialogue window	106
Figure 91. CircuitCAM Activation Wizard window	111
Figure 92. Standard toolbar	112
Figure 93. Zoom toolbar	113
Figure 94. Select toolbar	113
Figure 95. Insert toolbar	114
Figure 96. Layer toolbar	115
Figure 97. Unit / Grid toolbar	115
Figure 98. Relative Rotations toolbar	116
Figure 99. Flash Rotations toolbar	117
Figure 100. Full screen toolbar	117
Figure 101. GMC toolbar	117
Figure 102. Status bar	118
Figure 103. Guiding arrows and shaded designation area	119
Figure 104. Auto Hide item in view menu and pushpin button	120

Figure 105. Tabs of hidden views	120
Figure 106. Message log view, import log	121
Figure 107. Message Log view, export log	121
Figure 108. General Settings dialogue window	122
Figure 109. Measurement unit drop down list	122
Figure 110. Display panel	123
Figure 111. Directories panel	124
Figure 112. User Information panel	125
Figure 113. Default Configuration dialogue window	126
Figure 114. General Import panel	127
Figure 115. Excellon Import panel	128
Figure 116. Predefined pitches panel	128
Figure 117. Recognition panel	128
Figure 118. Diverse panel	129
Figure 119. Accuracy panel	129
Figure 120. Export Strategy panel	130
Figure 121. Colors panel	131
Figure 122. Import Assignment panel	132
Figure 123. Reset to Defaults panel	133
Figure 124. Customize dialogue window, Toolbars panel	134
Figure 125. New Toolbar dialogue window	134
Figure 126. Rename Toolbar dialogue window	134
Figure 127. Customize dialogue window, Commands panel	135
Figure 128. Customize dialogue window, Keyboard panel	136
Figure 129. Customize dialogue window, Options panel	137
Figure 130. Help -> Help Topics menu	138
Figure 131. World view of Graphic view	138
Figure 132. Preview of Tool	139
Figure 133. Preview of Library content (BGA package)	139
Figure 134. Selected layers in Layers view, highlighted path in Graphic view	140
Figure 135. Open file dialogue	141
Figure 136. Select files for import... dialogue window	144
Figure 137. Import dialogue window	145
Figure 138. Apertures panel	146
Figure 139. Gerber Options panel	147
Figure 140. Attribute panel	148
Figure 141. Save As dialogue window	149
Figure 142. If the CAM document contains any unsaved changes, an asterisk "*" appears next to the document name	150
Figure 143. Export menu content	151
Figure 144. Layout instances export format dialogue window	152
Figure 145. CircuitCAM Graphic view	155

Figure 146. Settings dialogue window	156
Figure 147. Print preview of graphic on multiple pages	157
Figure 148. Printer Calibration dialogue window	158
Figure 149. Printer Calibration dialogue window, Use Calibration check box	159
Figure 150. Status bar with Cartesian and polar coordinates	160
Figure 151. Pitch of two pads	161
Figure 152. Selected Object message window for area	161
Figure 153. Statistics window	162
Figure 154. Statistics dialogue for BGA-default library	163
Figure 155. Statistics entries grouped by Object type	164
Figure 156. Statistics entries grouped by Object type and Type	165
Figure 157. When a rectangle is rotated, it is converted to a polygon.	167
Figure 158. Unit/Grid toolbar	168
Figure 159. The process of Zoom Window	169
Figure 160. The process of Zoom Selected	169
Figure 161. Zoom Overview	170
Figure 162. Previous View; after clicking Insert the last view is restored	170
Figure 163. Right click panning in Graphic View.	171
Figure 164. Navigating by selecting a place from the Preview window	171
Figure 165. Circle markers	173
Figure 166. Rectangle markers	173
Figure 167. Path with arc markers	173
Figure 168. Flash marker	173
Figure 169. Combine operation initial data and result	175
Figure 170. Close operation initial (open path) data and result (closed path)	175
Figure 171. Path to Polygon initial data and result	176
Figure 172. Combine to Polygon initial data and result	176
Figure 173. Convert to Polygon initial data (rectangle, open path,closed path) and result (polygons)	177
Figure 174. Convert to Closed Outline Path initial data (circle, rectangle, polygon) and result (outlines)	177
Figure 175. Denest Polygon CutOut initial data (selected one hole) and result (denested all holes)	177
Figure 176. Denest Polygon CutOut Hole initial data (selected one hole) and result (denested only selected hole)	178
Figure 177. Convert to Circle initial data (selected rectangles) and result (circle)	178
Figure 178. Draw to Flash initial data (selected rectangles and overlapping circles) and result (generated flashes)	179
Figure 179. New Flash dialogue window	179
Figure 180. Convert to One Flash initial data (two object selected) and result (only one object selected)	180
Figure 181. New Layout dialogue window	180
Figure 182. Step and repeat dialogue window	181

Figure 183. Create Bounding Shape dialogue window	182
Figure 184. Create Outline of Shape dialogue window	183
Figure 185. Perfect shape tolerance dialogue window	184
Figure 186. The plain example used to demonstrate Boolean operations;object A is a rectangle, object B is a circle, object C is an open path	184
Figure 187. After the three objects are selected, the union operation is executed, the result is a new polygon D	185
Figure 188. After the two objects A and B are selected, A is the primary selected object (operator);the difference operation is executed, the result is a new polygon D	186
Figure 189. After the two objects A and B are selected, B is the primary selected object (operator);the difference operation is executed, the result is a new polygon D	186
Figure 190. After the three objects A, B, and C are selected, A is the primary selected object (operator);the difference operation is executed, the result is TWO new polygons D and E	186
Figure 191. After the two objects A and B are selected, the intersection operation is executed;the result is a new polygon D	187
Figure 192. After the three objects A, B, and C are selected, A is the primary selected object (operator);the intersection operation is executed, the result is a new polygon D	187
Figure 193. After the three objects A, B, and C are selected, B is the primary selected object (operator);the intersection operation is executed, the result is a new polygon D	188
Figure 194. After the two objects A and B are selected, the XOR operation is executed;the result is TWO new polygons D and E	188
Figure 195. After the three objects A, B, and C are selected, A is the primary selected object (operator);the XOR operation is executed, the result is a set of new polygons	189
Figure 196. After the three objects A, B, and C are selected, B is the primary selected object (operator);the XOR operation is executed, the result is a set of new polygons	189
Figure 197. After the three objects are selected, the Paths on Area operation is executed, the result is trimming of the path C as shown	190
Figure 198. After the two objects A and C are selected, the Paths on Area operation is executed, the result is trimming of the path C as shown	190
Figure 199. After the three objects are selected, the Paths off Area operation is executed, the result is trimming of the path C as shown	191
Figure 200. After the two objects B and C are selected, the Paths off Area operation is executed, the result is trimming of the path C as shown	191
Figure 201. Shape modification dialogue window, Scaling panel	192
Figure 202. Shape modification dialogue window, Polygon Operations panel	193
Figure 203. Rounding results: positive value (left) and negative value (right)	194
Figure 204. Properties panel, Document information	195
Figure 205. Properties panel, Closed path	195
Figure 206. Properties panel, Circle	196
Figure 207. Properties panel, Rectangle	197
Figure 208. Properties panel, Polygon	197
Figure 209. Properties panel, Flash	198
Figure 210. Properties panel, selected 43 items	198
Figure 211. Right mouse button menu (context sensitive menu)	201

Figure 212. Rotate dialogue window	203
Figure 213. Rotate function effects	204
Figure 214. Edit -> Set Flash Rotation Value menu content	205
Figure 215. Set Anchor Point panel	205
Figure 216. Set Cursor dialogue window	206
Figure 217. Status bar with flash number	206
Figure 218. Filled Circle (1- centre point, 2- point on edge)	207
Figure 219. Circle Diameter dialogue window	208
Figure 220. polygonCutOut with rectangular outer and circular inner shape	210
Figure 221. Types of cutouts identified in the status bar	210
Figure 222. Selected circle	210
Figure 223. Cutout menu item	211
Figure 224. Selected a boundary polygon for polygonCutOut	211
Figure 225. Result of Cutout operation	211
Figure 226. Insert text dialogue window	214
Figure 227. Barcode Sample	215
Figure 228. Installing a new font	216
Figure 229. D14 is the selected item and the Properties View shows its characteristics. D16 is the active item of the selected graphic object, shown in red bold color.	217
Figure 230. Paste Layer Dialogue	218
Figure 231. Layers view	218
Figure 232. Graphic view: a) True Width and Filled, b) True Width and Outline, c) Line Only and Filled, d) Line Only and Outline	220
Figure 233. Colors dialogue window	220
Figure 234. Layers view context sensitive menu	221
Figure 235. Layers view toolbar	222
Figure 236. Layer Properties dialogue window	223
Figure 237. Tool view	226
Figure 238. Combo box, standard toolbar view	226
Figure 239. Apertures/Tools list in Tools view	227
Figure 240. Tools view context sensitive menu	228
Figure 241. Tools view toolbar	228
Figure 242. Gerber (Excellon / IEC61182 / Sieb & Mayer) Format Properties dialogue window	230
Figure 243. Postscript Format Properties dialogue	230
Figure 244. HP-GL (Diverse / LMD) Format Properties dialogue window	231
Figure 245. DXF Format Properties dialogue window	232
Figure 246. Barco Format Properties dialogue window	232
Figure 247. G-Code Format Properties dialogue window	233
Figure 248. Aperture/Tool Properties dialogue window, Flash Attributes panel	234
Figure 249. Aperture/Tool Properties dialogue window, Transform panel	235
Figure 250. Aperture/Tool Properties dialogue window, Draw panel	236

Figure 251. Aperture/Tool Properties dialogue window, Diverse panel	237
Figure 252. Tool properties window.	238
Figure 253. Tool Edit mode.	238
Figure 254. Ending the item edit mode by selecting Save/Discard changes.	238
Figure 255. Jobs view	239
Figure 256. Jobs view context sensitive menus	240
Figure 257. Jobs view toolbar	241
Figure 258. Stencil Frame Generator dialogue window	242
Figure 259. Frame with circular glue holes	244
Figure 260. Export Job Properties dialogue window	245
Figure 261. Job processing window	246
Figure 262. Job finished window	246
Figure 263. Job finished with errors window	246
Figure 264. The Message Log showing the job execution details	247
Figure 265. Excellon Export Task dialogue same as (G-code, DXF, Barco)	248
Figure 266. GerberX Export Task dialogue	249
Figure 267. LMD Export Task dialogue	250
Figure 268. HP-GL Export Task dialogue	251
Figure 269. Swing-in and -out paths (blue lines)	251
Figure 270. Post Script Export Task dialogue	252
Figure 271. The Export Transform window.	253
Figure 272. Alignment is set to centre on X and Y axis	254
Figure 273. Alignment is set to centre on X and Y axis, and scale is set to "2" - Double ..	255
Figure 274. Alignment is set to centre on X axis, and bottom on Y axis, and scale is set to "1/2" Half, and 90 degree rotation	255
Figure 275. Alignment is set to centre on X and Y axis, and the inverse flag is on	255
Figure 276. Alignment is set to right on X axis, and centre on Y axis	256
Figure 277. Alignment is set to right on X axis, and centre on Y axis, and rotation is 270 degrees	256
Figure 278. left figure shows the auto origin off and right figure shows the auto origin on	257
Figure 279. left figure shows the auto scale off and right figure shows the auto scale on	258
Figure 280. Jobs view, listing available Scripts and Quick Access view with included Scripts	258
Figure 281. Script Properties dialogue window	259
Figure 282. Properties view of selected Script	260
Figure 283. Assigning image to the corresponding job	262
Figure 284. Script added with image and comment is written below the job name	262
Figure 285. Run Script dialogue window	263
Figure 286. New Derived Script Dialogue	265
Figure 287. New Script command dialogue window	266
Figure 288. Properties view, Script command of type Oversize properties	267
Figure 289. Combine closed path initial data (left) and result (right)	272

Figure 290. Create needle from concentric initial data (left) and result (right)	272
Figure 291. Find lead-in initial data (left) and result (right)	274
Figure 292. Get sub path by length initial data (left) and result (right)	274
Figure 293. Midline initial data (left) and result (right)	275
Figure 294. Layouts and Properties view	285
Figure 295. Layouts view context sensitive menu	286
Figure 296. Layouts view toolbar	287
Figure 297. Libraries and Properties view	288
Figure 298. Libraries view context sensitive menu	290
Figure 299. Libraries view toolbar	290
Figure 300. Select libraries dialogue window	292
Figure 301. New package dialogue window	292
Figure 302. Predefined pitches panel	293
Figure 303. Select layers dialogue window	294
Figure 304. a) Initial data for recognition, b) recognized data	295
Figure 305. Select dialogue window to select libraries	296
Figure 306. GMC main overview	297
Figure 307. GMC, Predefined Shapes panel	301
Figure 308. GMC, Shape Modification panel	302
Figure 309. Chop Up settings	303
Figure 310. Chop Up example	303
Figure 311. GMC, Size Modification panel	305
Figure 312. GMC, Glue Dots panel	306
Figure 313. GMC, Compare panel	307
Figure 314. GMC, Lib panel	309
Figure 315. Main ATE window	317
Figure 316. ATE Shape&Size toolbar	318
Figure 317. ATE Edit toolbar	319
Figure 318. ATE, Template Parameters' Setup dialogue window	319
Figure 319. Properties of a CAM file, referring to the underlying CAT file	325
Figure 320. File -> New menu item and list of templates	326
Figure 321. Save As dialogue window to save CAT files	326
Figure 322. Example of drawn PCB.	330
Figure 323. G-Code example.	330
Figure 324. Laser Scanner dialogue window	344
Figure 325. Tool Compensation dialogue window	345
Figure 326. Selected outline (light yellow), tool compensation result (on outer side (black line), Take care of path width On)	346
Figure 327. Contour Routing dialogue window	346
Figure 328. Layer dialogue window	347
Figure 329. Tools dialogue window	348
Figure 330. Contour routing result	348

Figure 331. Predefined insulate job	350
Figure 332. a) Old concentric paths with triangles; b) new concentric paths with needles (right side)	351
Figure 333. List of Scripts	351
Figure 334. Insulation tracks (4 tools, defined rubout area)	352
Figure 335. Insulation tracks (3 tools, defined rubout area)	352
Figure 336. Insulation tracks (3 tools, defined rubout area, additional channels around primary channel)	353
Figure 337. Insulation tracks (2 tools, defined rubout area)	353
Figure 338. Insulation tracks (2 tools, defined rubout area, additional channels around primary channel)	354
Figure 339. Insulation tracks (1 tool, additional channels around primary channel)	354
Figure 340. NC/Tool Path menu	356
Figure 341. Connecting to an NC sever; local machine (left), or remote machine (right) ..	357
Figure 342. Setting the active layer for NC mode	357
Figure 343. NC Mode layout	358
Figure 344. NC Mode layers	359
Figure 345. NC Control View	360
Figure 346. The connections status while connected to an I++NC server	361
Figure 347. NC Connection error	361
Figure 348. NC control standard view, displaying all tool paths in the "Prepare" layer ..	362
Figure 349. NC Machine progress window	363
Figure 350. NC control, displaying the tool paths distributed among layers, according to their progress	363
Figure 351. NC control displaying the current coordinates for machine head position ..	364
Figure 352. Machine head controls in NC mode	364
Figure 353. Changing the machine head position using "Direct move"	365
Figure 354. Base Material dialogue	366
Figure 355. Drilling Tools List	367
Figure 356. Example of the final Base Material	367
Figure 357. Insert -> Copper Pouring menu item	368
Figure 358. Copper pouring dialogue window	368
Figure 359. Flash-finger and flash-circle with paths and rectangular copper area	368
Figure 360. Result of copper pouring	368
Figure 361. Insert Data Matrix dialogue window	369
Figure 362. Example of Data Matrix code, encoding text "Always a cut ahead"	370
Figure 363. Selection by Pitch dialogue window	370
Figure 364. Design Rule Check dialogue window	372
Figure 365. Graphic view with errors (white areas)	372
Figure 366. Stencil Design Rule Check dialogue window	373

1. Introduction

Usage Guide and Reference Handbook for *CircuitCAM* version 7.3

CircuitCAM is a software application for computer aided manufacturing (**CAM**) of printed circuit boards (**PCB**). The main functionalities are:

- Importing of **CAM** files generated by **PCB CAD** systems.
- Verification and design checks of the imported data.
- Modifying the layout data for manufacturing purposes.
- Generating tool paths for NC machines, including drilling, milling, laser cutting and structuring.
- Exporting of **CAM** files.

CircuitCAM is available in five different variants, optimized for different purposes:

- *CircuitCAM VIEWER* for viewing **CAM** files generated by *CircuitCAM*.
- *CircuitCAM BASIC* with all typical **CAM** functionalities for milling and drilling including insulate.
- *CircuitCAM STANDARD* with full flexible **CAM** functionalities, including **Laser** structuring and cutting.
- *CircuitCAM PRO* with all the functionalities of *CircuitCAM STANDARD* plus the ability to modify and analyse big sets of data, including extensive library functionality.
- *CircuitCAM Laser Plus* with all the functionalities of *CircuitCAM PRO* in addition to **Laser Fill, Laser Insulate, Striping and Stripping**.

Category	Feature	Viewer	Basic	Standard	Pro	Laser Plus
CAM files						
	Open	x	x	x	x	x
	Save	-	x	x	x	x
Import						
	Gerber	x	x	x	x	x
	GerberX	x	x	x	x	x
	Excellon/Sieb&Meyer	x	x	x	x	x
	LMD	x	x	x	x	x
	HP-GL	x	x	x	x	x
	DXF	x	x	x	x	x
	Barco DPF	-	-	x	x	x
	ODB++	-	-	x	x	x
Export						
	Gerber	-	x	x	x	x
	Excellon	-	x	x	x	x
	LMD	-	x	x	x	x
	HP-GL	-	x	x	x	x
	G-Code	-	x	x	x	x
	GerberX	-	-	x	x	x
	HP-GL with Pen Width	-	-	x	x	x
	DXF	-	-	x	x	x
	Barco DPF	-	-	x	x	x

Category	Feature	Viewer	Basic	Standard	Pro	Laser Plus
	Postscript	-	-	x	x	x
	List of Sub-Layout Instances	-	-	-	x	x
Special						
	DataMatrix	-	-	x	x	x
	PCB DRC	-	-	x	x	x
	Stencil DRC	-	-	-	x	x
Editing						
	Editing of Primitives and Flashes	-	x	x	x	x
	Multi level Undo/Redo	-	x	x	x	x
	CSG (Set Operations)	-	-	x	x	x
	Polygon with Cutouts	-	-	x	x	x
	Convert to Flash & Comparison	-	-	x	x	x
	Draw to Flash	-	-	x	x	x
	Denest Flash	-	-	x	x	x
	Create Outline	-	-	x	x	x
	Step&Repeat	-	-	x	x	x
	Create Bounding Shape	-	-	x	x	x
	Post Script	-	-	x	x	x
	Shape Manipulations	-	-	x	x	x
	Sub-Layout instances	-	-	-	x	x
	Geometric Manipulation Centre (GMC)	-	-	-	x	x
Tool Path						
	Insulation Milling	-	x	x	x	x
	Contour Routing	-	x	x	x	x
	Base Material & Fixture	-	x	x	x	x
	Tool compensation	-	x	x	x	x
	Swap & Sequence	-	x	x	x	x
	Sorting / path optimization	-	x	x	x	x
	NC Mode	-	x	x	x	x
	Configurable Toolpath Engine (Universal / Laser / Milling / Drilling)	-	-	x	x	x
	Laser Fill	-	-	x	x	x
	Laser Insulate	-	-	x	x	x
	Scanner Fields	-	-	x	x	x
	SMT Stencil calculation	-	-	-	x	x
	SMT Stencil Frame Generator	-	-	-	x	x
	Striping & Stripping	-	-	-	-	x
Library						
	Create Package	-	-	-	x	x
	Create Footprint	-	-	-	x	x

Category	Feature	Viewer	Basic	Standard	Pro	Laser Plus
	Package Recognition	-	-	-	X	X
	GMC Exchange	-	-	-	X	X
	Apply Footprints	-	-	-	X	X
	Import Library	-	-	-	X	X

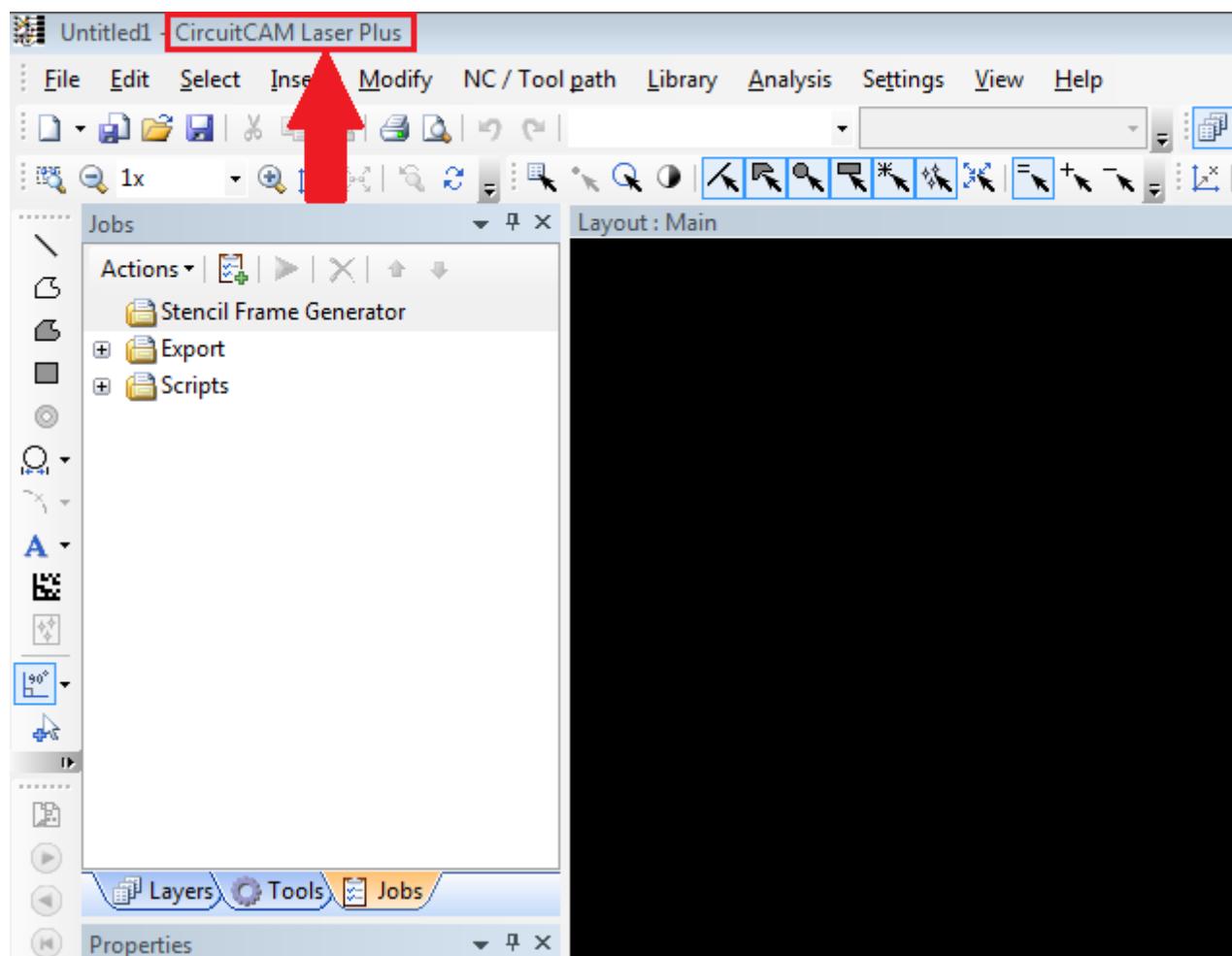


Figure 1. The current installed variant of *CircuitCAM* is displayed in the title bar.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

Installation and Registration

CircuitCAM installation provides instructions on how to prepare and install *CircuitCAM*.

1.1. System Requirements

First of all you have to ensure, that your machine meets the minimum system requirements to avoid problems during installation, or in your future work with *CircuitCAM*.

Minimum system requirements for installing and using *CircuitCAM* are:

Operating Systems	<ul style="list-style-type: none"> ■ Microsoft™ Windows® XP ■ Microsoft™ Windows® Vista ■ Microsoft™ Windows® 7 ■ Microsoft™ Windows® 8
Memory	512 MB of RAM (1 GB recommended)
Display resolution	1024 x 768 with True Color
Hard disk	Installation requires 50 MB of free disk space

1.2. Installation

Follow this routine to start your installation:

- Navigate to the installer file **setup.exe** on your machine.
- Double click on the file icon to begin the Setup program.
- Follow the instructions in the setup program.



Note: Use **Cancel** button to close the dialogue window and cancel the installation.

Installation Folder

The installation folder is the folder where *CircuitCAM* application is installed. By default *CircuitCAM* is installed under **C:\Program Files\CircuitCAM7 or C:\Program Files (x86)\CircuitCAM7 (if you are using an x64 operating system)**. The following sub-folders will be created in the *CircuitCAM* sub-folder:

- **Ape_Templates** contains translation files, which are used to import aperture and tool lists.
- **Example data** contains *CircuitCAM* input and output data files. This is the default directory in which all newly created data are stored.
- **Libraries** contains package libraries, which are used for replacing pad shapes.
- **New_Templates** contains template files for *CircuitCAM*. To view the list of available templates, select **File -> New** menu item.
- **Translations** contains language localization files (for different languages user interface).

You can modify those directories in the [General Settings](#) window.



Note: Stored settings directories cannot be changed.

Running CircuitCAM

To start *CircuitCAM* application, press on the *CircuitCAM* program group in the Windows® Start menu.



Note: You can start *CircuitCAM* via the shortcut on your computer's desktop, or you can navigate to the installation folder and double click on the *CircuitCAM* icon.

1.3. Registration and Activation

After *CircuitCAM* has started, the **CircuitCAM Activation Wizard** pops up on the screen:

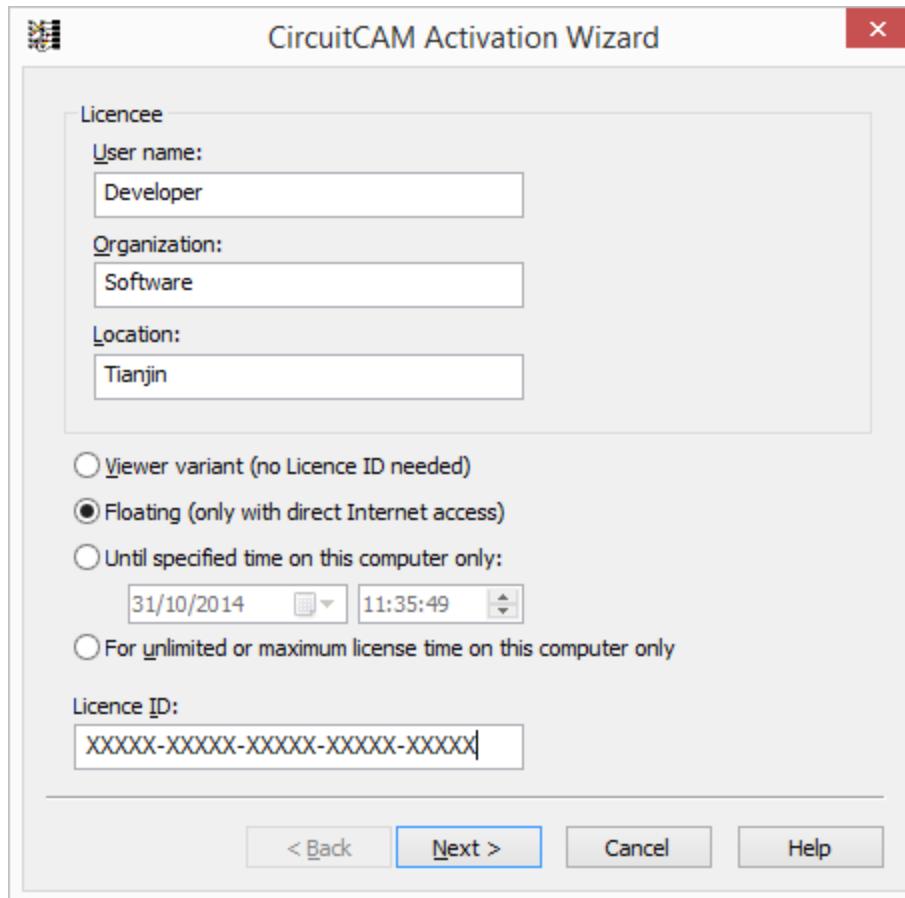


Figure 2. *CircuitCAM Activation Wizard*

If you have a licence ID, you have to enter it in the bottom field. Otherwise, you can only use the *Viewer* variant and leave the "Licence ID" field empty.

Starting with version *CircuitCAM 7.3* it is possible to use the licence in different modes: either floating or for limited time or for unlimited time on one computer. All *CircuitCAM* licences are single computer licences, which means that they can be used on one computer only. However as long as a *CircuitCAM* licence is covered under a maintenance contract the licence can be used on more than one computer, but only on one computer at a time. Once the maintenance period is expired the licence can only be used on one computer.

Activation of a *CircuitCAM* licence requires an internet connection. When *CircuitCAM* is able to directly access the internet, the user can go for the so-called Online registration, that also is required for the floating mode. Otherwise user can do the activation through a web browser on another computer using the so-called Offline registration.

- The floating mode is the default one and allows to use the licence dynamically on different computers, e.g.: the licence can be used in the morning on a desktop computer

and in the afternoon on a mobile computer. The user does not need to say in advance how long he wants to use the licence on one computer. The floating mode can only be used with permanent internet access, because communication with the CircuitCAM licence server is needed. After closing CircuitCAM on one computer, it may take up to 15 minutes till the licence can be used on another computer. If internet becomes unavailable for more than 10 minutes, CircuitCAM stops working and the user can only save the current state in a CAM file.

- If the licence is needed on one computer for a limited time without permanent internet access (e.g.: during travel), user can request the licence for this computer for a specified time, which may last just hours or a few days. This is enable by the mode "Until specified time..." which requires the specification of an end time when the licence is no longer needed on this computer. Once this mode is chosen, the time can be extended, but not shortened. During the occupied time the licence cannot be used on another computer. The time is entered and displayed in the local time zone which is set up by the Windows system. Note: the time that is shown in the "About" dialog box is provided in Coordinated Universal Time (UTC).
- Once the maintenance mode is expired, the user can choose the last option "For unlimited or maximum licence time...". A further usage of the licence on another computer is then no longer possible. Therefore it is recommended to use this option only as last resource.

For the registration follow this sequence:

- Click on the **User name** text field and enter your full name.
- Click on the **Organization** text field and enter your organization name.
- Click on the **Location** text field and enter the address of your organization.
- Select the licence mode: Viewer, Floating, Limited time, Unlimited/maximum time.
- Click on the **Licence ID** field and enter the identifier key of your *CircuitCAM* licence (not for Viewer).
- Click on the **Next** button to open the next window of the activation wizard.

1.3.1. Online Registration

The default registration method is **Online registration**. To select this option, your computer must be connected to the Internet. *CircuitCAM* will connect to the licence server and will register your licence automatically. This is the default and simplest way.

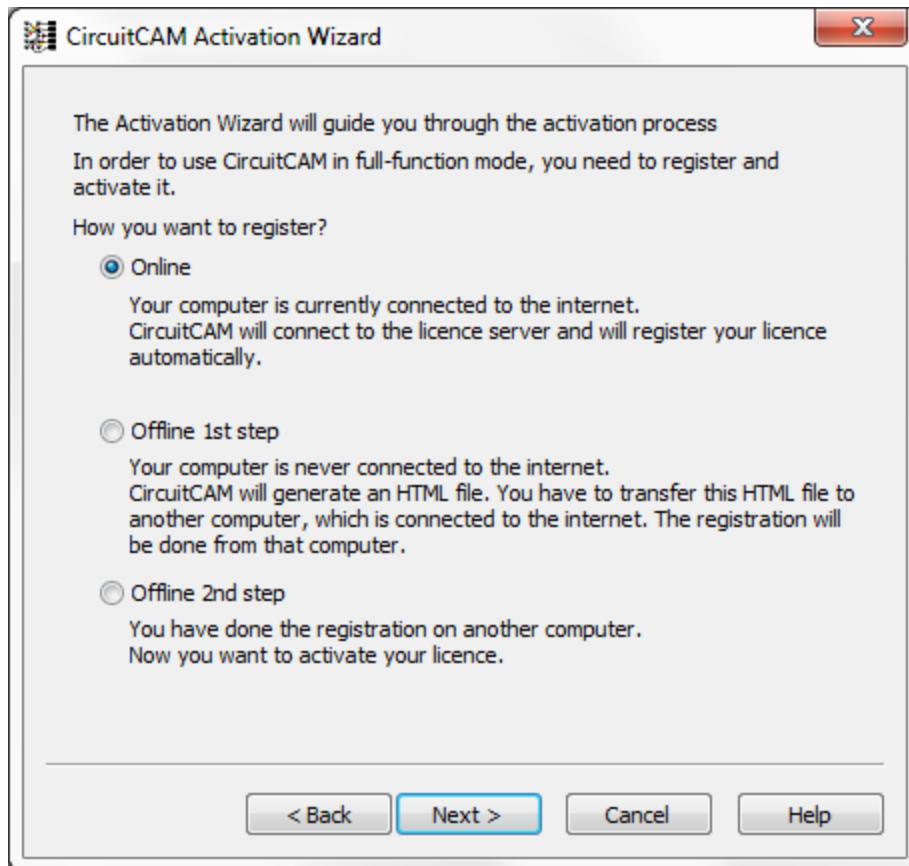


Figure 3. *CircuitCAM Activation Wizard, Online activation*

- Click on the **Online** selection button (see [Figure above](#)).
- Click on the **Next** button to continue the activation. The new activation wizard window appears on the screen (see [Figure below](#)).

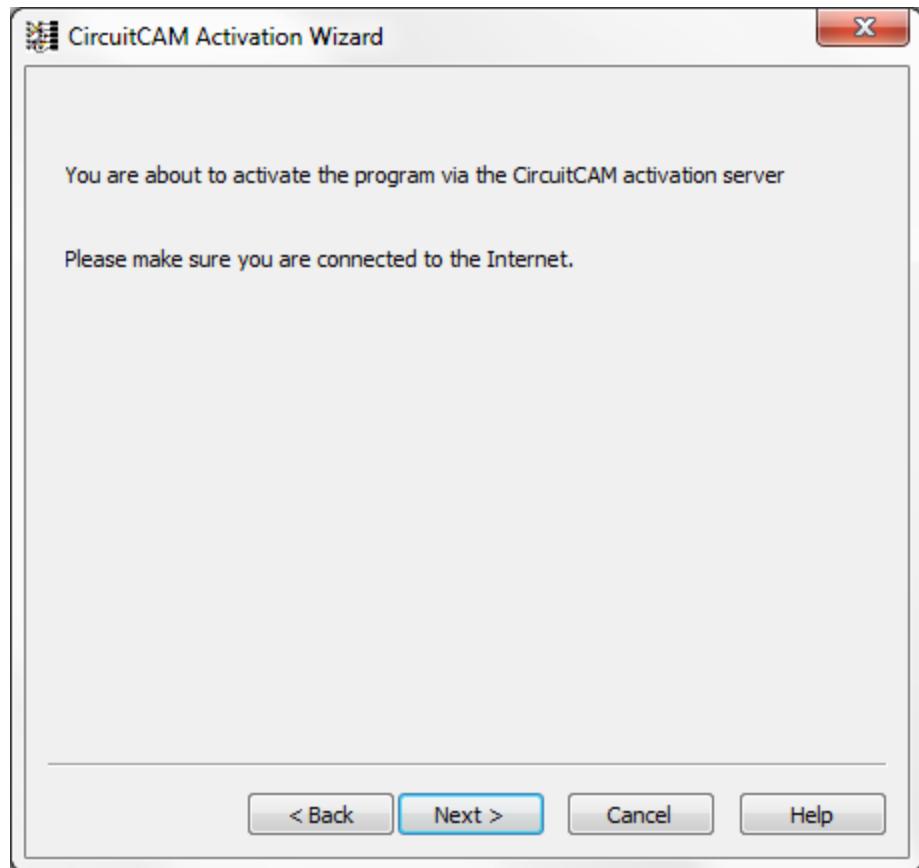


Figure 4. *CircuitCAM* Activation Wizard, to setup connection



Note: Make sure you are connected to the Internet.

- Click on the **Next** button to continue the activation. *CircuitCAM* sets up a connection to the licence server.

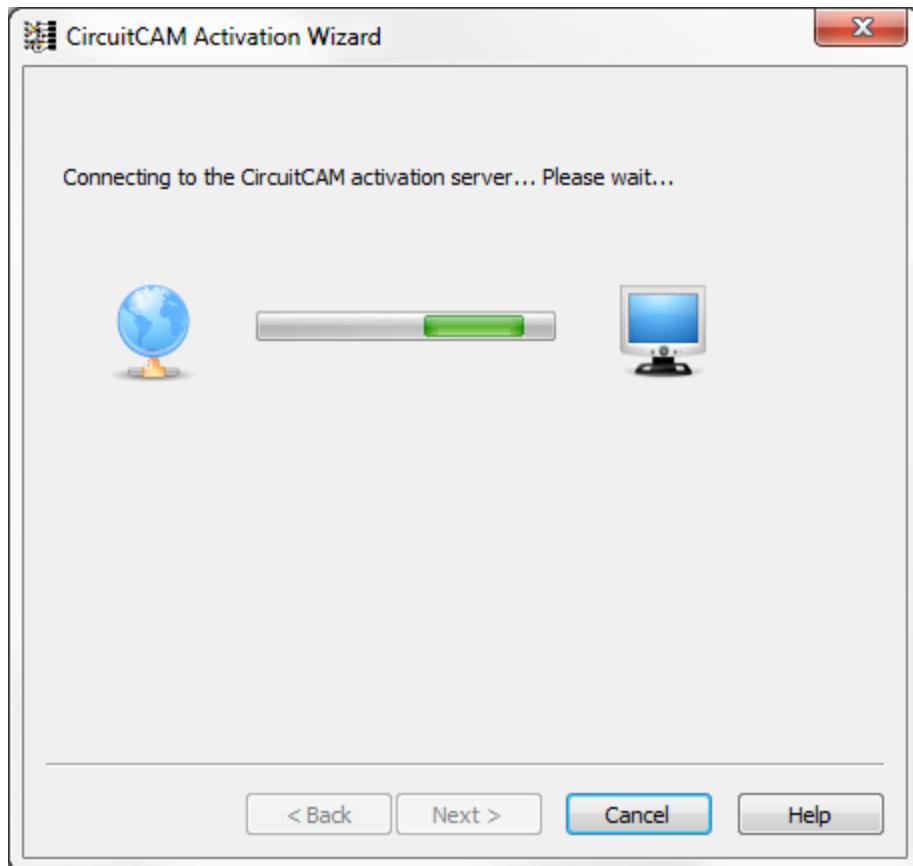


Figure 5. *CircuitCAM* Activation Wizard, connection to server and data transferring



Note: Do not interrupt data transferring.

The activated *CircuitCAM* version is identified and displayed in the wizard window (see [Figure below](#)).

After *CircuitCAM* was successfully registered, you can terminate the activation window by clicking the **Finish** button.

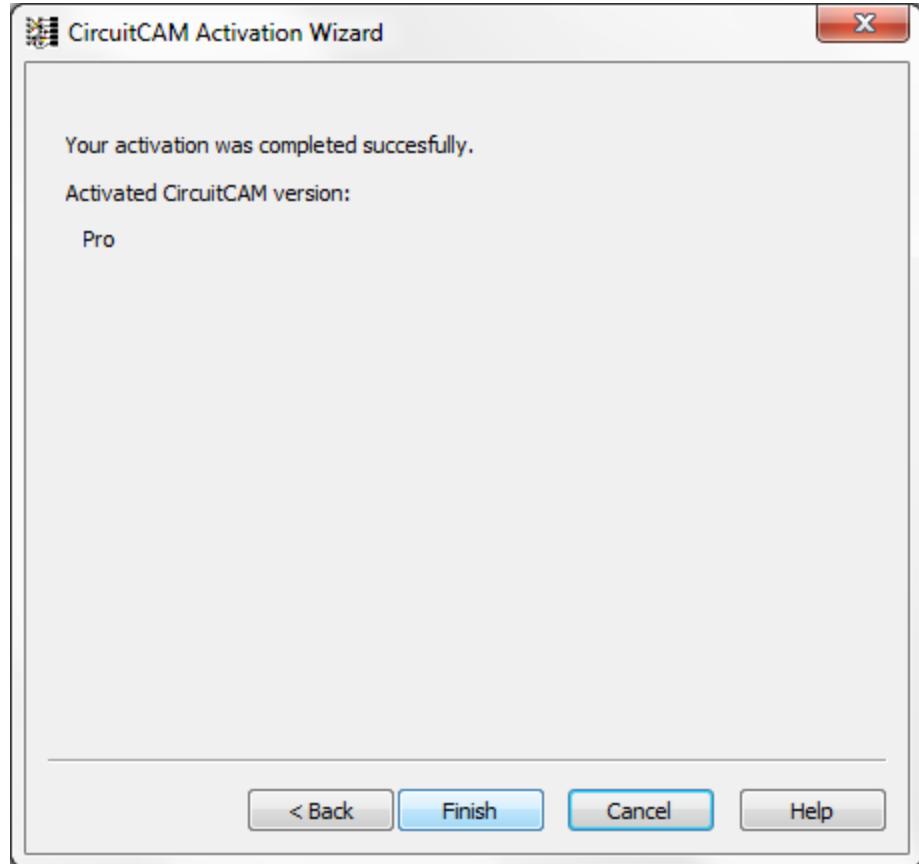


Figure 6. *CircuitCAM* Activation wizard, completed activation with activated *CircuitCAM* version

1.3.2. Offline Registration



Note: If your computer is not connected to the Internet, you can use offline activation method.

For the offline activation you will need a licence file **licence.lic**, this file should be obtained from *DCT* by offline registration.

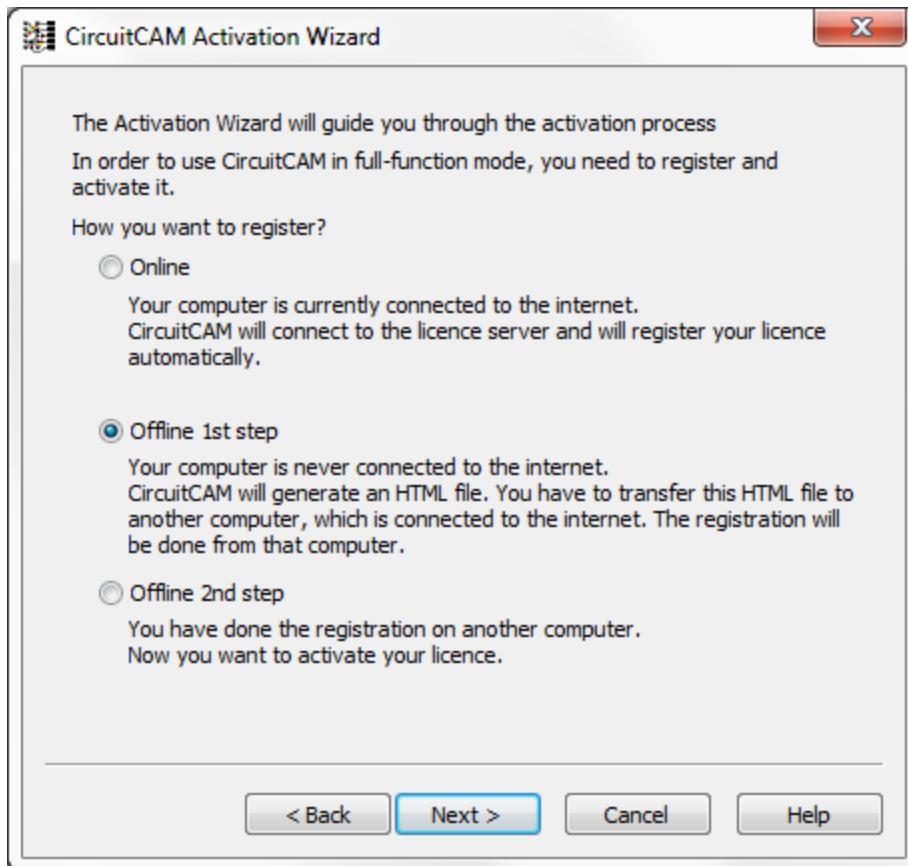


Figure 7. *CircuitCAM* Activation Wizard, Offline registration, 1st step

- Click on the **Offline 1st step** selection button.
- Click on the **Next** button.

CircuitCAM will generate an **HTML** file. You have to transfer this file to another computer, which is connected to the Internet. This registration will be done from that computer.

The new **CircuitCAM Activation Wizard** window appears (see [Figure below](#)) to select **HTML** file location:

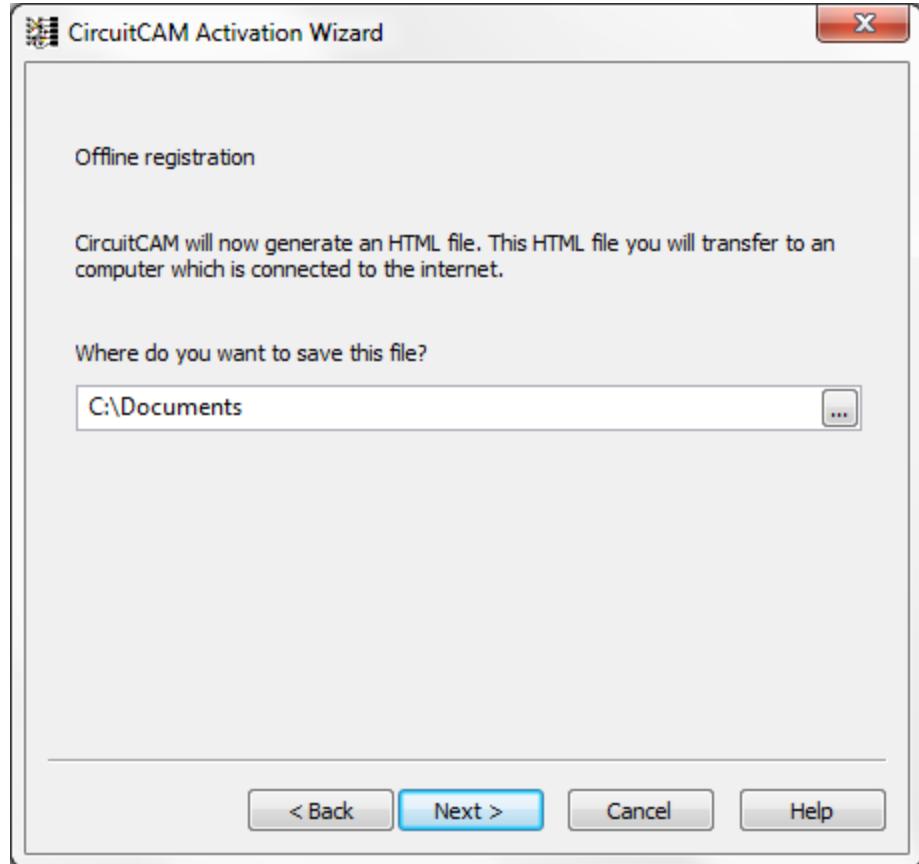


Figure 8. *CircuitCAM* Activation Wizard, Offline registration, to save HTML file

- Click on the lookup button next to the text field and select the location of the **HTML** file on your hard drive. After the location is determined, the path is displayed in the text field.
- Click on the **Next** button. A new window of the activation wizard appears on the screen, which displays a path to the created **HTML** file (see [Figure below](#)):

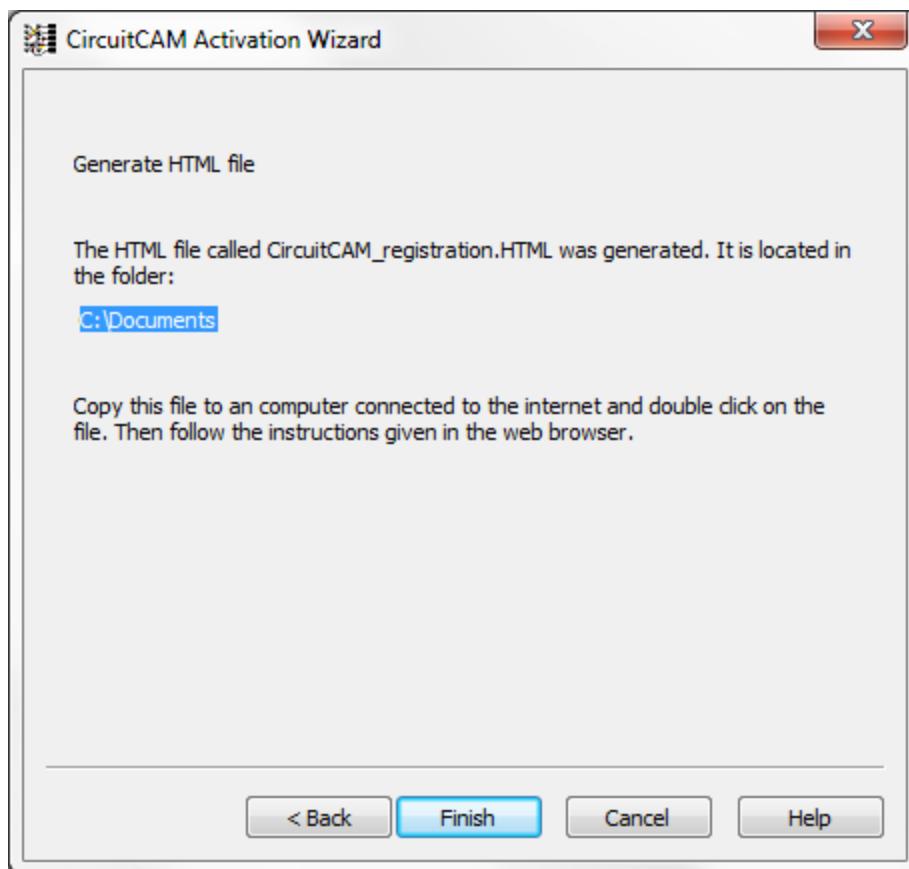


Figure 9. *CircuitCAM* Activation Wizard, creation of HTML file

- Click on the **Finish** button.
- Copy the recently created **HTML** file and paste it to a computer, which has an active Internet connection.
- Double click on the file to open it in the default browser (see [Figure below](#)).

The screenshot shows a web browser window titled 'CircuitCAM online Registr...'. The address bar shows 'C:\Documents\CircuitCAM_registration.HTML'. The page content is titled 'CircuitCAM registration'. It has two sections: 'User data' and 'License data'. Under 'User data', there are four input fields: 'Name' (Your Name), 'Organization' (Your Organization), 'Location' (Your Organization Address), and 'Operating system' (6.1 7601 Service Pack 1). Under 'License data', there are two input fields: 'License ID' (xxxxx-xxxxx-xxxxx-xxxxx-xxxxx) and 'Product version' (7.0.0 build 2001). At the bottom, there is a button labeled 'Request license file' with the URL '(https://services.circuitcam.eu)' next to it.

Figure 10. *CircuitCAM* registration page in browser window

- Fill in your information as required.
- Press on the **Request licence file** button to send registration data to the licence server and request for a licence file.
- Save the activation file in a desired directory. You have done the registration on this computer.
- Copy the file to the machine with the *CircuitCAM* software installed on.
- Open the **Help** menu and select the **Activate Product** menu item. The **CircuitCAM Activation Wizard** page appears on the screen.
- Click on **Next** button to go to the offline activation page.
- Click on the **Offline 2nd step** selection button to activate your licence (see [Figure below](#)).

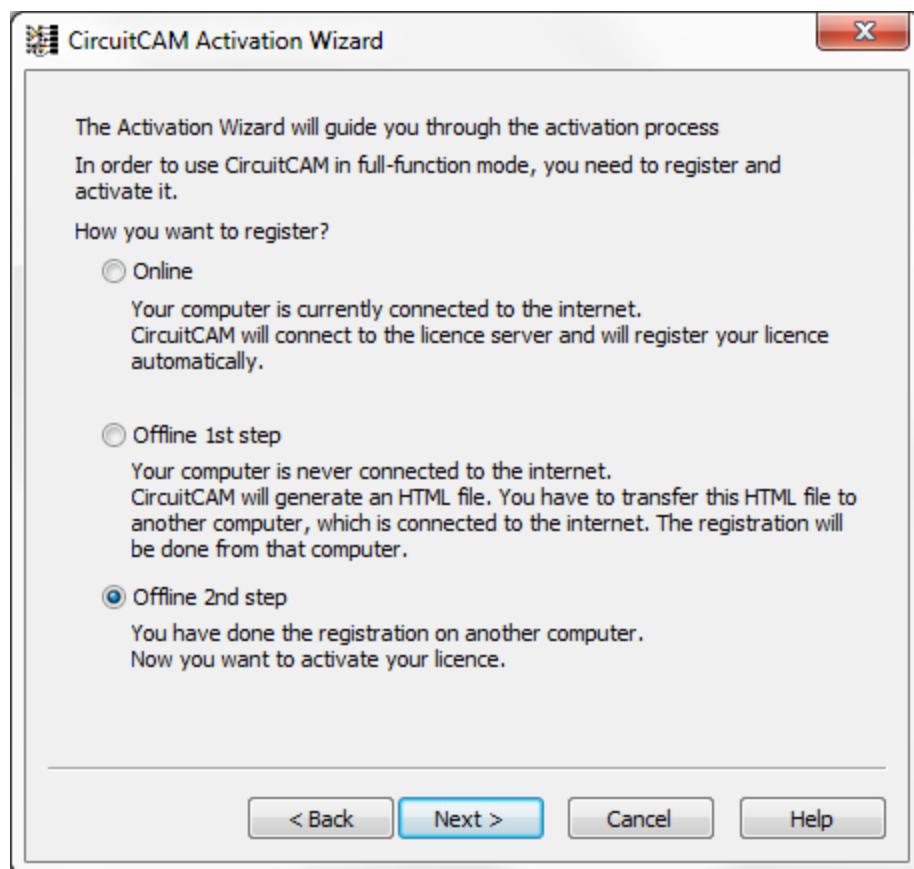


Figure 11. *CircuitCAM* Activation Wizard, Offline activation, 2nd step

- Click on the **Next** button to continue licence activation. A new wizard window appears (see [Figure below](#)).

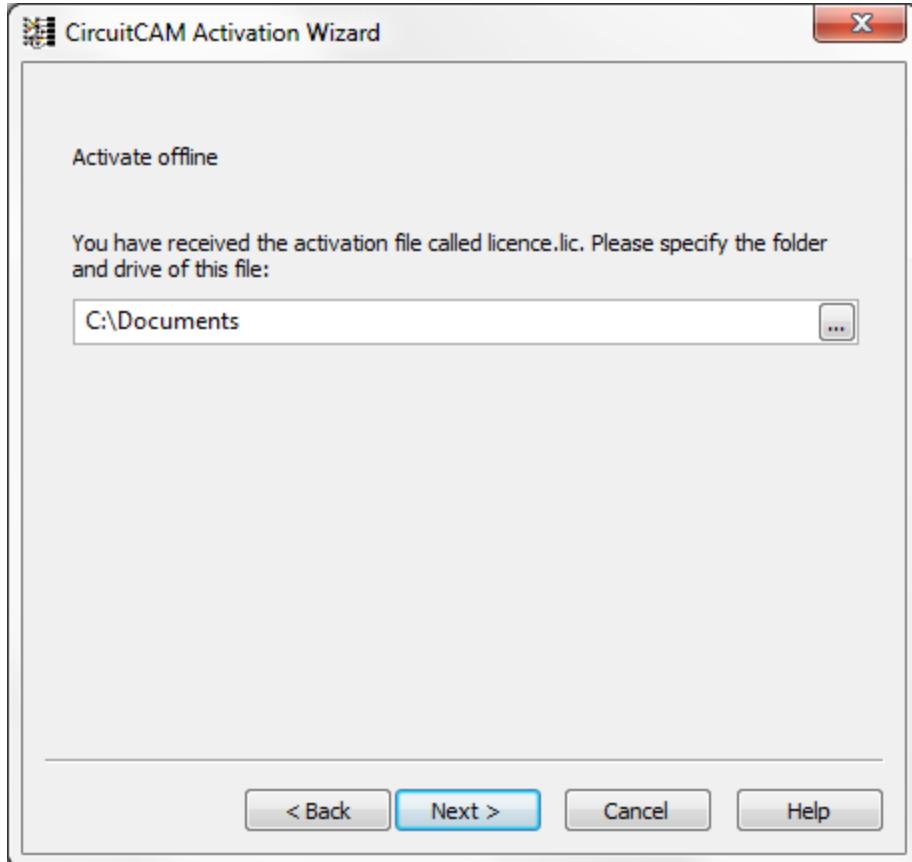


Figure 12. *CircuitCAM* Activation Wizard, to specify location of licence file

- Click on the lookup button next to the text field.
- Select the directory, where the **licence.lic** file is located.
- Click on the **Next** button.

The registration file is copied to the program directory and activates the software version. The activated *CircuitCAM* version is identified in the wizard window.

1.4. User Settings

Users have the possibility to change the *CircuitCAM* user data and the location of directories for the translation and template files. To do this, navigate to the menu; **Settings ---> General Settings**. The **General Settings** window appears on the screen. Information, about the user, organization and address, which was entered during registration process, can be edited in the **User Information** page. Select the **User Information** branch in the tree on the left to activate the particular page (see [Figure below](#)).

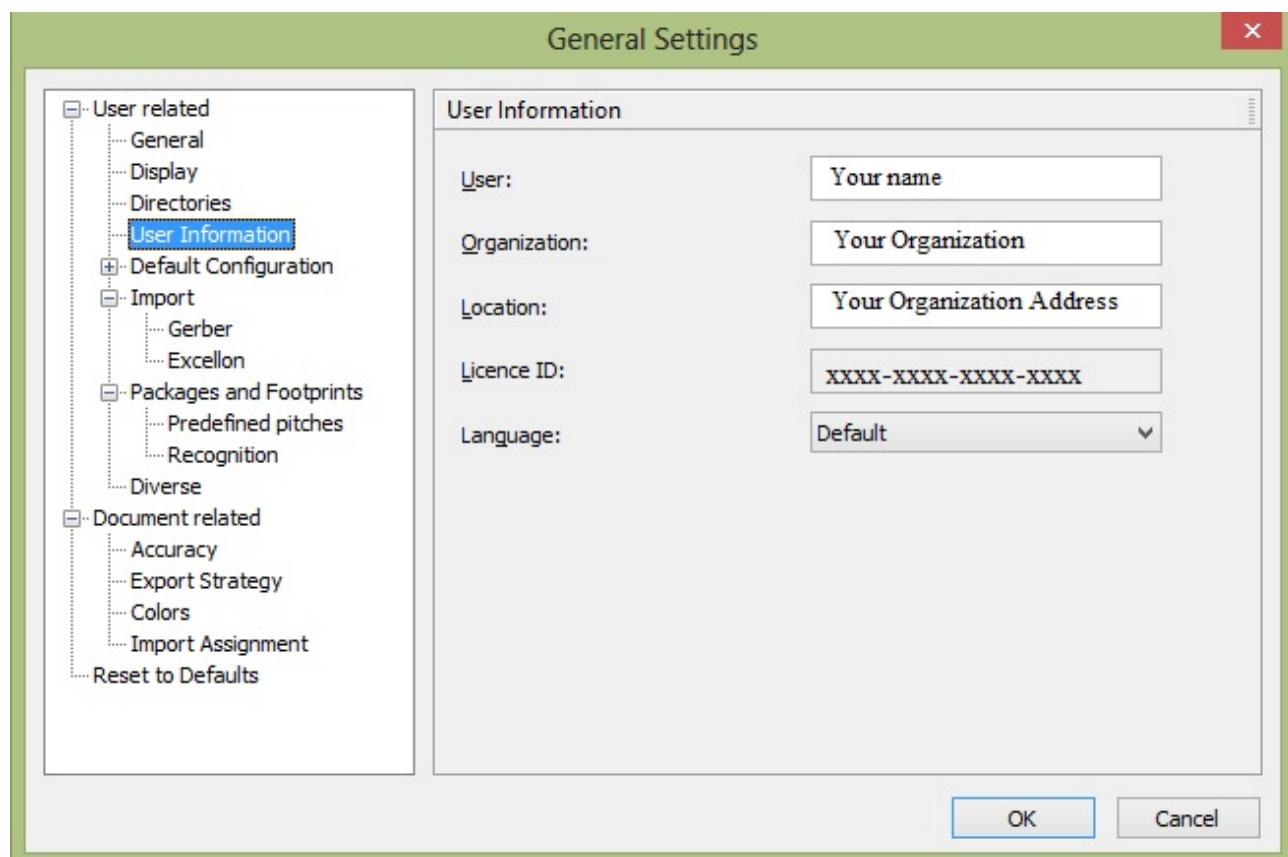


Figure 13. General Settings, User Information

You can change your personal data:

- Click on the **User** field to edit the existing data.
- Click on the **Organization** field to edit the existing data.
- Click on the **Location** field to edit the existing data.
- Click on the arrow next to the **Language** field to see a list of available languages. Select an applicable item in the drop down list.
- Click on the **OK** button to confirm your changes.



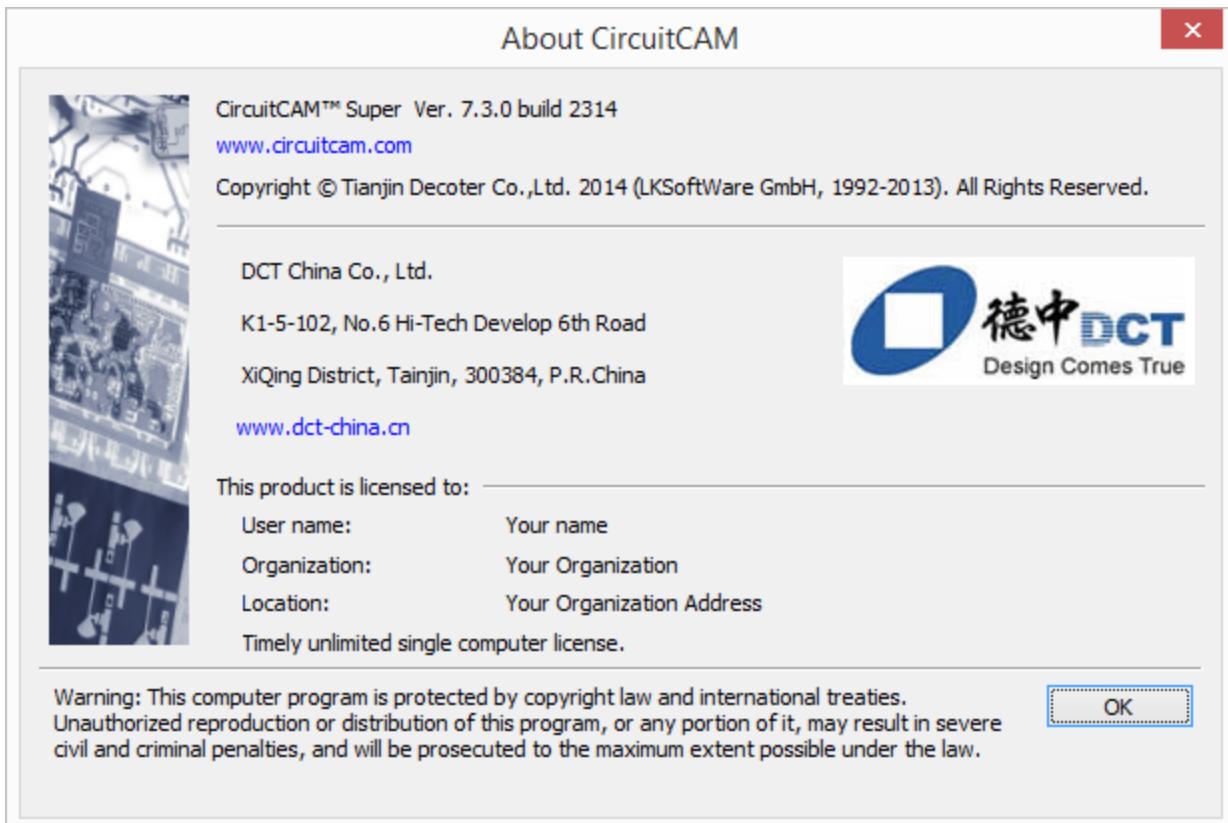
Note: The licence ID field is read only and can not be edited.

1.5. About dialog box

Essential info about the current licence is displayed in "About" dialog box(from main menu "Help"->"About CircuitCAM...")



Note: the licence information indicates on whether the licence is timely limited or unlimited.



Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

2. Usage Guide

The purpose of this manual is to introduce the user how to use *CircuitCAM* for Windows™. It explains how to import CAM files generated by **PCB CAD** systems, verify and check the design of the imported data, modify the layout data for manufacturing purposes, export **CAM** files.

In addition, the user can find more information about; what are the main user interface elements and be guided through the basic features of *CircuitCAM* via exercises.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

2.1. Import PCB Layout

2.1.1. Default Template File (**default_insulate.cat**)

With the installation of *CircuitCAM*, the default template file **default_insulate.cat** is installed with the content. This contents of this file are explained on this page.

For more information how to create a template file see [Create CircuitCAM template file](#).

Predefined Layers

When you start *CircuitCAM* the specified set of layers included in the template file "**default_insulate.cat**" will be presented in the **Layers** view:

Layer name	Use
Fiducial	Little target registration marks
TopLayer	Layout component side
BottomLayer	Layout solder side
DrillPlated	Holes (through plated)
DrillUnplated	Holes (without plating)
BoardOutline	Circuit board outline
TextTop	Text for component side
TextBottom	Text for solder side
CuttingOutside	Contour milling outside
CuttingInside	Contour milling inside
SolderMaskTop	Solder resist mask component side
SolderMaskBottom	Solder resist mask solder side
SolderPasteTop	Solder paste component side
SolderPasteBottom	Solder paste solder side
SilkScreenTop	Component printing on component side
SilkScreenBottom	Component printing on solder side
RuboutTop	Rubout area on component side
RuboutBottom	Rubout area on solder side
RuboutAllLayer	Rubout area on all layers
FrameTop	Frame on component side
FrameBottom	Frame on solder side

FrameAllLayer	Frame on all layers
InsulateTop_Primary	Primary insulation channels on the component side
InsulateTop_Small	Small insulation channels on the component side
InsulateTop_Big	Big insulation channels on the component side
InsulateTop_Bigger	Bigger insulation channels on the component side
InsulateBottom_Primary	Primary insulation channels on the solder side
InsulateBottom_Small	Small insulation channels on the solder side
InsulateBottom_Big	Big insulation channels on the solder side
InsulateBottom_Bigger	Bigger insulation channels on the solder side
TopInsulate	Tool path for insulation milling on the component side
BottomInsulate	Tool path for insulation milling on the solder side
Layer2	Layout Multilayer 2
Layer3	Layout Multilayer 3
Layer4	Layout Multilayer 4
Layer5	Layout Multilayer 5

According to **IEC 1182-1** and **IPC D-350** the physical layer on the primary side, which is the one in default viewing direction, is numbered as layer 1. For *CircuitCAM* the *TopLayer* is the primary physical layer and this is displayed in normal way. The *BottomLayer* is the layer with the highest physical layer number and it is displayed in mirrored way. So for machining the *BottomLayer* from the bottom side the tool-paths must be mirrored.



Note: *CircuitCAM* provides predefined jobs for the above layers, so keep to these names of layers.

Predefined Tools

LMD contains the list of tools, needed for milling and drilling operations.

Gerber contains the default Gerber list of tools.

HP-GL contains the default HP-GL list of tools

Predefined Jobs

Insulation jobs with up to 4 tools:

- Insulate_Top_Primary,
- Insulate_Top_Big_Primary,
- Insulate_Top_Primary_Small,
- Insulate_Top_Bigger_Big_Primary,
- Insulate_Top_Big_Primary_Small,
- Insulate_Top_Bigger_Big_Primary_Small,
- Insulate_Bottom_Primary,
- Insulate_Bottom_Big_Primary,
- Insulate_Bottom_Bigger_Big_Primary,
- Insulate_Bottom_Big_Primary_Small,
- Insulate_Bottom_Bigger_Big_Primary_Small.

Insulation jobs with up to 4 tools for defined rubout area:

- Insulate_Top_Rubout_Primary,
- Insulate_Top_Rubout_Big_Primary,
- Insulate_Top_Rubout_Primary_Small,
- Insulate_Top_Rubout_Bigger_Big_Primary,
- Insulate_Top_Rubout_Big_Primary_Small,
- Insulate_Top_Rubout_Bigger_Big_Primary_Small,
- Insulate_Bottom_Rubout_Primary,
- Insulate_Bottom_Rubout_Big_Primary,
- Insulate_Bottom_Rubout_Bigger_Big_Primary,
- Insulate_Bottom_Rubout_Big_Primary_Small,
- Insulate_Bottom_Rubout_Bigger_Big_Primary_Small.

Predefined Libraries

The **BGA-default** library contains all typical BGA packages, needed for *Package recognition*.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

2.1.2. Tutor example

In the following sections of this manual, you will find an explanation of basic *CircuitCAM* functions, working through examples.

"Tutor" sample circuit board

The circuit board, which will be used in exercises: double-sided PCB with conventional and SMD components, where SMD components are located on the solder side of the circuit board (PCB bottom face).

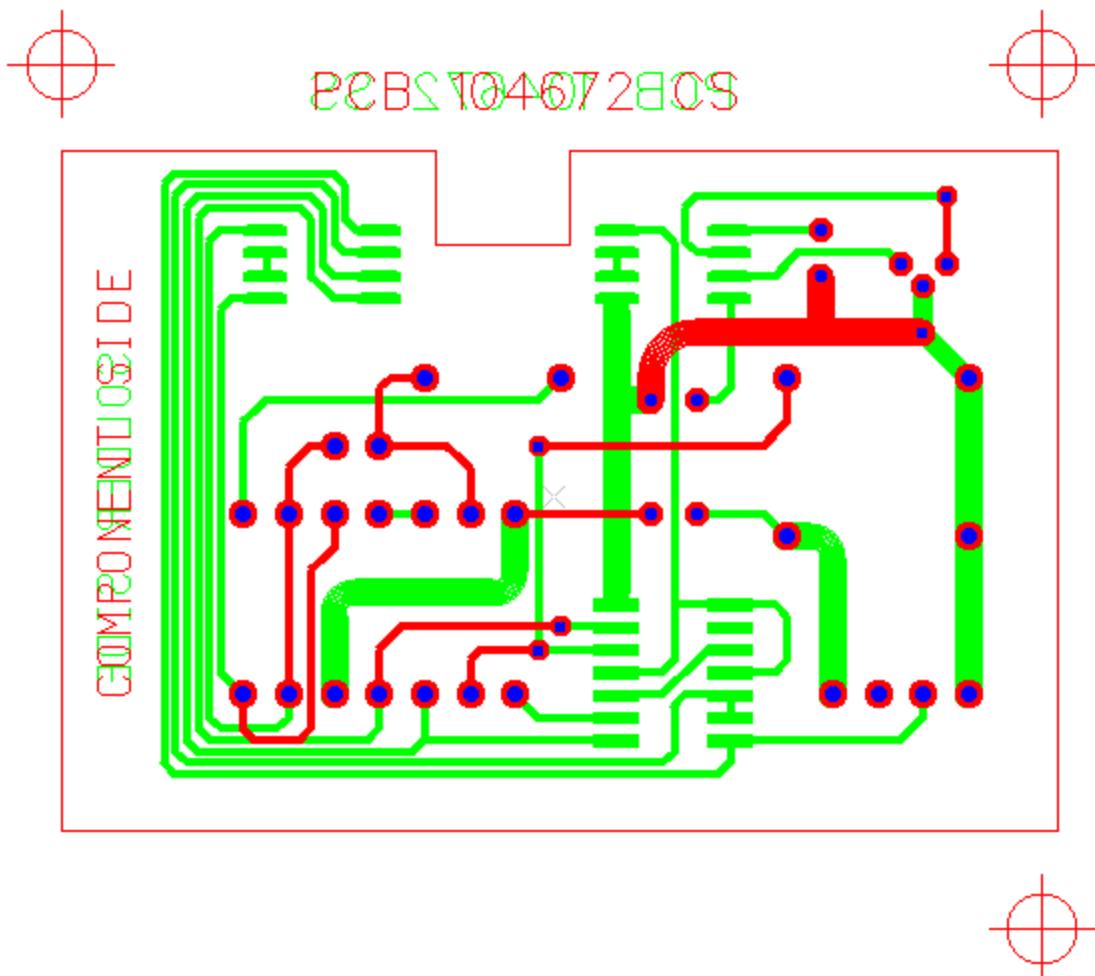


Figure 14. PCB top and bottom faces

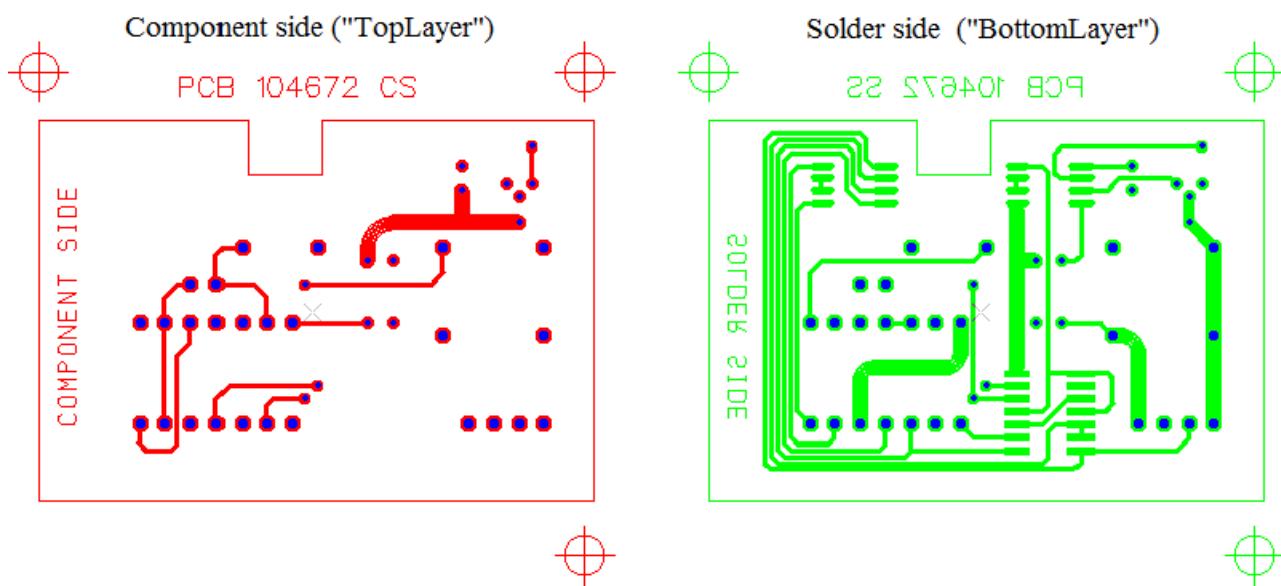


Figure 15. PCB component side on the left side (default color: red) and PCB solder side on the right (default color: green)

Example files

The example files for the exercises you will find in the *CircuitCAM* installation directory (i.e. "**Program Files (x86)\CircuitCAM**"). The following files should appear:

..\Ape_Templates\ folder	
TutorApe.TXT	Translation file for the aperture lists
TutorTools.TXT	Translation file for the tool lists
..\ExampleData\Prototyping\ folder	
Tutor.cam	CAM data, CircuitCAM format. It is the result of import.
Tutor.nct	Tool list for Excellon data
Tutor.whl	Aperture list for Gerber data
Layers are aligned	
— Tutor1.cmp	Gerber data for the component side
— Tutor1.drl	Excellon data
— Tutor1.sld	Gerber data for the solder side
Layer has offset	
— Tutor2.cmp	Gerber data for the component side
— Tutor2.drl	Excellon data with offset to the Gerber data
— Tutor2.sld	Gerber data for the solder side
Layer is rotated	
— Tutor3.cmp	Gerber data for the component side
— Tutor3.drl	Excellon data rotated 90 degrees against the Gerber data
— Tutor3.sld	Gerber data for the solder side
Layer is mirrored	
— Tutor4.cmp	Gerber data for the component side
— Tutor4.drl	Excellon data
— Tutor4.sld	Gerber data for the solder side mirrored
Extended Gerber	
— Tutor.top	Extended Gerber data for the component side
— Tutor.bot	Extended Gerber data for the solder side
— Tutor.drd	Excellon data

2.1.3. Open sample circuit board

In this exercise, use the document based on the template file "**Default**". This document opens automatically after CircuitCAM starts.

How to create a new CircuitCAM document:

- Start CircuitCAM. The "**Untitled1**" document is opened.
- Select in the main menu **File -> Save As**. The **Save As** dialogue window appears on the screen.
- Enter the file name **Tutor1** into the **File name** field.
- Click on **Save** button to save document as a file with a name "**Tutor1.cam**". The file extension **.cam** is appended to the file name automatically.



Note: To abort the operation and close the dialogue window, click on **Cancel** button or press **ESC** key.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

2.1.4. Import Production Data

The following steps will guide you through the import of production data, which are generated by various CAD programs, into *CircuitCAM*.

In this exercise we have the following:

- Aperture and tool lists with corresponding definitions;
- Circuit board data for the *Component side* in Gerber format;
- Circuit board data for the *Solder side* in Gerber format.
- Drilling data in Excellon format.

How to import production data:

- Select in the main menu **File -> Import...** or click on in the **Standard** toolbar. The **Select files for import** dialogue window appears on the screen.

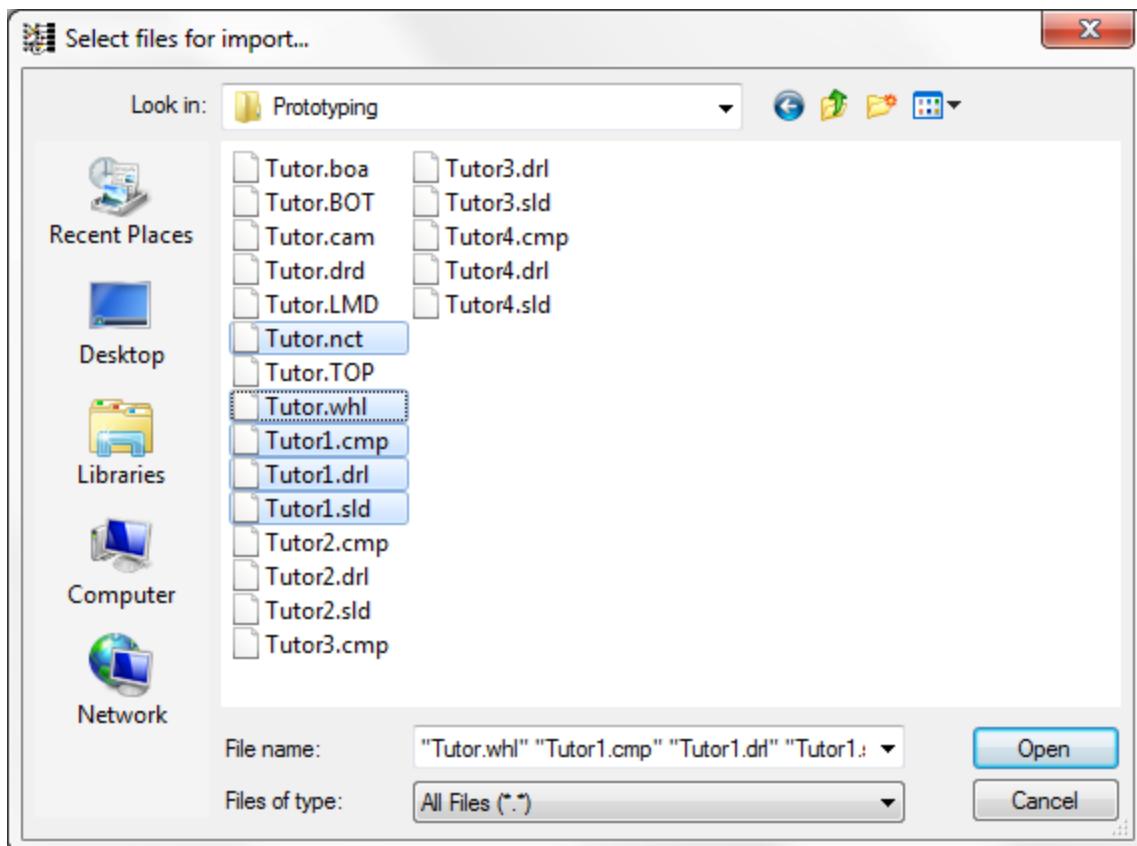


Figure 16. Select files for import dialogue window

- Navigate to the folder with the example files.
- Press and hold down the **CTRL** key and click on files *Tutor.nct*, *Tutor.whl*, *Tutor1.cmp*, *Tutor1.drl* and *Tutor1.sld*.



Note. *Gerber* and *Excellon* files miss the extended information and therefore we need the aperture (*Tutor.whl*) and tool (*Tutor.nct*) files . They are not standard, so we need a special *Template files* (i.e. "*TutorApe.txt*" and "*TutorTools.txt*") to decode the information. For more information see [Aperture Template Editor](#).

- Click on **Open** button.

The **Import** dialogue window appears on the screen.

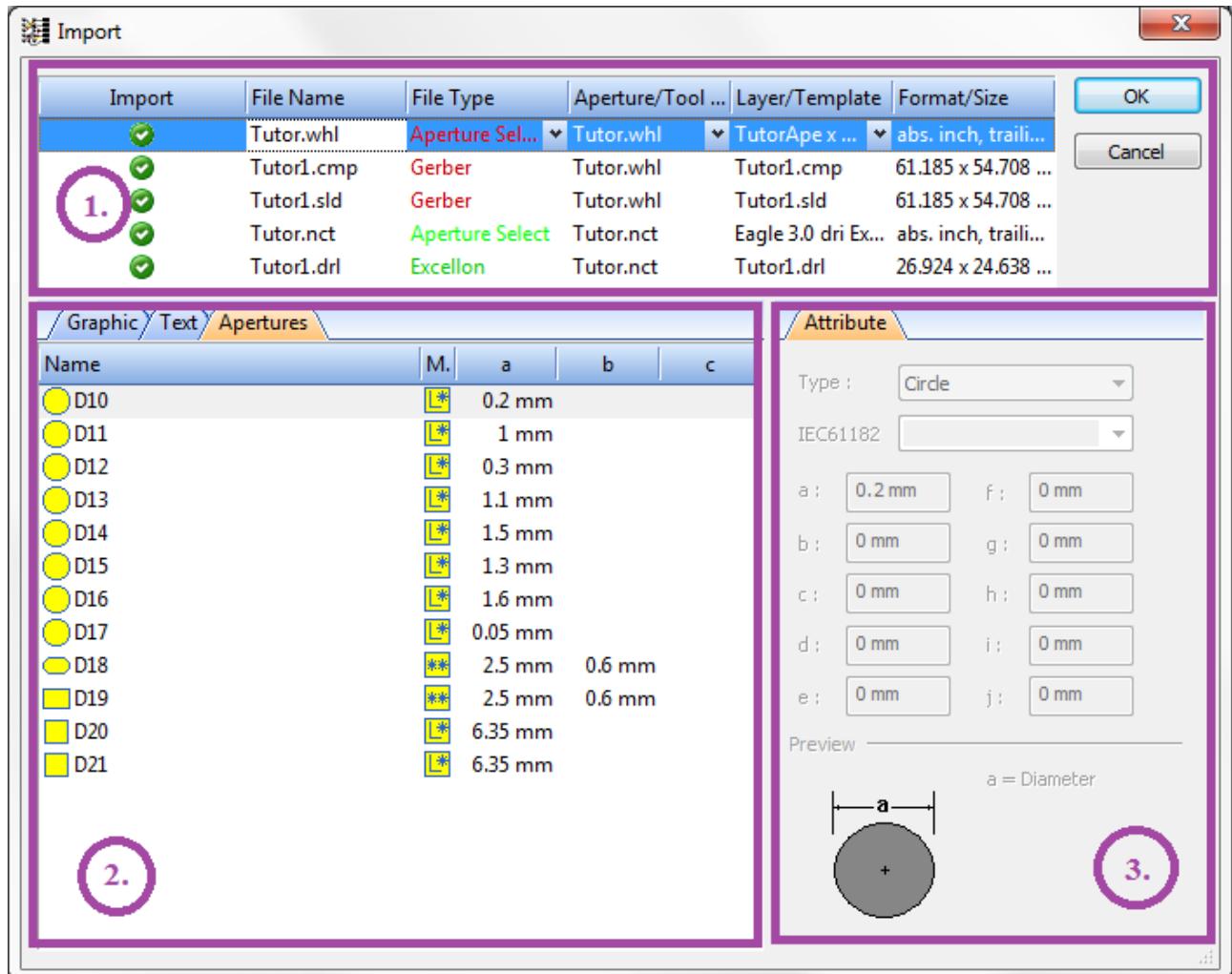


Figure 17. Import dialogue window: 1-list of imported files; 2-Graphic, Text and Apertures panels; 3-Attribute panel

CircuitCAM analysed the selected files and sorted the file list (1) into the following order:

1. Aperture list
2. Gerber files
3. Tools list
4. Excellon file.

Aperture List

How to assign translation file for aperture list:

- Select **Tutor.whl** in the **File Name** of the **Import** window (1st section).
- Click on the **Layer/Template** field and select a translation file for the aperture list **TutorApe.TXT**.

Import	File Name	File Type	Aperture/Tool List	Layer/Template	Format/Size
✓	Tutor.whl	Aperture Select	Tutor.whl	TutorApe.TXT	abs. inch, trailing,...
✓	Tutor1.cmp	Gerber	TutorApe x whl Ger 0.01mm Inch23AbsTrail.txt		61.185 x 54.708 mm
✓	Tutor1.sld	Gerber	TutorApe.TXT		61.185 x 54.708 mm
✓	Tutor.nct	Aperture Select	Tutor.nct	Eagle 3.0 dri Exc I...	abs. inch, trailing,...
✓	Tutor1.drl	Excellon	Tutor.nct	Tutor1.drl	26.924 x 24.638 mm

Figure 18. Select translation file for aperture list

How to check the translation results:

- Click on the **Text** tab to activate the panel. You can see that the aperture **D10** is defined as a circle with a diameter 0.2 mm (see [Figure below](#)).
- Inspect all apertures, if the conversation results are correct.

Aperture Table for Tutor sample files
Format Unit: Inch
m.n Code: 2.3
Mode: absolute
Zeros: leading suppressed

Aperture unit: 1/100 mm

D_Code	Mode	Shape	X-size	Ysize
D10	Draw	Circle	20	
D11	Draw	Circle	100	
D12	Draw	Circle	30	
D13	Flash	Circle	110	
D14	Draw	Circle	150	
D15	Flash	Circle	130	
D16	Flash	Circle	160	
D17	Draw	Circle	5	
D18	Flash	Finger	250	60
D19	Flash	Rectangle	250	60
D20	Flash	Square	635	
D21	Draw	Square	635	

Figure 19. Text panel with translation results

Assignments for Gerber Files

How to assign layers for Gerber files:

- Select **Tutor1.cmp** file in the **File Name** field. This file contains the data for the component side of the circuit board.

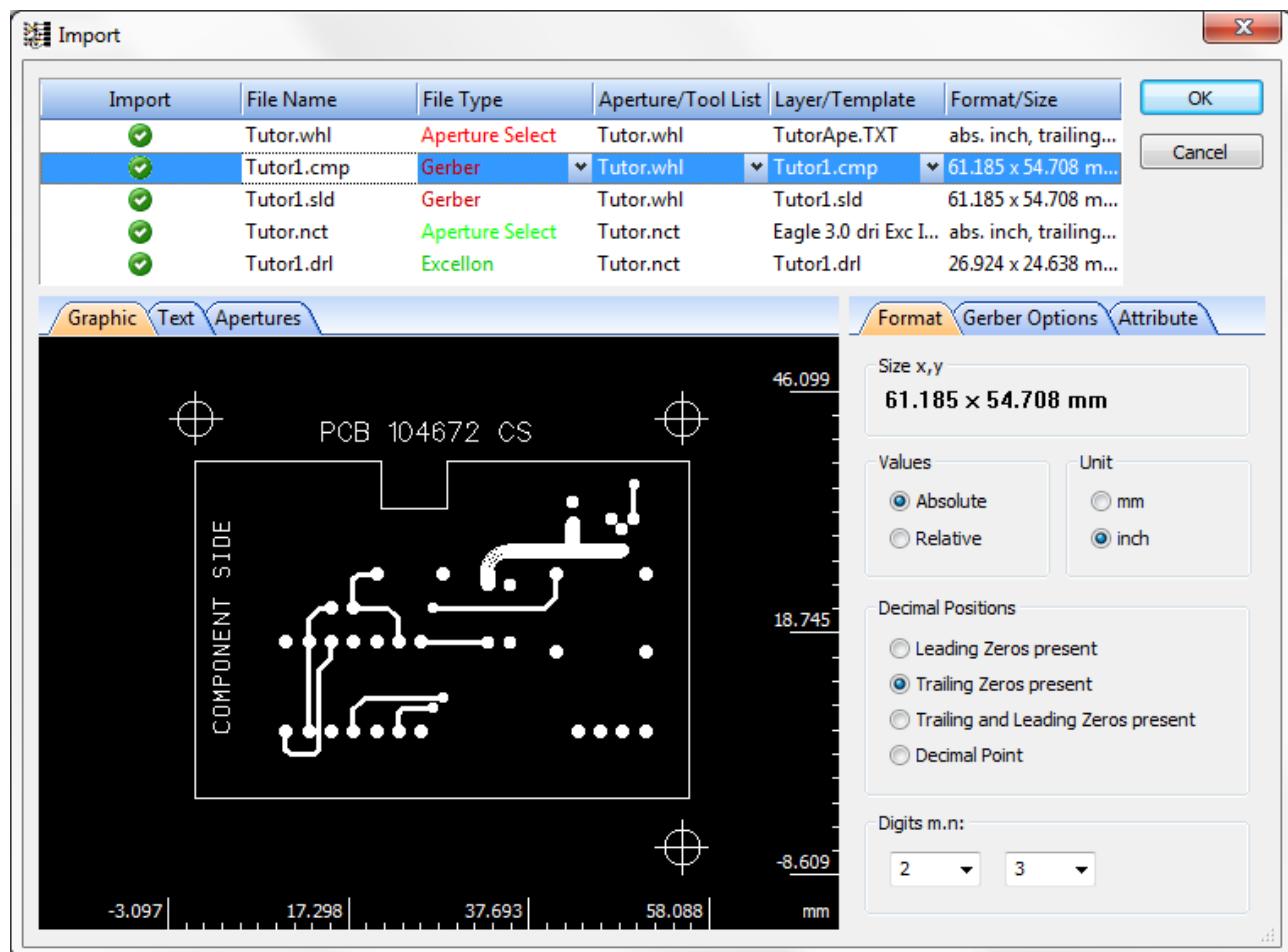


Figure 20. Gerber file content, circuit board component side

- Click on the **Layer/Template** field and select *TopLayer* in the drop down list. The component side of the circuit board is assigned to the *TopLayer* (see [Figure below](#)).

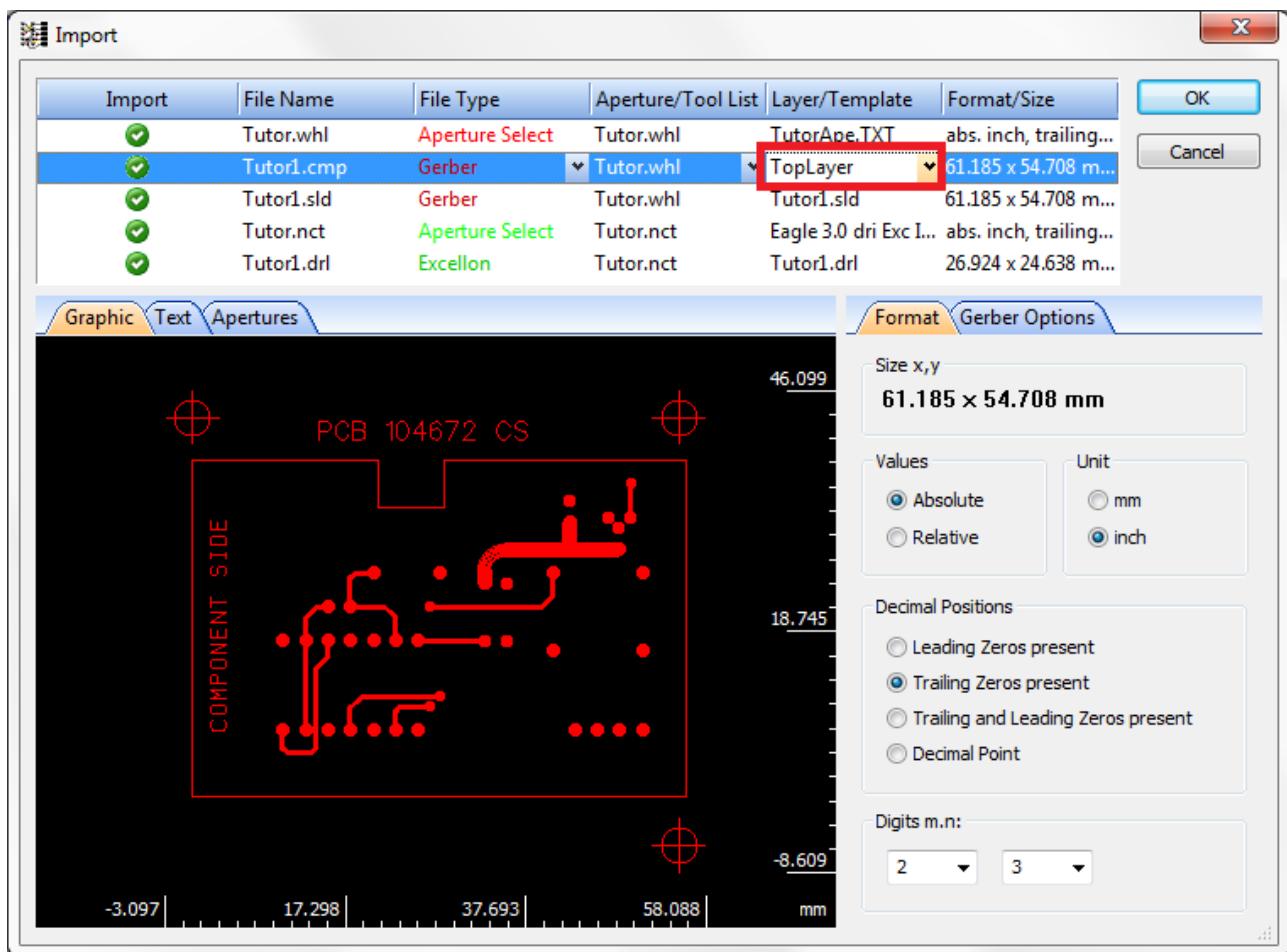


Figure 21. Component side is defined as *TopLayer*

- Select **Tutor1.sld** in the **File Name** field. This **Gerber** file contains the data for the solder side of the circuit board.

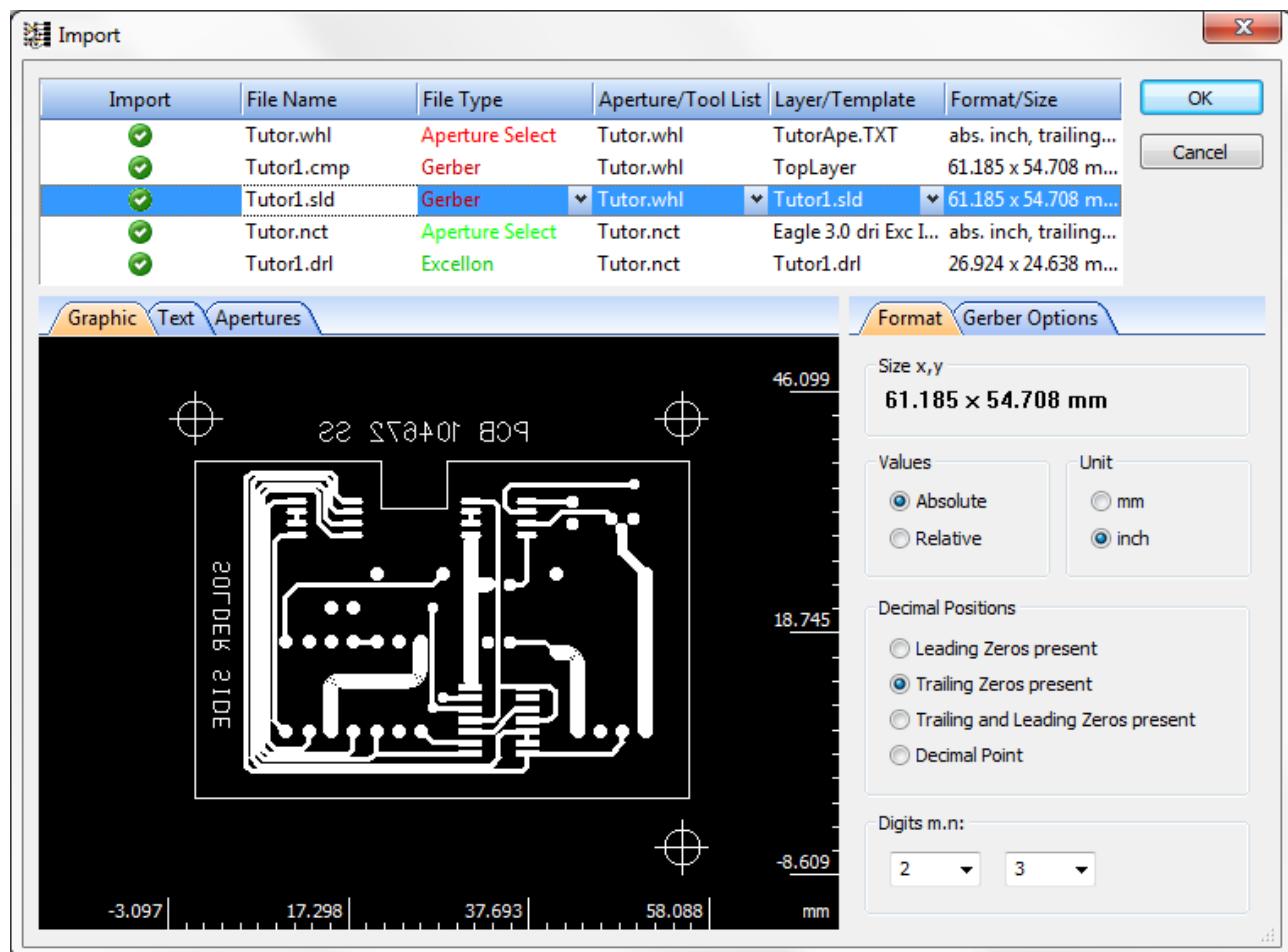


Figure 22. Gerber file content, circuit board solder side (mirror presentation)

- Click on the **Layer/Template** field and select **BottomLayer** in the drop down list. The solder side of the circuit board is assigned to the *BottomLayer* (see [Figure below](#)).

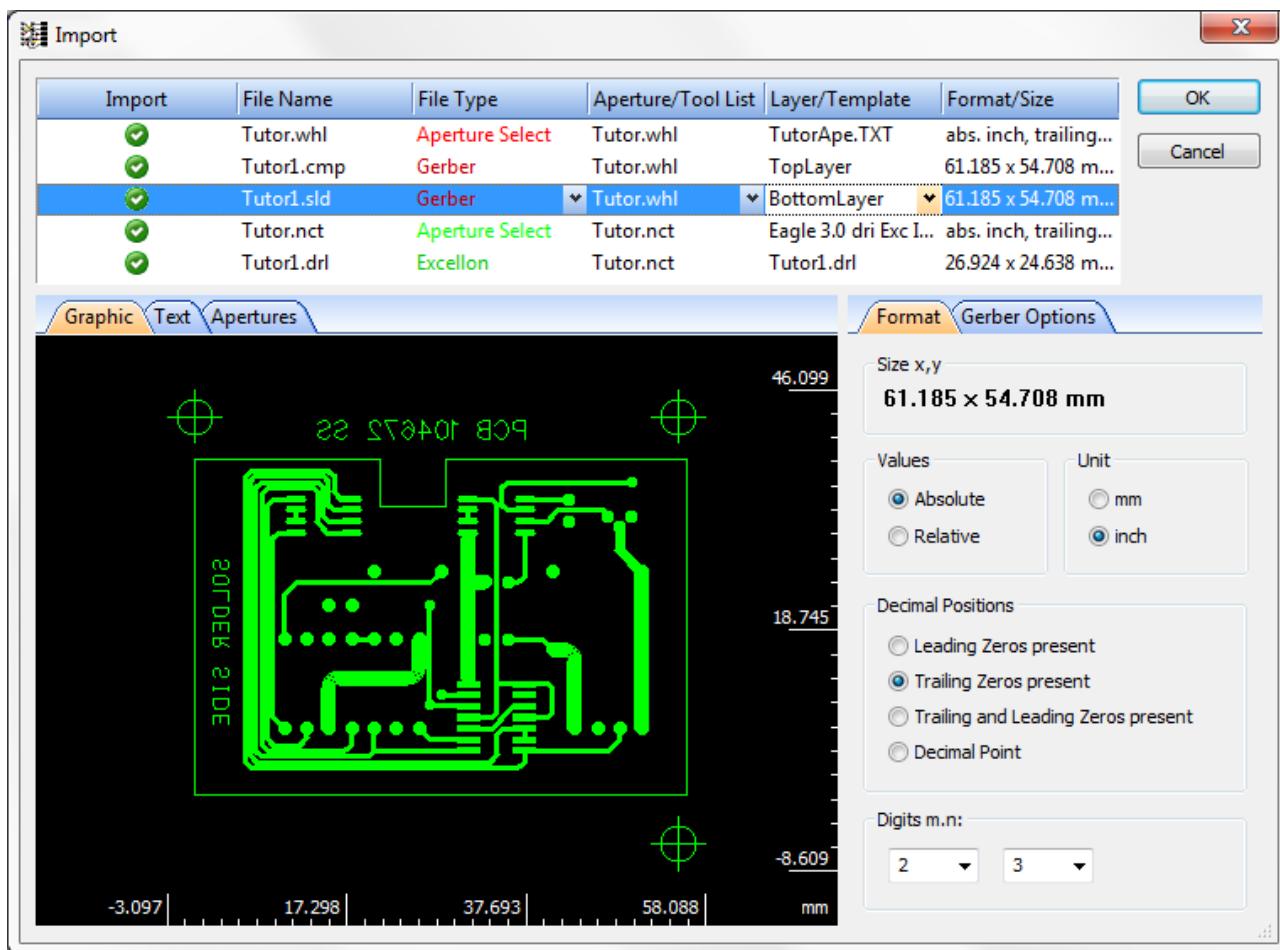


Figure 23. Solder side is defined as *BottomLayer*



Note. You can import data without defining the layer. The file name will be treated as a layer name. You can assign layers after the import is completed.

Tool List

How to assign translation file for a tool list:

- Select **Tutor.nct** file in the **File Name** field. The file contains the list of tools, which are displayed in the **Apertures** panel (2nd section). The tool list contains three tools. Check, if the individual diameter of each tool satisfy your requirements. In our example, diameters are too large (see Apertures panel, column a). That means, that not correct translation file for the tool list is used.

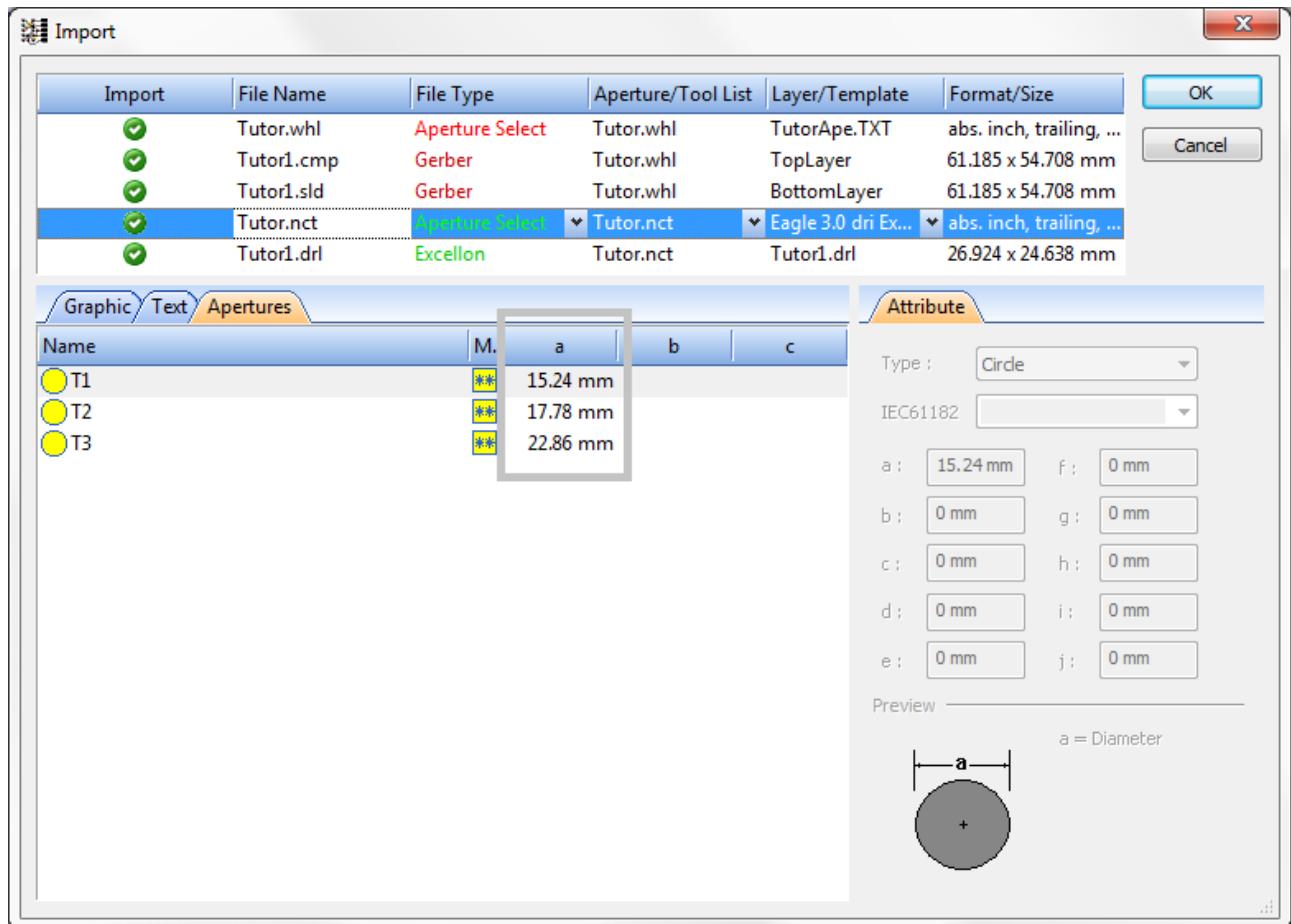


Figure 24. Contents of tool list in Apertures panel

- Click on the **Layer/Template** field and select a translation file for the tool list **Tutor-Tools.TXT**. Now the selected translation file produces the desired results.

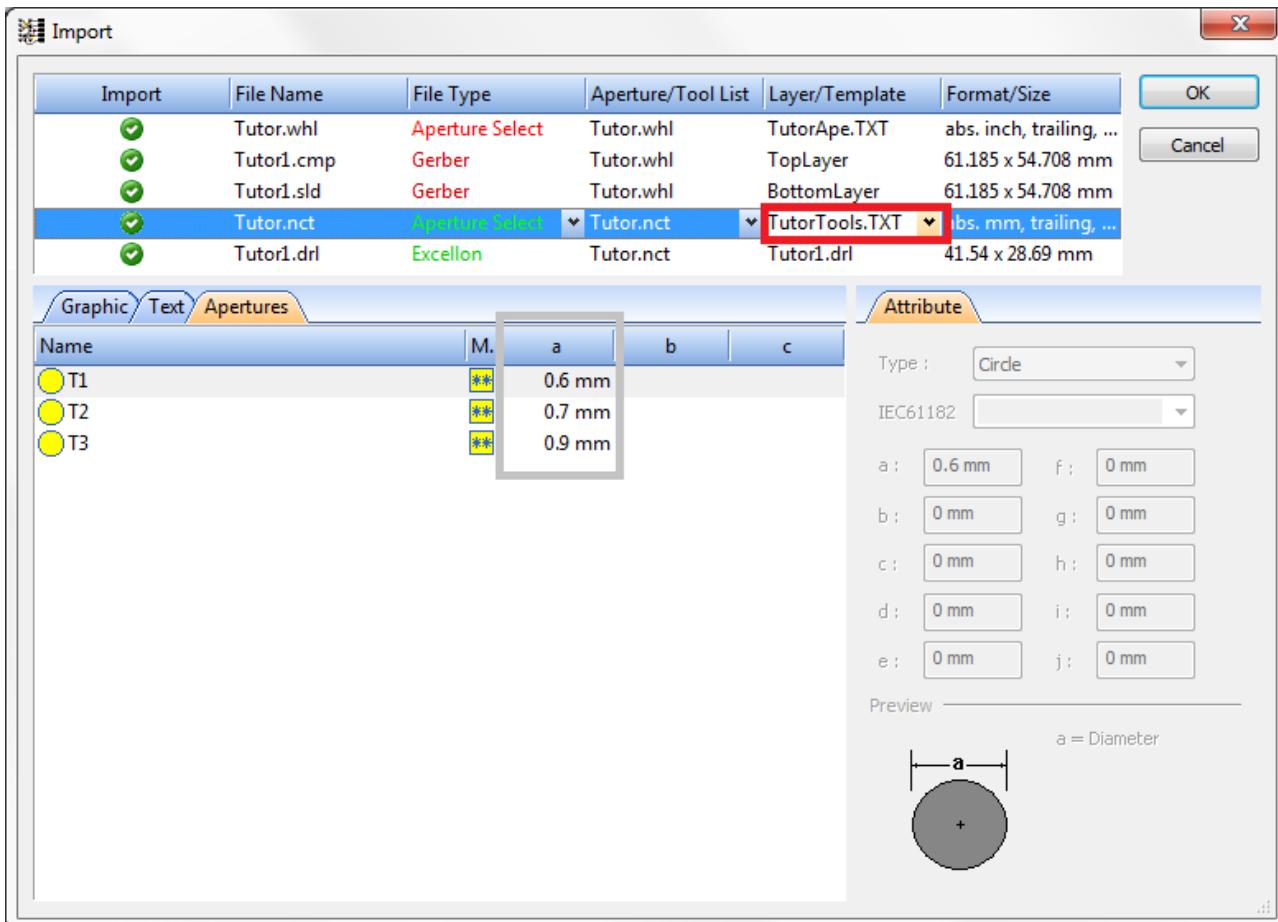


Figure 25. Selected translation file and tool list with parameters in Apertures panel

Assignments for Excellon Files (drills)

How to assign layer to *Excellon* file:

- Select *Tutor1.drl* in the **File Name** field. The translation result is previewed in the **Graphic** panel.

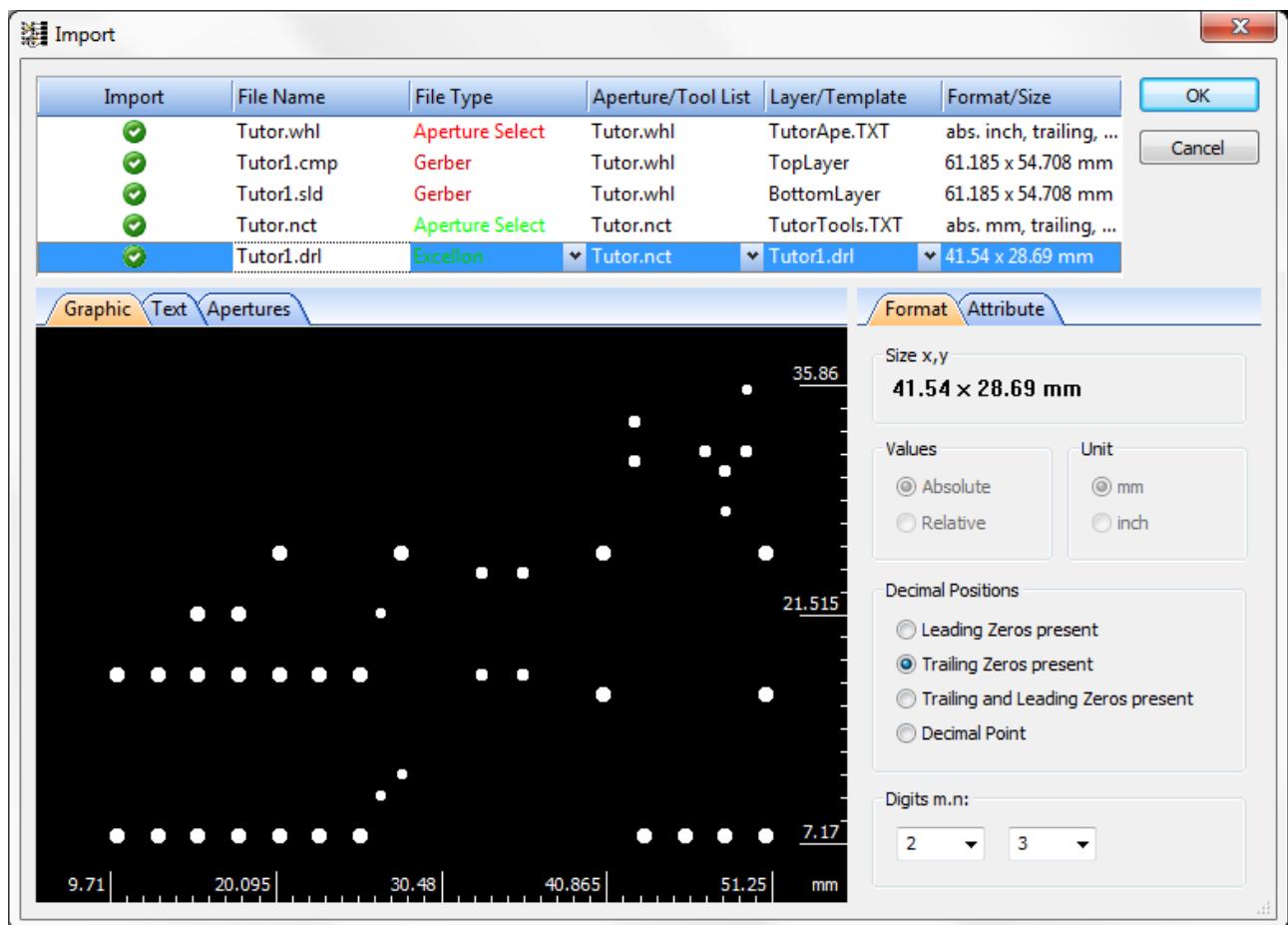


Figure 26. Graphic view content of selected Excellon file

- Click on the Layer/Template field and select **DrillPlated** in the drop down list. The holes of **Excellon** file are assigned to the *DrillPlated* layer.

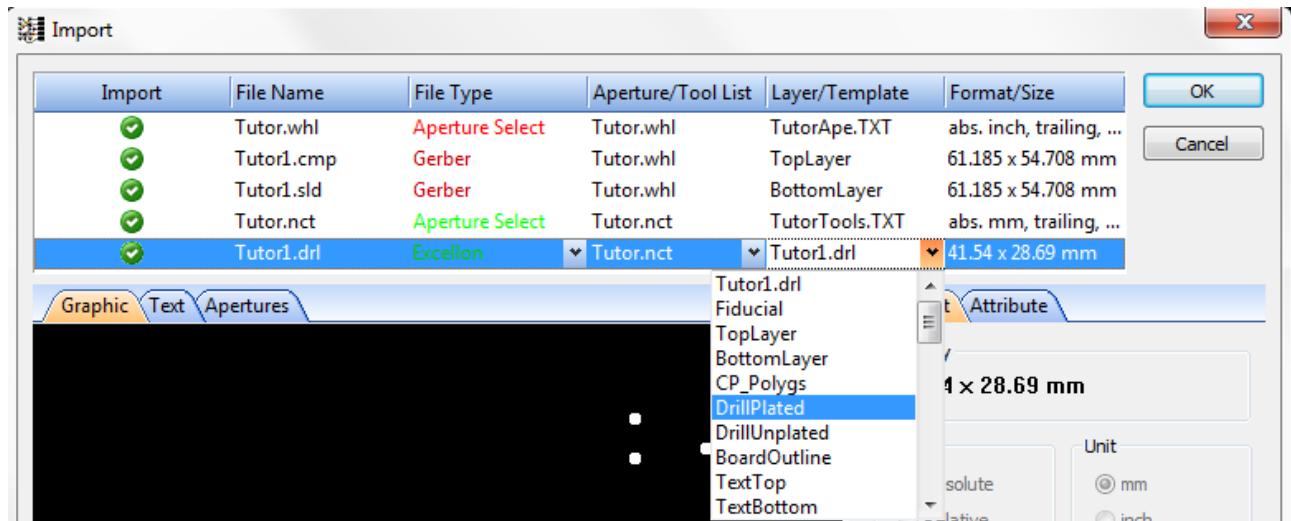


Figure 27. Select DrillPlated layer



Note: In our example the circuit board is two-sided, thus holes are plated through. In case of single-sided board, the *DrillUnplated* layer should be used.



Note: In some cases the formatting parameters for holes can be not correct (see [Figure below](#)): holes overlap and the dimensions (red square), which appear in the **Format/Size** field do not correspond to the ones for **Gerber** files (green square).

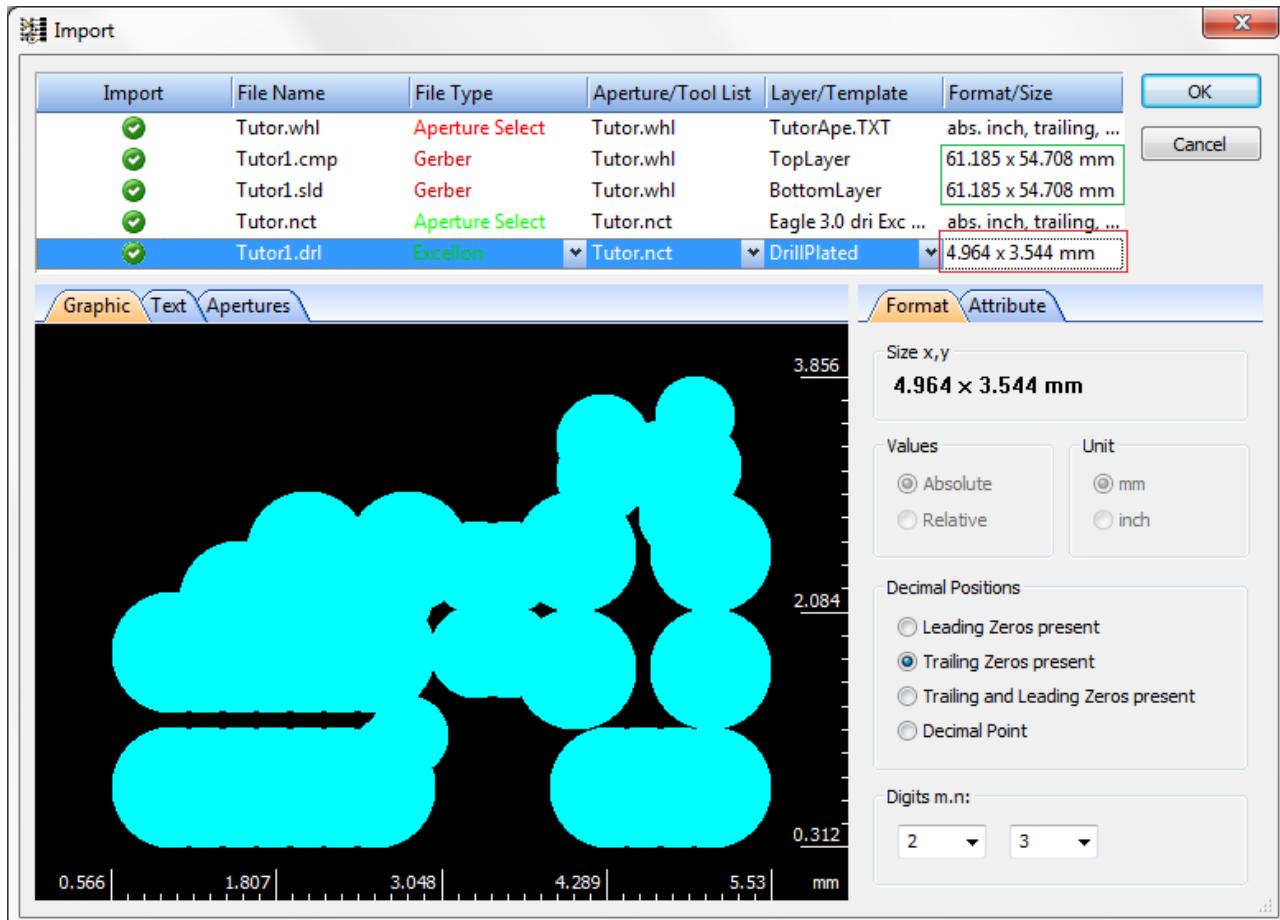


Figure 28. Overlapping holes in Graphic view

To fix dimensions, the dimensions on the graphic should be multiplied by approximately 10. To change the dimension size by 10, move the decimal place.

How to change format parameters for **Excellon** file:

- Click on the arrow next to the decimal places field under the **Digits** in the **Format** panel.
- Select the value 2. The dimensions of the graphic are changed and holes do not overlap.

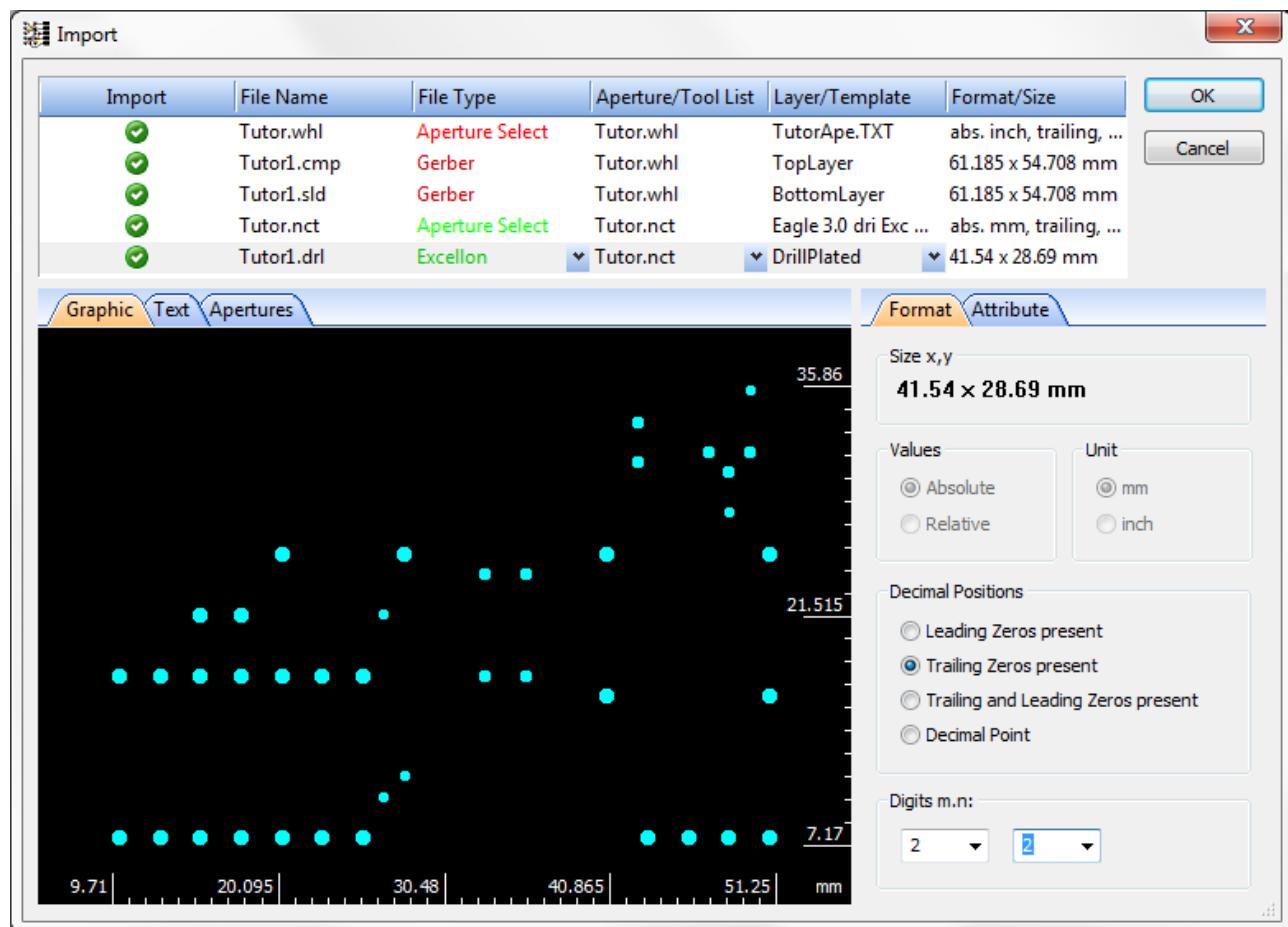


Figure 29. Graphic view with correct format parameters

All changes are made in the **Import** dialogue window.

- Click on **OK** button to import selected files.
CircuitCAM displays imported product data.

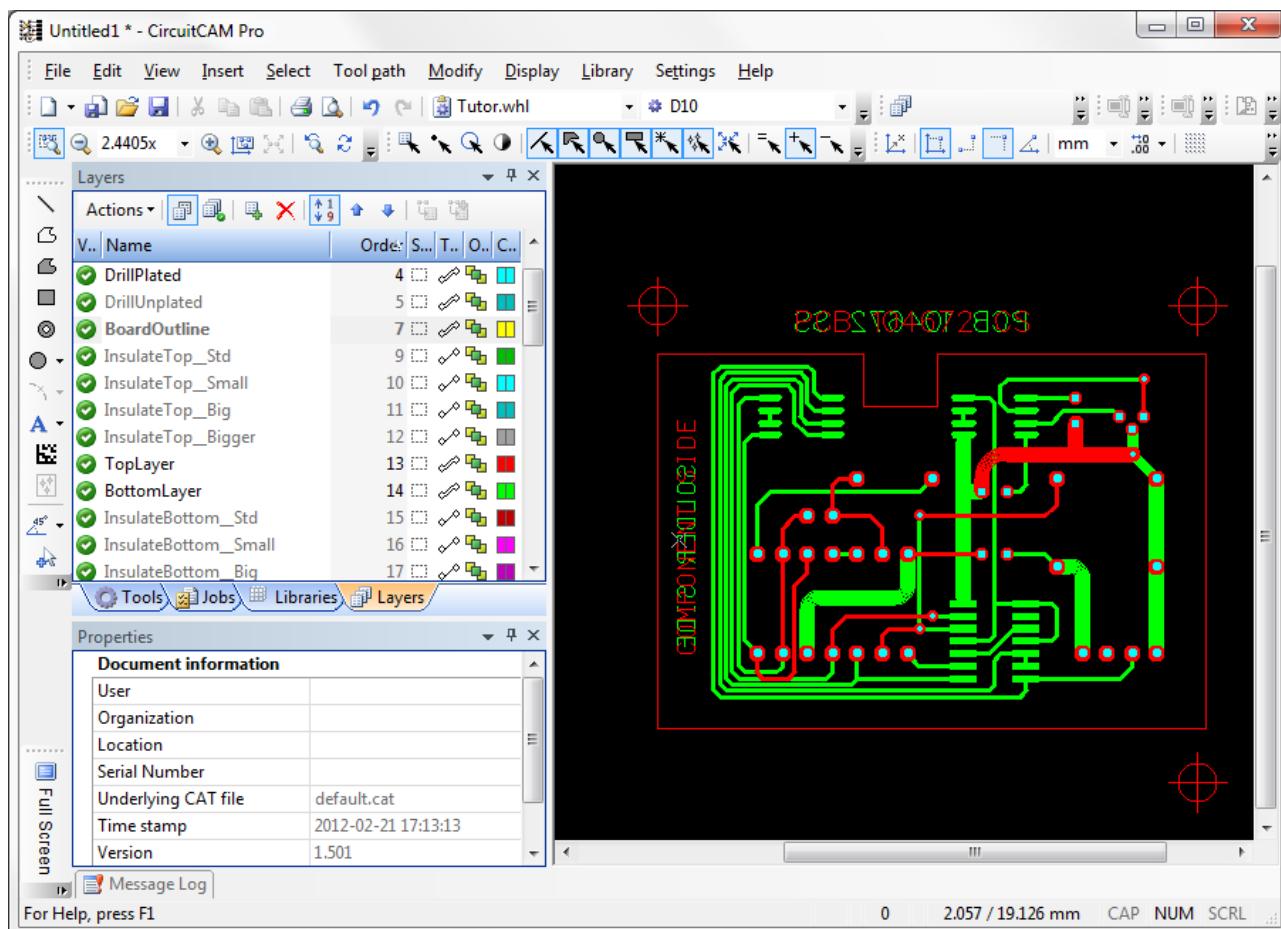


Figure 30. *CircuitCAM* displays imported product data



Save your data! (under *Tutor1.cam* name).

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

2.1.5. Import Extended Gerber Data

To import *Extended Gerber* files, you do not need to specify aperture/tools list or template file. All needed information is included in the *Extended Gerber* file.

In our example we will introduce, how to import the following Extended Gerber format files:

- *Tutor.top*
- *Tutor.bot*
- *Tutor.drd*



Note: Open a new document.

How to import Extended Gerber production data:

- Select in the main menu **File -> Import** or click on in the **Standard** toolbar. The **Select files for import** dialogue window appears on the screen.
- Navigate to the folder with the example files.
- Press and hold down the **CTRL** key and click on files *Tutor.top*, *Tutor.bot*, *Tutor.drd*.



Note: You can see, that we do not specify any additional aperture/tool lists, because the needed lists are already included in the Extended Gerber files.

- Click on **Open** button. The **Import** dialogue window appears on the screen.

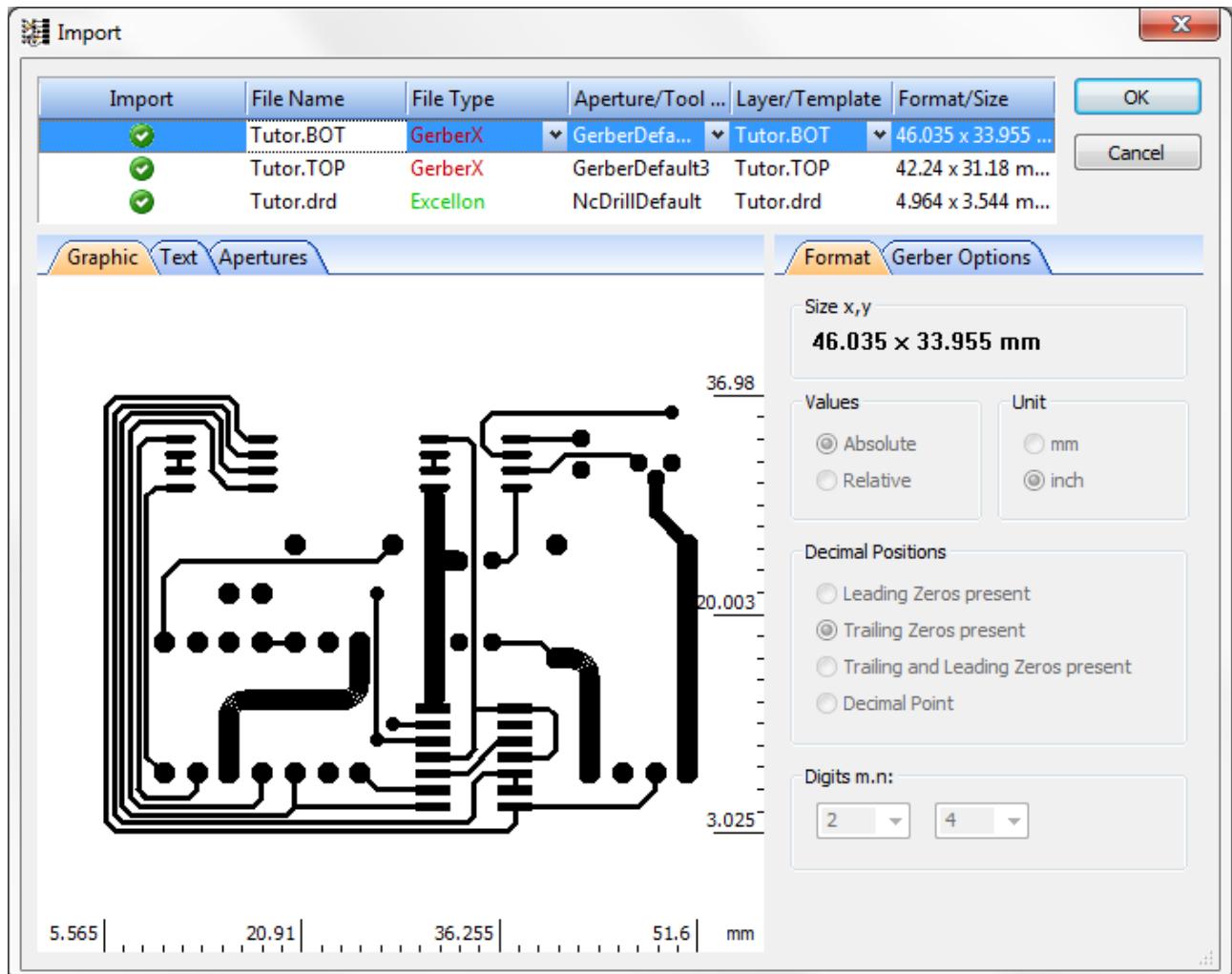


Figure 31. Import dialogue window with Extended Gerber files list

Now let's assign layers

- Select *Tutor.bot* file in the list to assign layer for a solder side of the circuit board.
- Click on the **Layer/Template** field and select *BottomLayer* in the drop down list.
- Select *Tutor.top* file in the list to assign layer for a component side of the circuit board.
- Click on the **Layer/Template** field and select *TopLayer* in the drop down list.
- Select *Tutor.drd* file in the list to assign layer for drills.
- Click on the **Layer/Template** field and select *DrillPlated* in the drop down list.

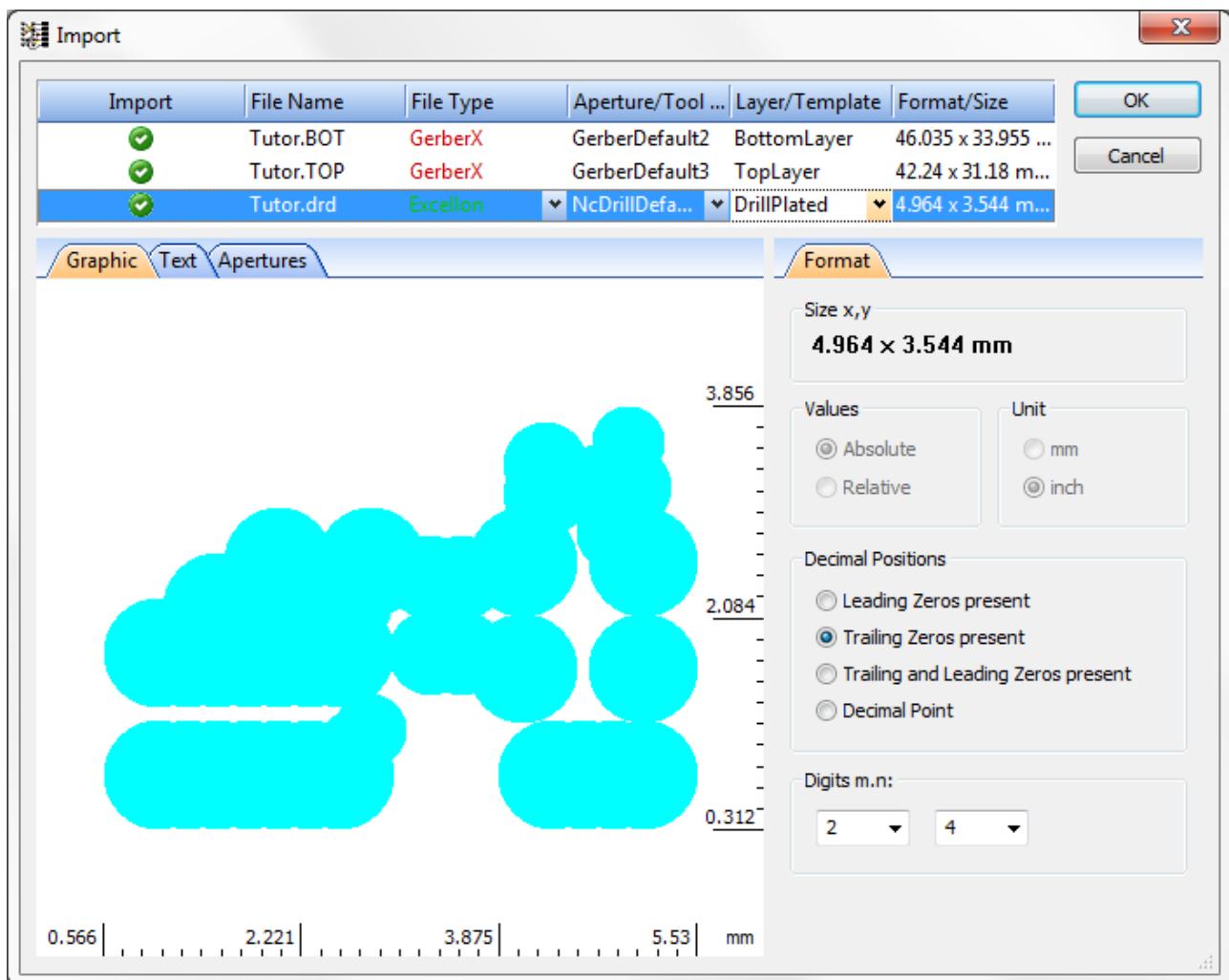


Figure 32. Import dialogue window with graphic view of drill file content

- In the Format panel, Digits layout, set **n** value to 3 to set a right size for drill data.

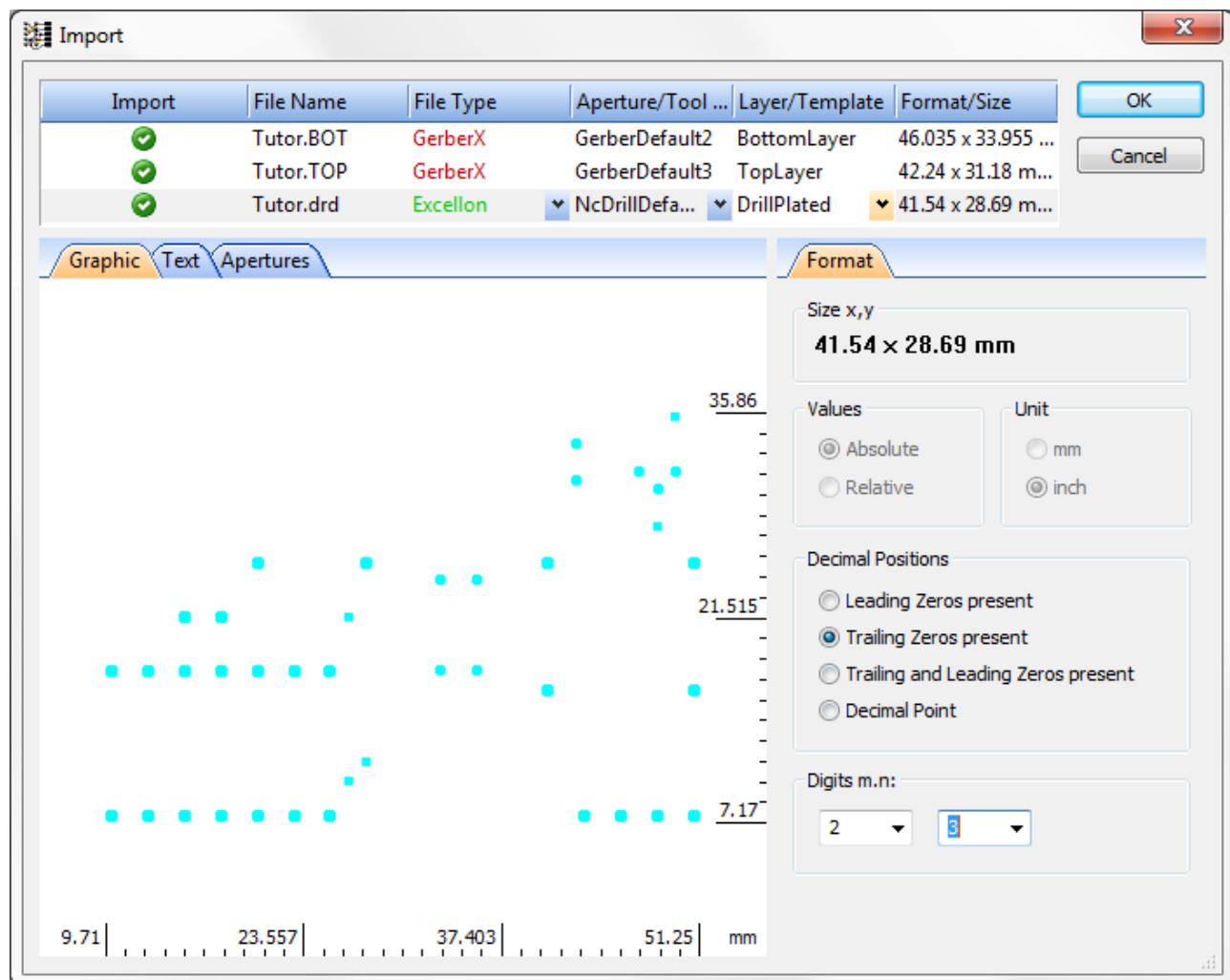


Figure 33. Import dialogue window with prepared Extended Gerber data for import

- Click on the **OK** button.

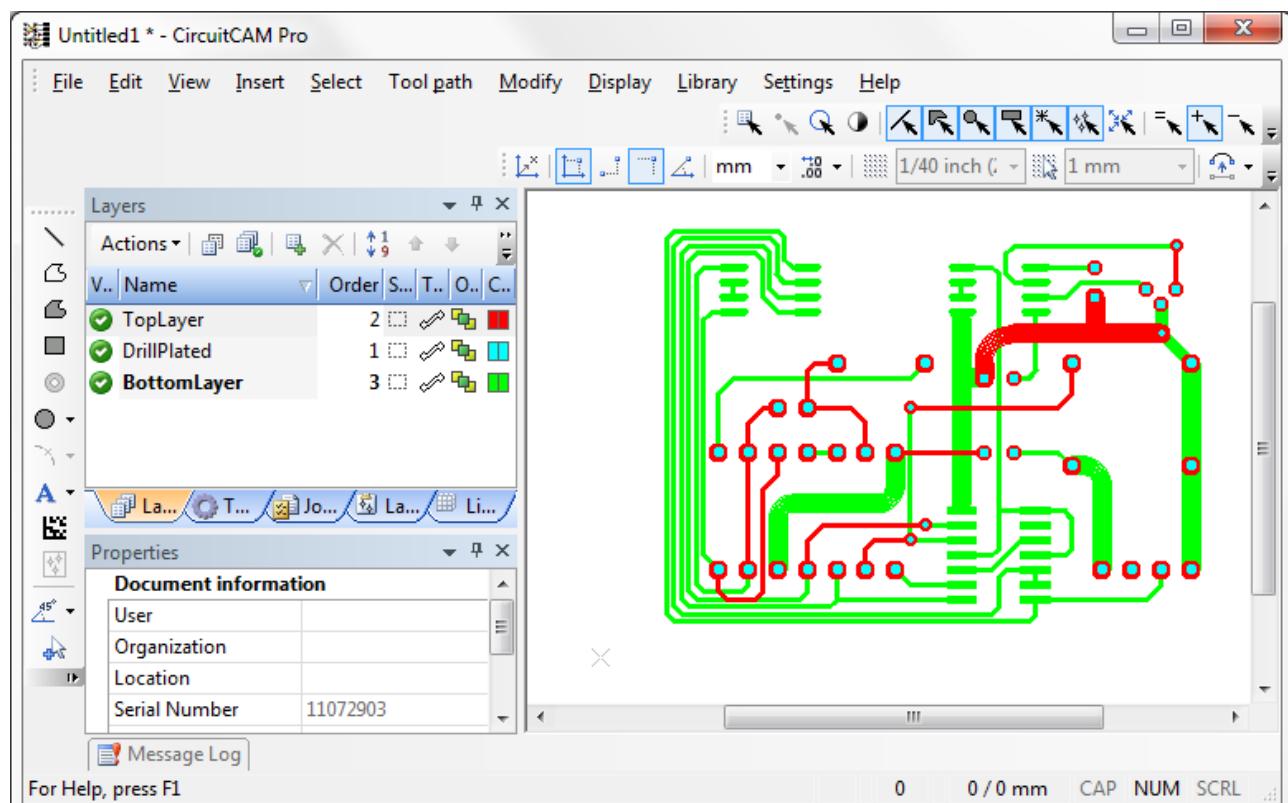


Figure 34. CircuitCAM window with imported Extended Gerber production data

Note: You can save the document (i.e.: under *Tutor6.cam* name).

Close the current file, because we will not use these data in further examples.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

2.1.6. Save Data

To save modified production data:

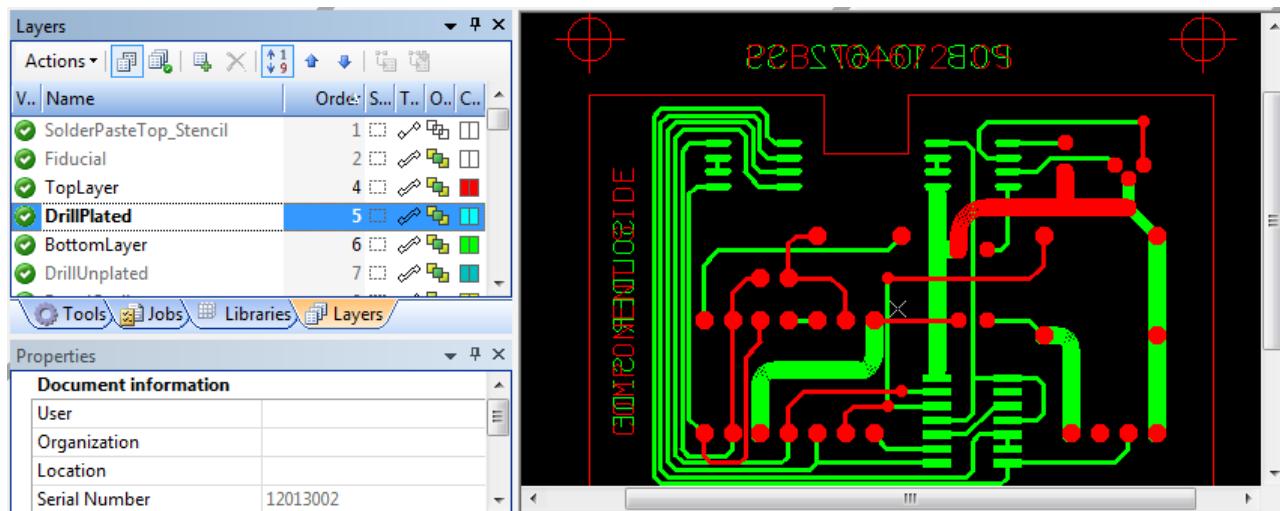
- Select in the main menu **File -> Save**. In case the first save action, the **Save as** dialogue window appears on the screen.
- Specify the name into the **File** name field and click on **Save** button.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

2.2. Prepare PCB Layout

2.2.1. Change Order of Layers

After product data are imported to *CircuitCAM*, they are located on the specified [layers](#). In case the order of layers does not satisfy your requirements (i.e.: some layers are not visible), you may change the order of layers.

**Figure 35.** Objects of active *DrillPlated* layer are not visible in Graphic view

To change the order of layers:

- Press with a mouse on the layer name in the **Layers** panel.
- Hold and drag the layer to the desired position (up or down). In our example drag up. While dragging, the red line indicates where the layer could be placed.

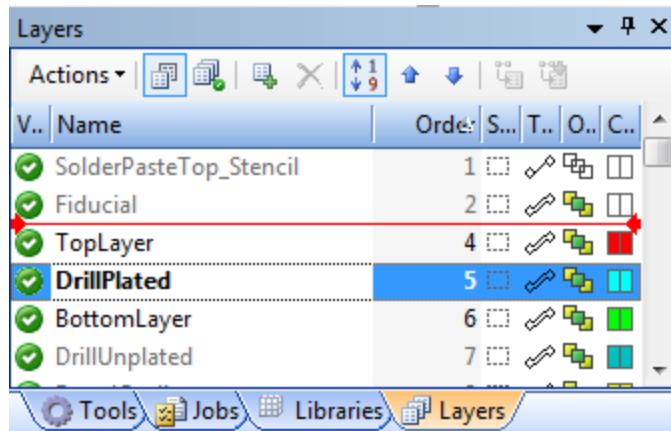


Figure 36. Dragging of **DrillPlated** layer over **TopLayer**

- Release the mouse button, when the layer is in the desired position. The order of the selected layer is changed (see **Order** cell). Components of the selected layer appear in the **Graphic** view.

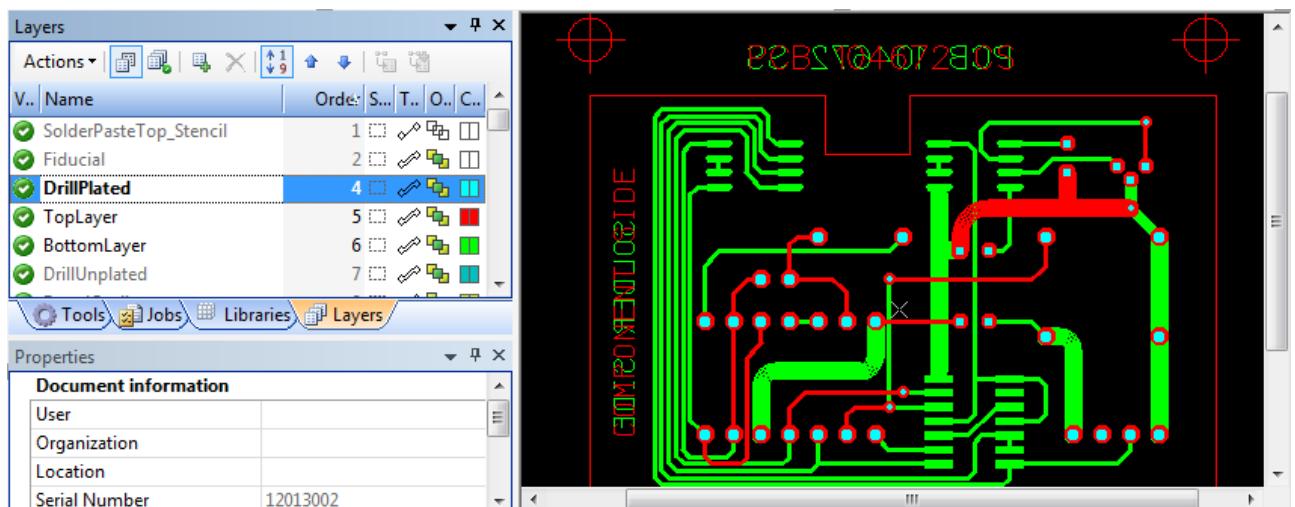


Figure 37. Components of **DrillPlated** layer are displayed in Graphic view (light blue color)

You can also change the order of the selected layer by using tool bar buttons:

- Press on **↑↓ 1 9** button to turn the **Order Arrangement Mode On**.
- Use arrow buttons **↑ Move up** and **↓ Move down** to locate the selected layer.

To make the selected layer on the very top of the list - double click on the **Order** cell.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

2.2.2. Move Layer

The production data presented in the previous example, represents an ideal case.

Usually production data not match because of different data origins.

CircuitCAM helps you to correct production data according your needs.

Lets' import not ideally matching production data and try to correct them:

- Open a new document, based on **default_insulate.cat** template file, i.e.: select in the main menu **File -> New -> default**.
- Import the following production data from the "..\ExampleData\Prototyping" sub-directory:
 - Tutor2.cmp
 - Tutor2.sld
 - Tutor2.drl
 - Tutor.whl
 - Tutor.nct

For "Tutor.nct" file Layer / Template transition file must be changed into "ToturTools.TXT". This file will load the new aperture tool size.

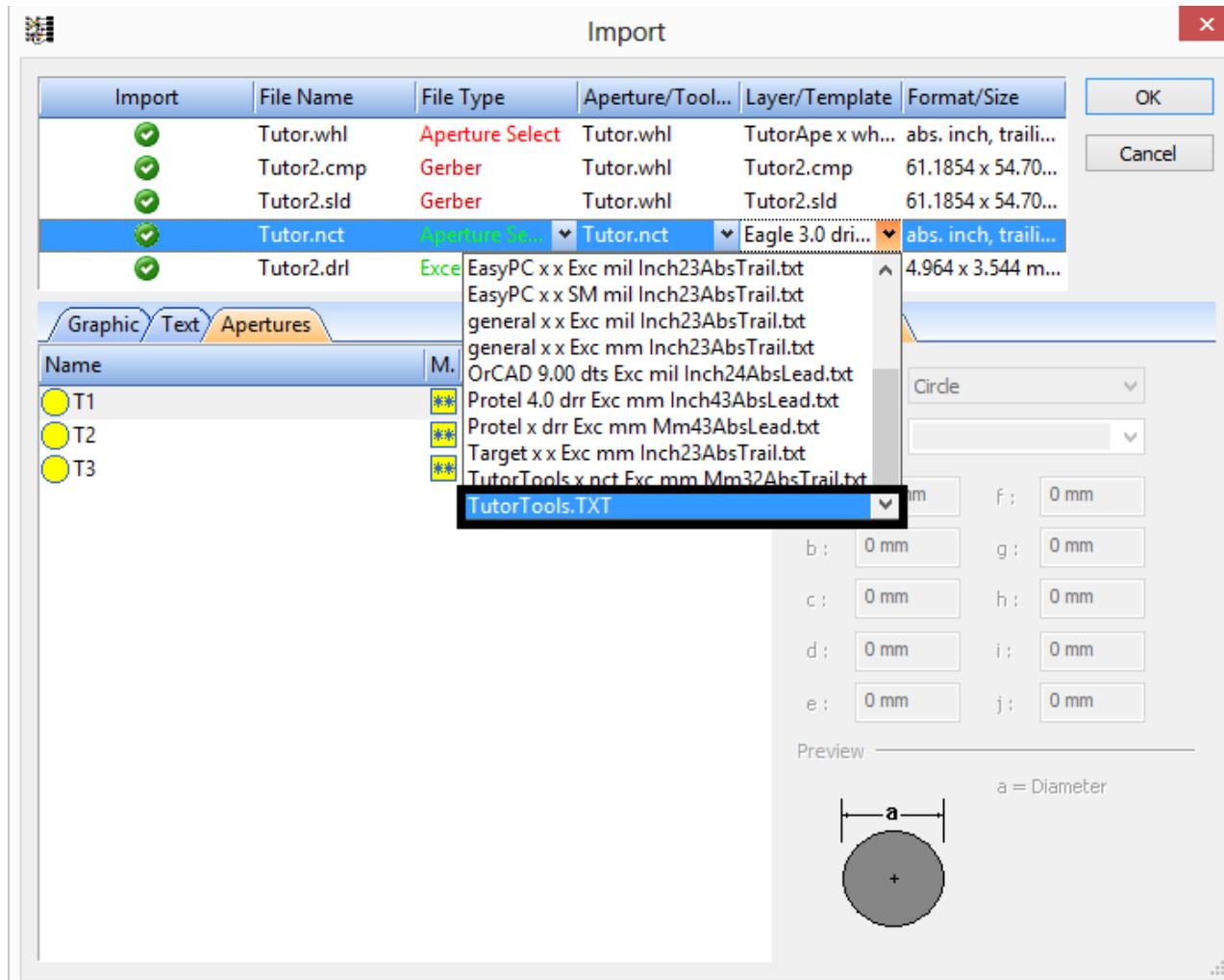


Figure 38. Layer / Template is changed to ToturTools.TXT for changing aperture tool size.

Imported production data appear in *CircuitCAM* window.



Note: Save data under "**Tutor5.cam**" file name.

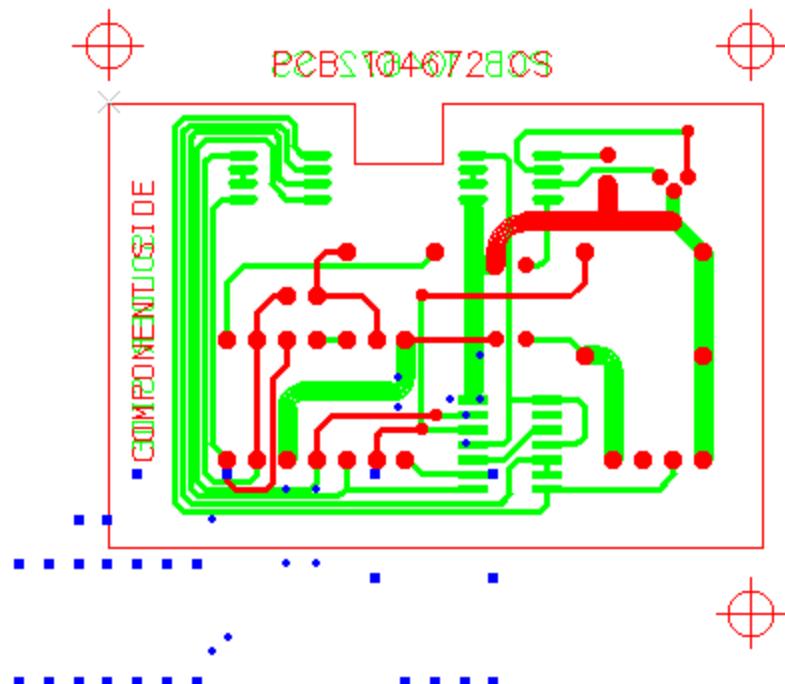


Figure 39. Circuit board graphic view

In this example **Excellon** data do not coincide with **Gerber** data.

How to make layers coincide, move to correct position:

- Click on **Match up Layers** button in the **Layer** tool bar.



Note: You have to select a significant object, which is present on both layers. A significant object may be a drill hole (1) and the associated pad (2). Both graphic objects have a centre coordinate, which clearly defines a position.

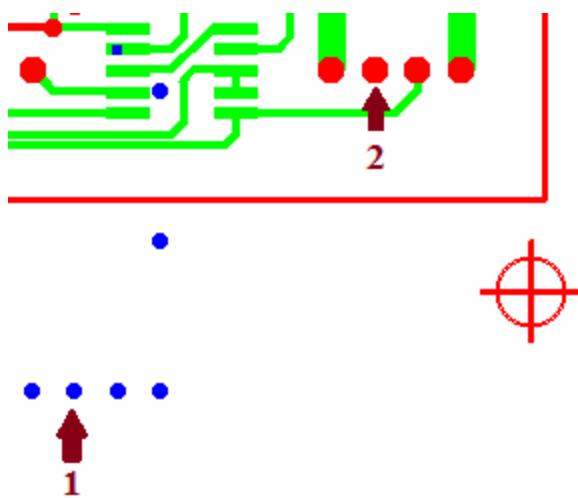


Figure 40. Moving layer

- Click on the lower hole on the *DrillPlated* layer (1) to select it.
- Move mouse to another selection point . You can see the line, which stretches from the previously selected point (1).
- Click on the upper pad on the *TopLayer* (2) to select it.

The layer *DrillPlated* moves to the correct position. The layers *DrillPlated* and *TopLayer* coincide now.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

2.2.3. Rotate Layer

In this example we will show, how to correct data, when the ***Excellon*** file includes data, which are an offset against ***Gerber*** data and rotated by 90°.

- Open a new document, based on the "***default_insulate.cat***" template file. Select in the main menu **FILE -> New -> default_insulate**.
- Import the following production data from the "..\ExampleData\Prototyping\" subdirectory:
 - Tutor3.cmp
 - Tutor3.sld
 - Tutor3.drl
 - Tutor.whl
 - Tutor.nct

For "Tutor.nct" file Layer / Template transition file must be changed into "Totur-Tools.TXT". This file will load the new aperture tool size.

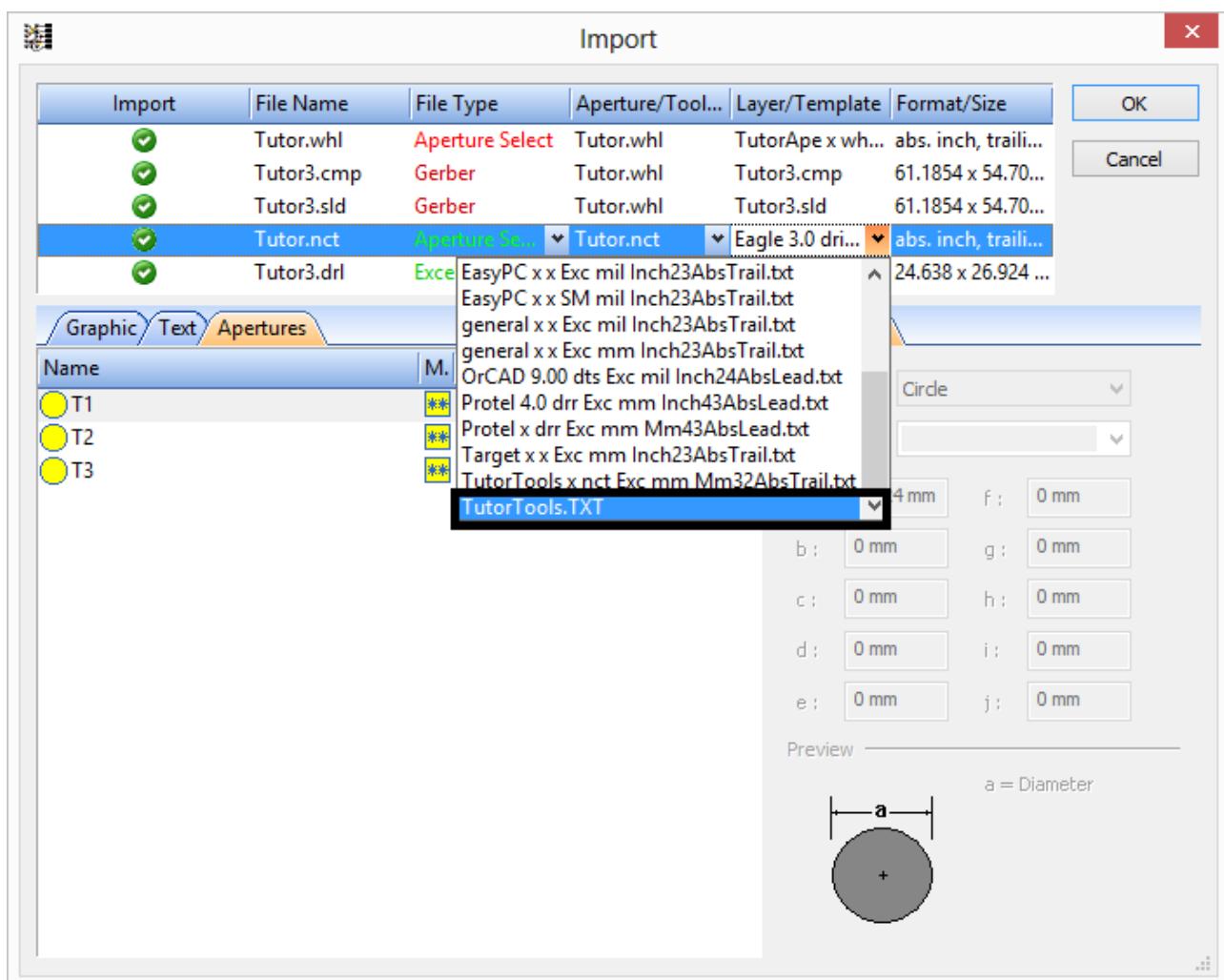


Figure 41. Layer / Template is changed to ToturTools.TXT for changing aperture tool size.

Imported production data appear in *CircuitCAM* window. To correct data, we have to rotate the *DrillPlated* layer holes and match up with a *TopLayer*.



Note: Save data under "**Tutor7.cam**" file name.

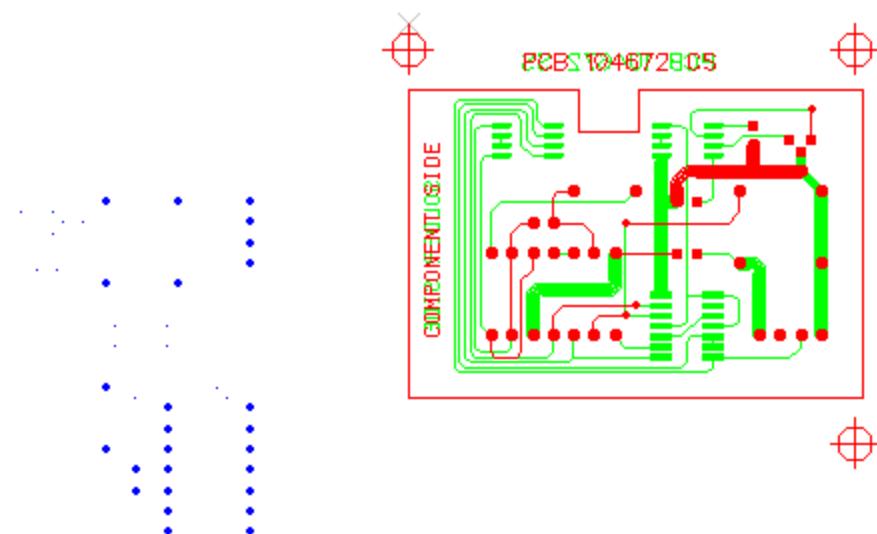


Figure 42. Circuit board graphic view

How to rotate layer:



Note: A selected object always rotates around the anchor point.

- Click on the drill hole at the bottom right to set an anchor point.



Figure 43. Select hole (set anchor point)

The selected drill hole is a reference point around which the layer should be rotated. The active layer *DrillPlated* is highlighted in the **Layers** panel.

- Click on the **Select Layer** button in the **Select** toolbar to select all objects on the *DrillPlated* layer.

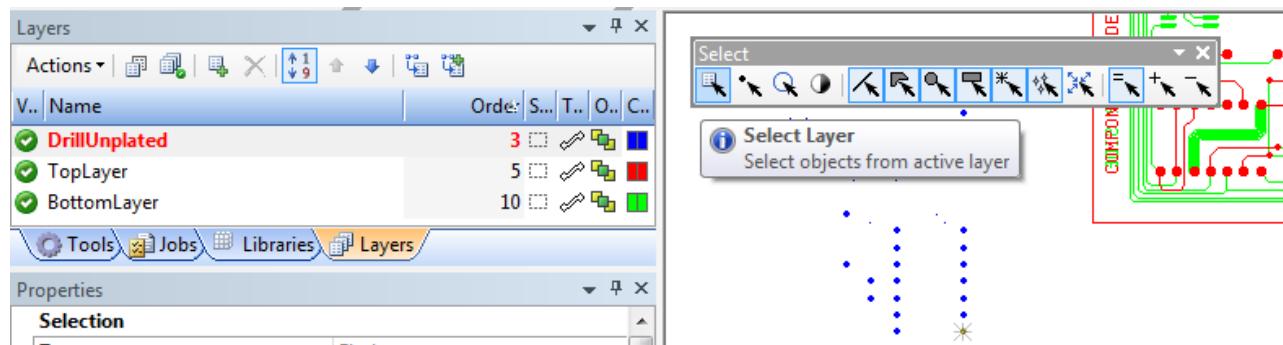


Figure 44. Active layer in Layers view and Select Layer button in Select tool bar

- Click on the 270° button in Rotate Relative to rotate the currently selected layer by 270° anticlockwise.

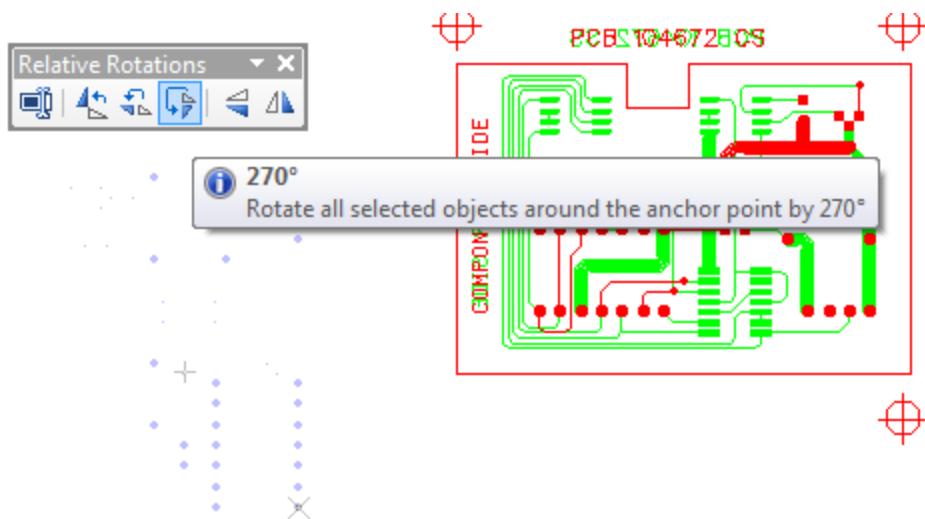
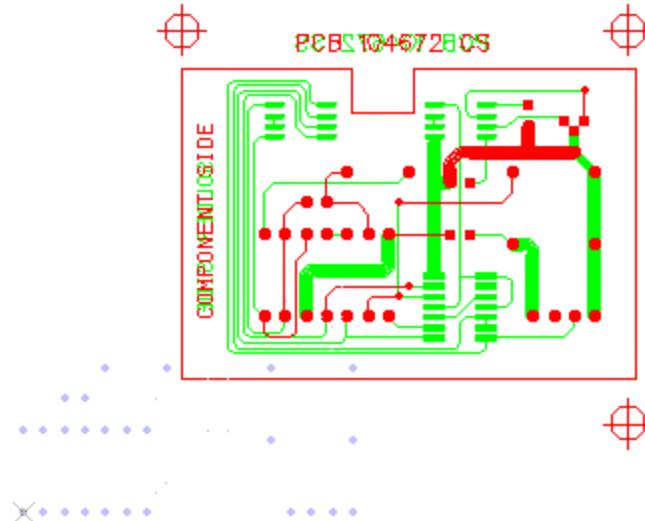


Figure 45. Rotate layer 270° around anchor point

The Excellon data is aligned in the same way as Gerber data, but still has an offset.

**Figure 46. Rotated layer**

- Use the **Match up Layers** button to coincide *DrillPlated* and *TopLayer*, as described in the previous example.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

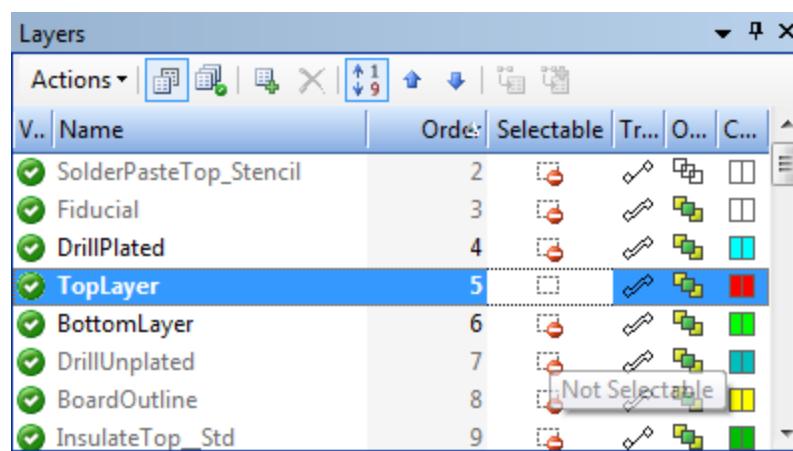
2.2.4. Move Board Outline to Separate Layer

In our sample circuit board **TUTOR**, the outline of the circuit board is located on the solder and component side (electrical layer). It is advisable to place the circuit board outline on a separate mechanical layer.

Let's select the circuit board outline on the component side (*TopLayer*) and move it to the *BoardOutline* layer.

How to select circuit board outline on the TopLayer:

- Double click on the **Selectable** cell of the *TopLayer* entry. All layers, except the *TopLayer*, are made **Not Selectable**. It will be easier to select the outline of the circuit board on the *TopLayer* in the **Graphic** view.

**Figure 47. All layer, excluding TopLayer, are reset to Not selectable mode**

- Click on the upper left corner of the circuit board outline in the **Graphic** view.

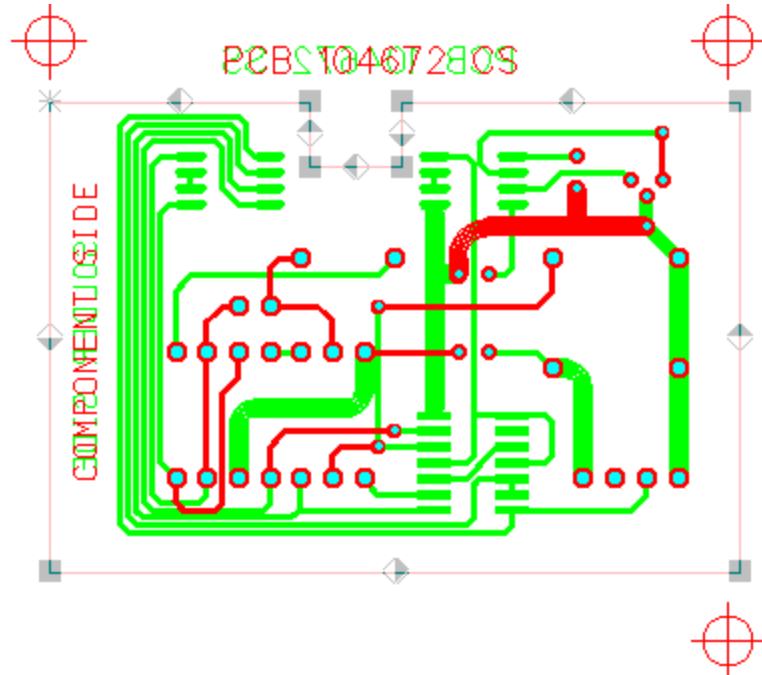


Figure 48. Graphic view with selected circuit board outline



Note: Always click on the corner of the object to select it. Otherwise the nearby object can be selected.

How to move circuit board outline to separate layer:

- Right click with a mouse on the *BoardOutline* layer name in the **Layers** panel. The context sensitive menu appears on the screen.
- Select the **Move to Layer** menu item. After the mouse button is released, the selected circuit board outline is moved from the *TopLayer* to the *BoardOutline* layer.

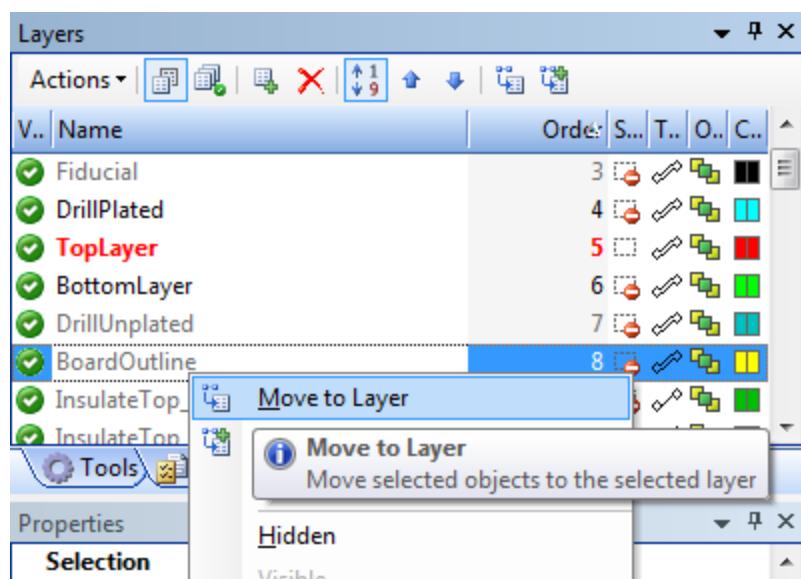


Figure 49. Move to Layer context sensitive menu

Let's remove the circuit board outline from the *BottomLayer* (solder side).

How to remove circuit board outline from the solder side:

- Double click on the **Selectable** cell of the *BottomLayer* entry in the **Layers** panel. The *BottomLayer* is made selectable and rest of layers are switched to **Not selectable** mode.
- Select the circuit board outline on the *BottomLayer* in the **Graphic** view.
- Right click with a mouse and select **Remove** item in the appeared context sensitive menu.
- Click on **All Visible and Selectable** button in the toolbar of the **Layers** panel to make all layers **Selectable**.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

2.2.5. Place Objects on Separate Layers

CircuitCAM does not distinguish between text objects and conductor tracks when they are present on the same layer. We advise you to move text objects to the separate layers.

How to select text objects on component side:

- First of all double click on the **Visible** cell of the *TopLayer* entry in the **Layer** view. Only the component side of the board is visible in the [Graphic view](#).

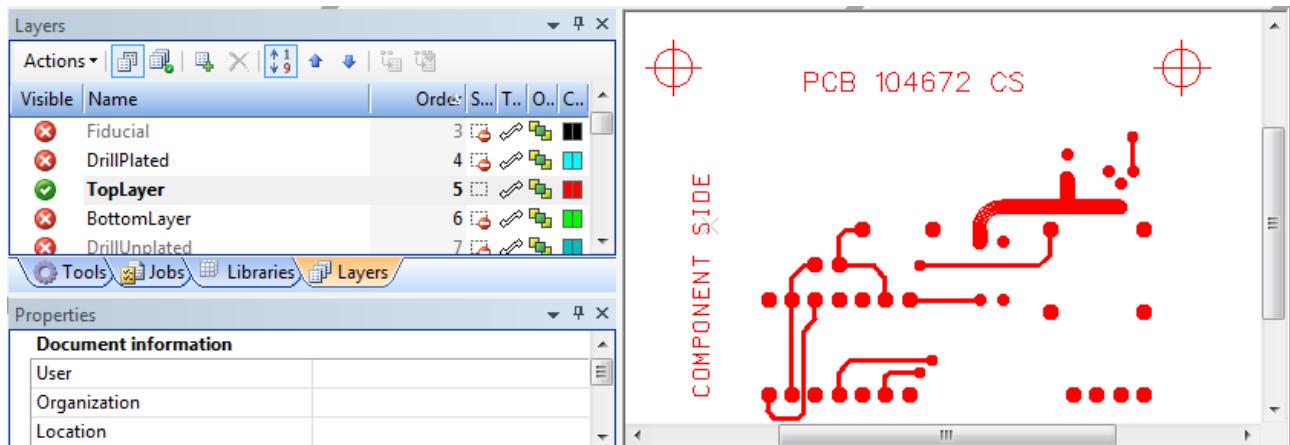


Figure 50. *TopLayer* the only visible layer in Layers list and component side in graphic view

- Check, if the **Plus Selection** mode is switched *On*. If not, click on **Plus Selection** button in the **Select** toolbar.
- To select text objects in **Graphic** view, use the area selection. Point with a mouse on one corner of the supposed rectangle, which surrounds the text objects. Hold and drag to the opposite corner of this supposed rectangle. Release the mouse button. Select another group of text objects.

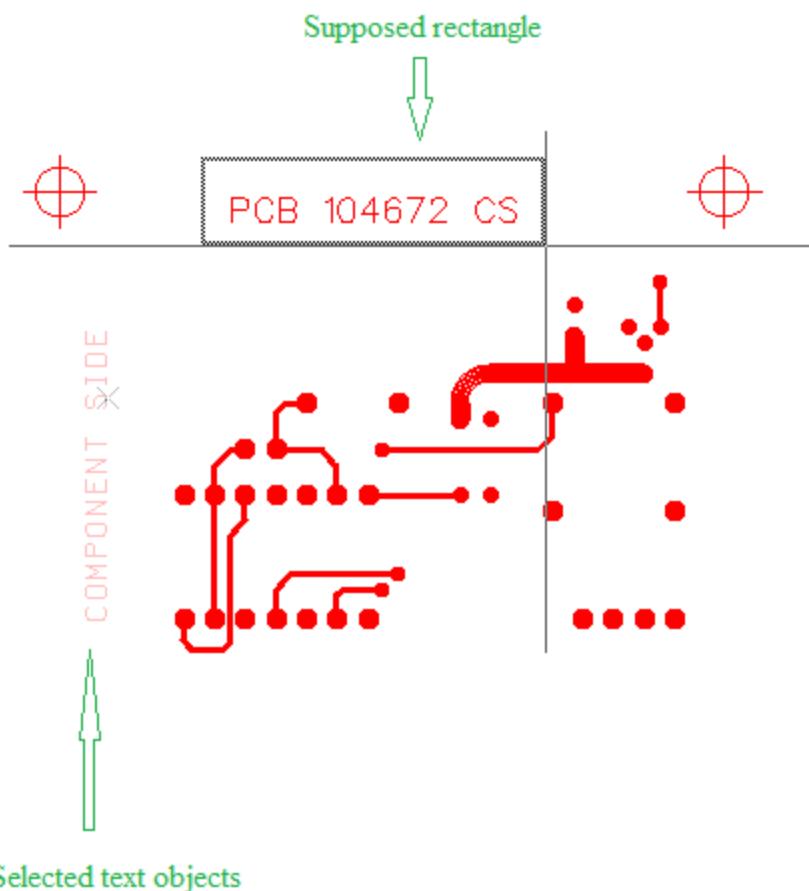


Figure 51. Select text objects on *TopLayer*

Let's move selected text objects on the component side to the *TextTop* layer.

How to place text objects on separate layers:

- Right click on the *TextTop* layer in the **Layers** panel. The context sensitive menu appears on the screen.
- Select the **Move to Layer** item. The text objects are moved from the *TopLayer* to the *TextTop* layer.
- Press on the **Visible** cell in front of the *TextTop* layer name to make this layer visible in the **Graphic** view.

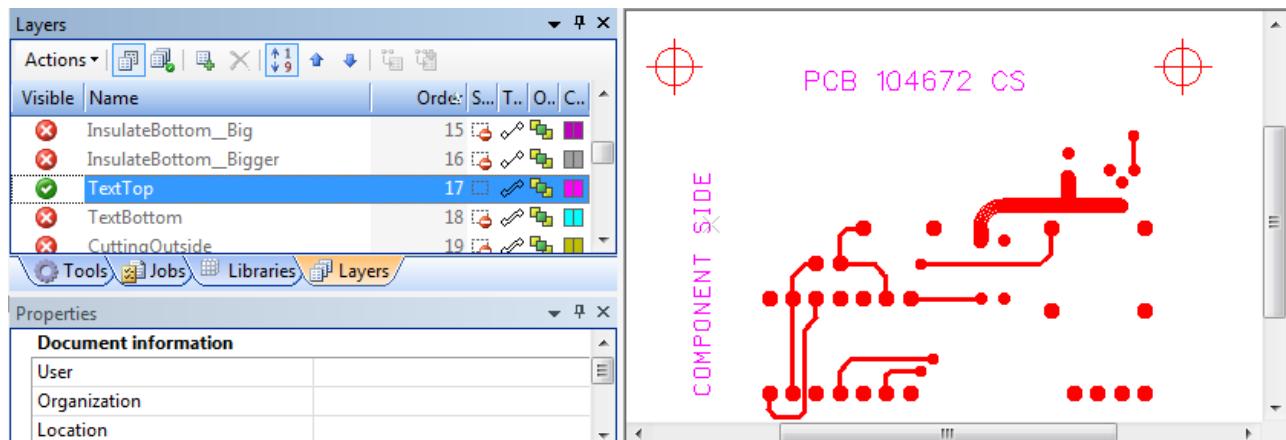


Figure 52. Text objects on *TextTop* layer are presented in Graphic view

Let's move text objects from the *BottomLayer* to *TextBottom* layer:

- Double click on the **Visible** cell of the *BottomLayer* entry in the **Layer** view. Only the solder side of the board is visible in the **Graphic** view.

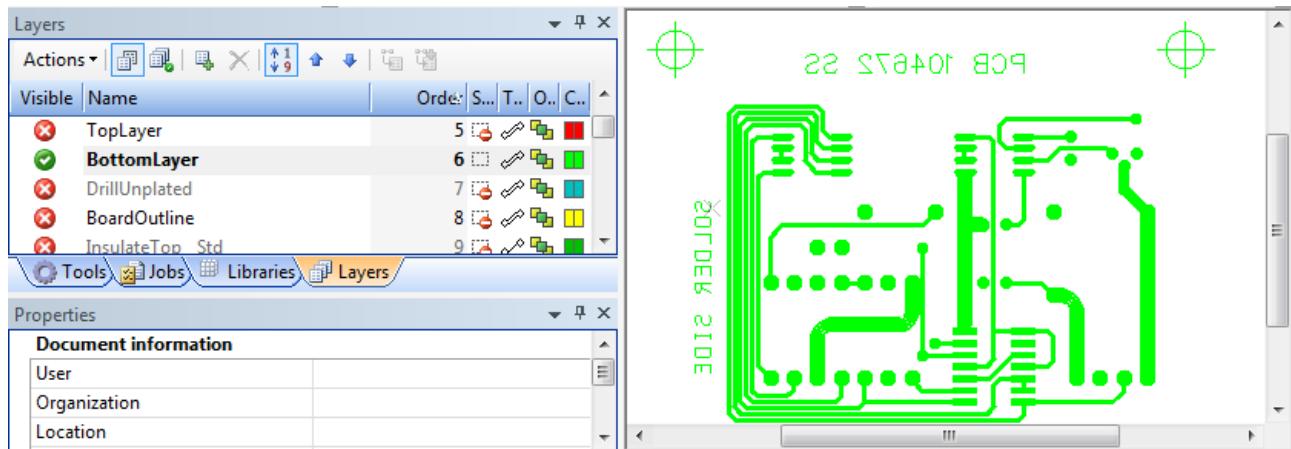


Figure 53. Bottom layer components in Graphic view

- Select text objects in the **Graphic** view.
- Right click on the *TextBottom* layer name in the **Layers** panel. The context sensitive menu appears on the screen.
- Select **Move to Layer** item. The text objects are moved from the *BottomLayer* to the *TextBottom* layer.
- Press on the **Visible** cell in front of the *TextBottom* layer name to make this layer visible in the **Graphic** view.

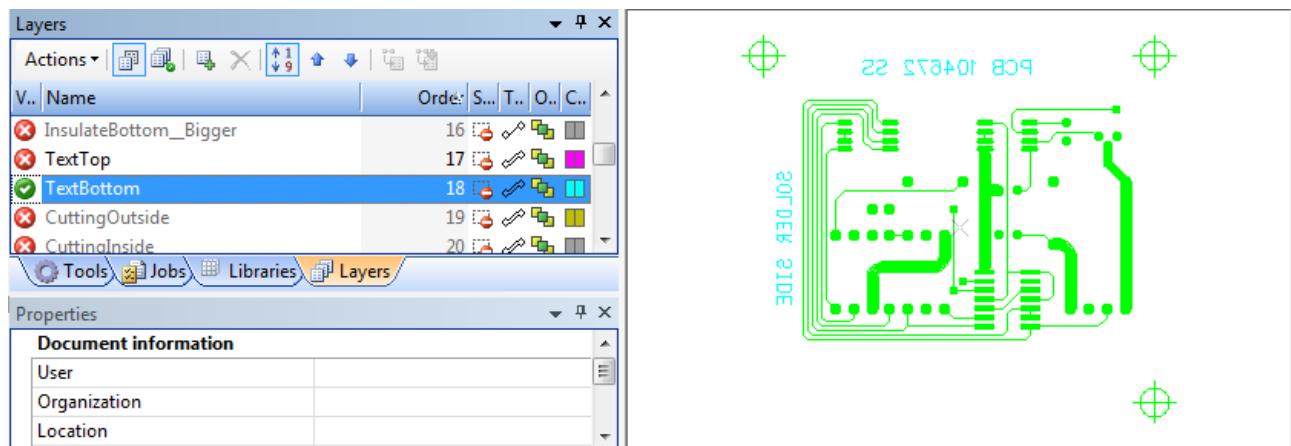


Figure 54. Text objects on *TextBottom* layer are presented in Graphic view

- Click on the **All Visible and Selectable** button in the **Layers** panel tool bar. All circuit board components appear in the **Graphic** view.

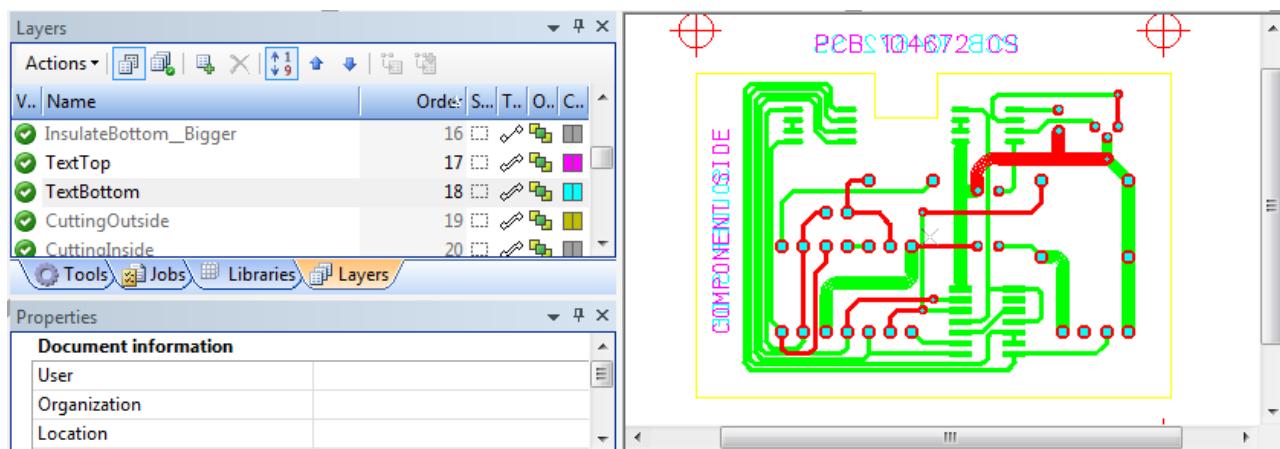


Figure 55. Circuit board presentation in Graphic view

Move fastening holes from an electrical layer (i.e. "TopLayer") to a mechanical layer (i.e. "DrillUnplated").

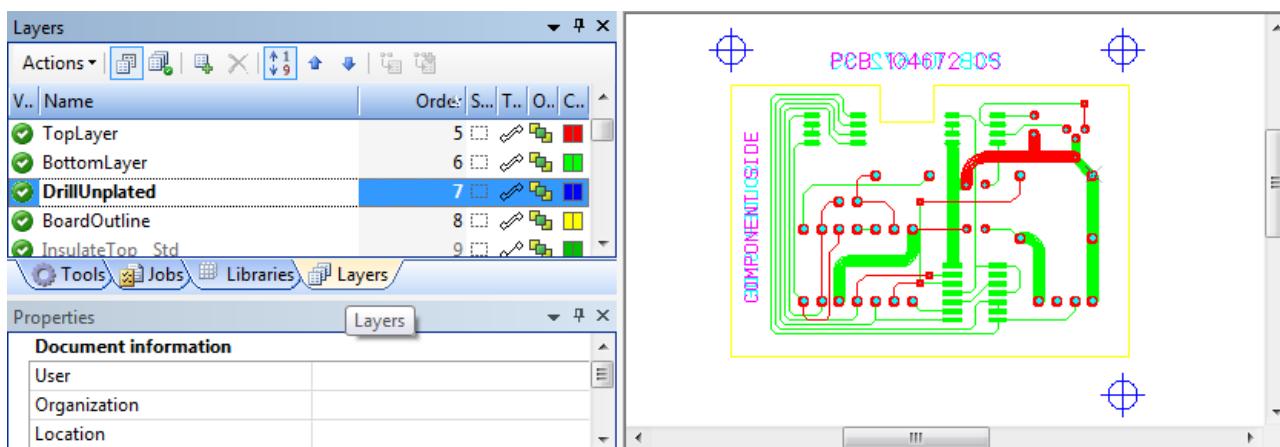


Figure 56. Fastening holes on DrillUnplated layer (dark blue color in Graphic view)



Note: Save your data under the **Tutor2.cam** name.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

2.3. Edit SMT Stencils

2.3.1. GMC Tutorial

2.3.2. Package Library Tutorial

2.3.3. Create Package Tutorial

2.4. Generating NC Data

2.4.1. Create Milling Contour

In this example, we will introduce how to use the contour generator, and insert breakout tabs to the printed circuit board milling contour. In addition we will explain, how to change a milling direction and sequence.

How to create milling contour:

Lets examine how to calculate the milling contour for the circuit board.

- Select in the main menu **File -> Open**.
- Select the **Tutor1.cam** file.
- Select an outline of the circle board in the **Graphic** view. In our example the circuit board outline is located in the *BoardOutline* layer.

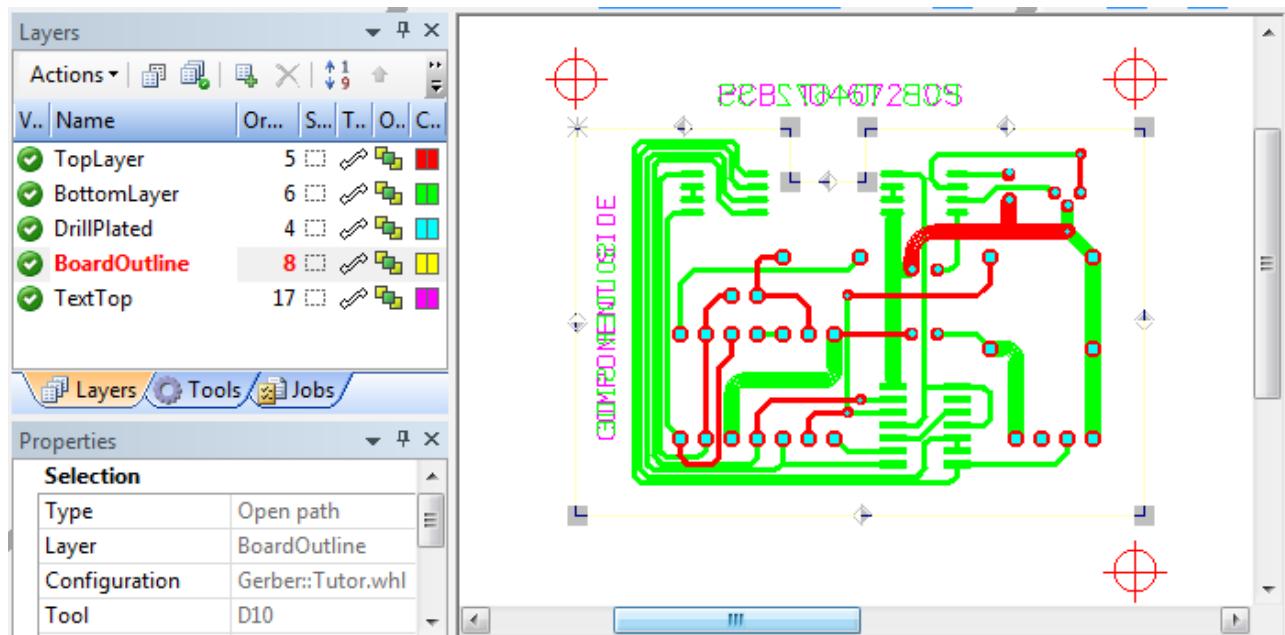


Figure 57. Selected circuit board outline in Graphic view and highlighted *BoardOutline* layer in Layers panel

- Select in the main menu **Tool Path -> Contour Routing**. The **Contour Routing** dialogue window appear son the screen.

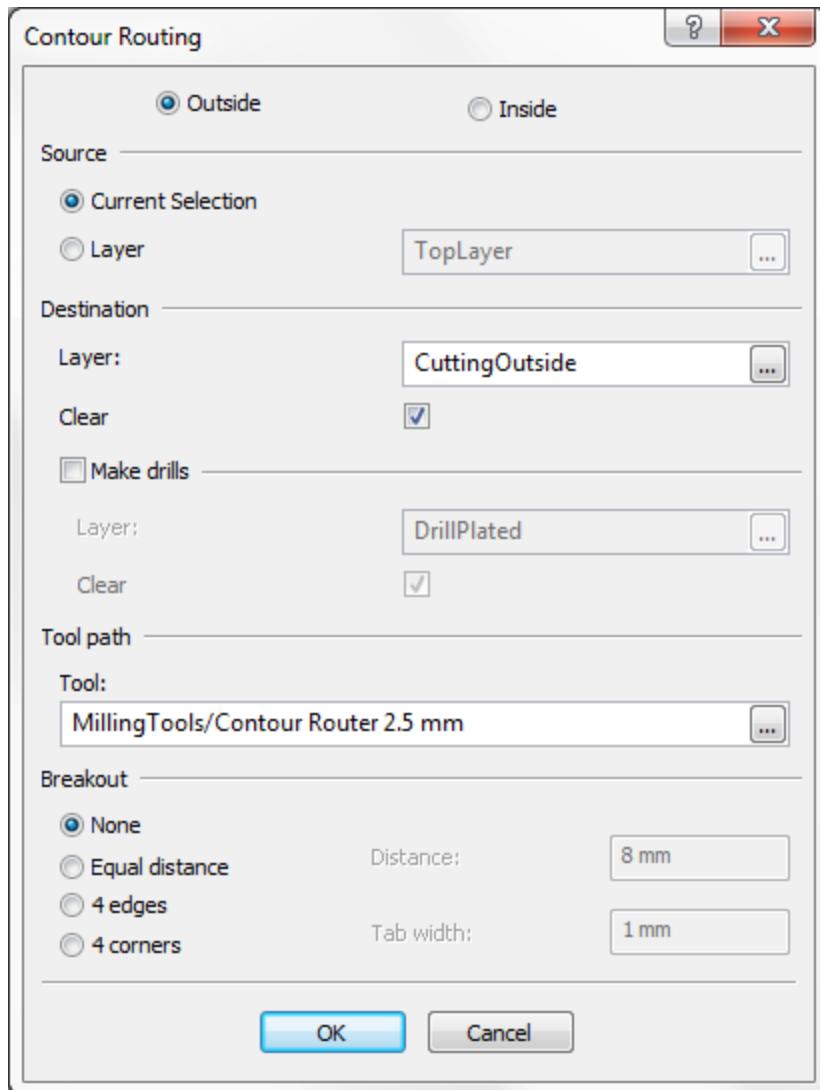


Figure 58. Contour routing dialogue window

- The **Outside** selection button is selected, which defines the milling area.
- The source of the milling contour is the **Current selection**.



Note: In case you want to change the source layer, click the **Layer** selection button and select an applicable layer in the list box.

- The destination layer *CuttingOutside* is pre-selected in the **Destination Layer** list box. Turn the **Clear** option *On*.
- Specify the used tool for milling in the **Tool path** layout:
 - Click the "..." button next to the Tool field. The **Tools** dialogue window appears on the screen.
 - Expand the **LMD** branch and select the tool **End Mill 2.0 mm (79 mil)** under the **MillingTools** branch.
 - Click on **OK**.
- You can define the location and size of breakout tabs in the **Breakout** layout. We will skip this for now and later we will introduce how to create breakout tabs in an applicable places.
- Click on the **OK** button to generate the milling contour. After the calculation is completed, the milling contour is displayed in the **Graphic** view.

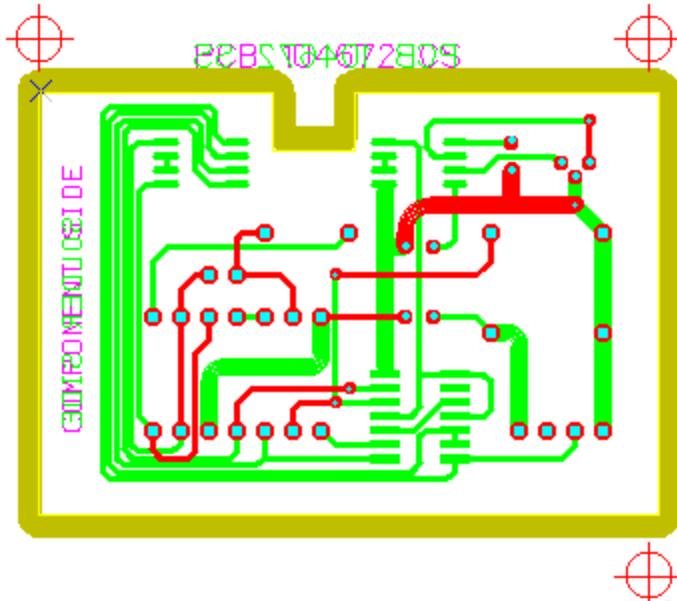


Figure 59. Milling contour in Graphic view



Note: The circuit board will be milled out of the board with the current milling contour.

Insert Breakout Tabs

How to insert breakout tabs in milling contour:

- Select the contour on the *CuttingOutside* layer in the **Graphic** view.

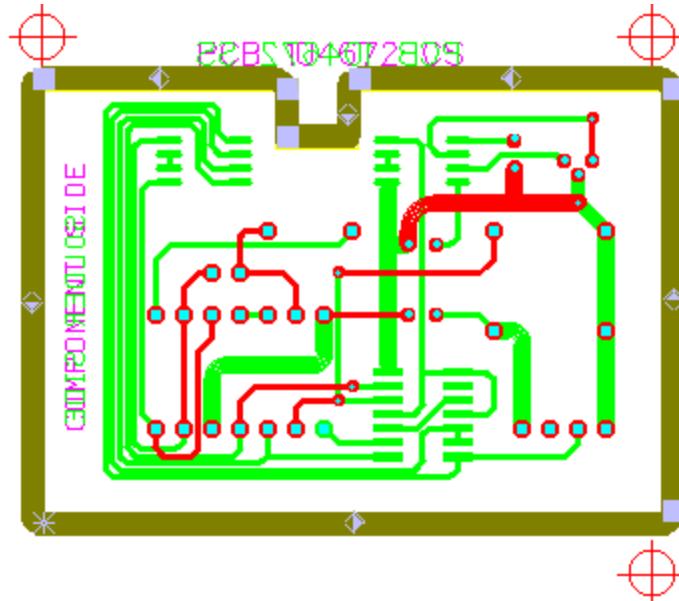


Figure 60. Contour selected in Graphic view

- Set an * anchor point using + and - keys or click with a mouse on an applicable marker point to set an anchor point.

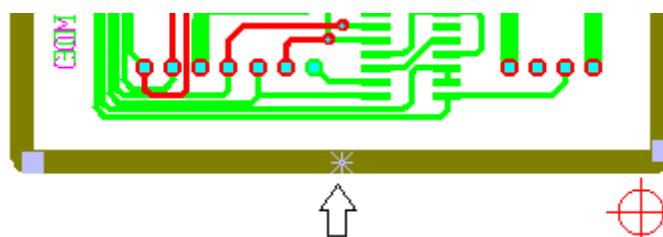


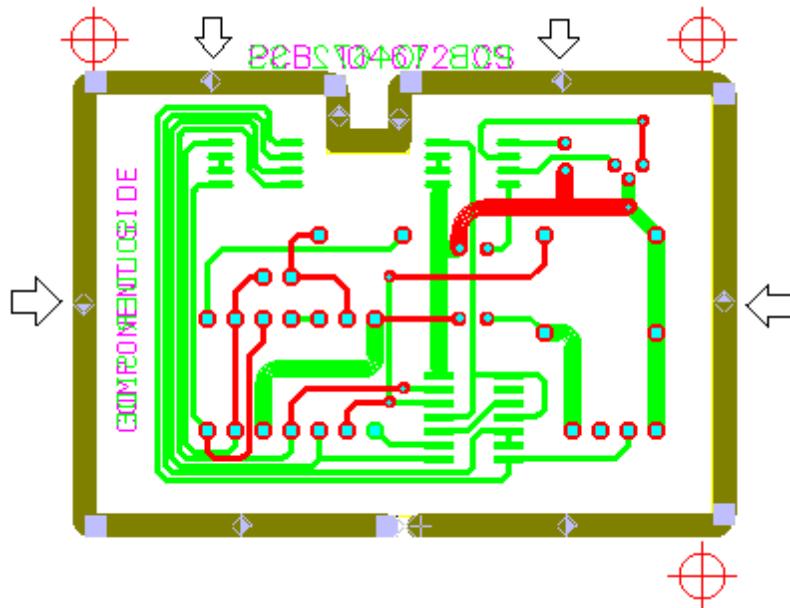
Figure 61. Setting anchor point

- Select in the main menu **Tool Path -> Create Gap** or use keyboard keys **Ctrl + G**.

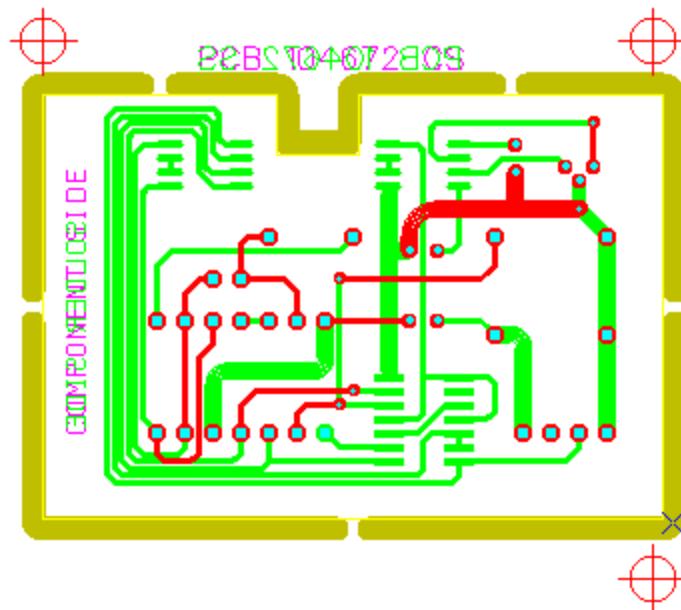


Note: The size of a breakout tab can be changed in the General Settings ---> **Diverse page, Breakout Tab** field.

- Repeat the previous steps to insert breakout tabs at particular positions (see figure below).

**Figure 62. Positions of anchor point for breakout tabs**

You can see the milling contour with breakout tabs in the **Graphic** view. The circuit board is connected to the board material via breakout tabs.

**Figure 63. Contour with breakout tabs**

After you have generated a milling contour and inserted breakout tabs, you can define a milling sequence and direction.

Change Milling Direction and Sequence

How to change milling direction and sequence:

- Select a closed path with a left mouse button in the **Graphic** view. The sequence number of the selected path is displayed in the status bar.

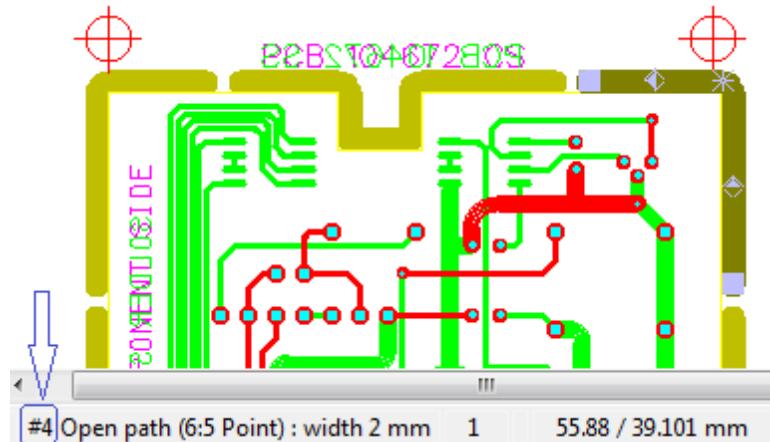


Figure 64. Sequence number of selected path in Status bar

- Select in the main menu **Tool Path -> Swap**. The submenu appears on the screen.
- Select the **Move to First** menu item to define the selected path to be cut first.



Note: To cut the selected path last, select the **Move to Last** menu item. To move the selected path one step forwards/backwards, select the **Move to Next/Prev** menu item.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

2.4.2. Create Base Material and Fixture Holes

2.4.3. Create Rubout Area

A **rubout area** is a zone where the isolation channels around conductor tracks and pads are milled, and also all residual copper between the conductor tracks and pads is removed from the circuit board.

How to create rubout area:

- Open **Tutor2.cam** document file.
- Select in the main menu **Insert -> Rubout Area**. The submenu appears on the screen with available values: **Bottom Layer**, **Top Layer** and **All Layers**.
- Select the **Top Layer** menu item to create a rectangular rubout area on the *RuboutTop* layer.
- Click in the **Graphic** view are to set the start point for a rectangle.
- Move the cursor down and to the right.
- Click again to set the end point for the rectangle.
- Press **Esc** key to switch *Off* insert mode.

The rubout area is defined on the *RuboutTop* layer in the **Graphic** view.

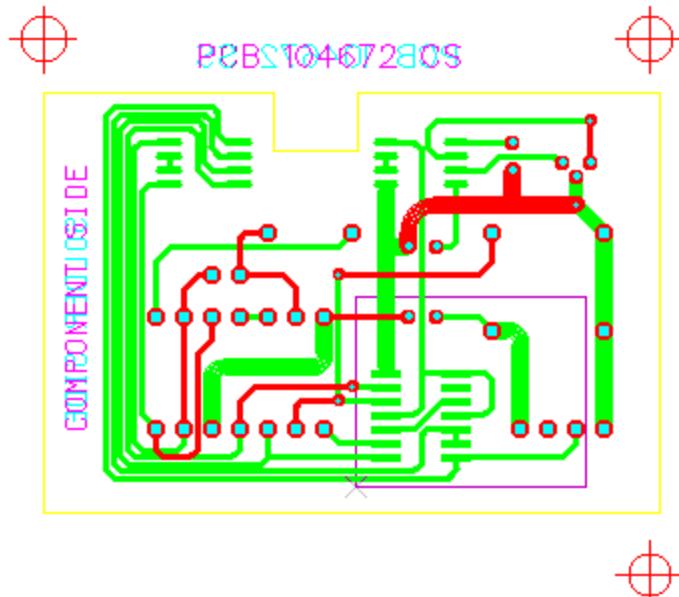


Figure 65. Defined rubout rectangle area in Graphic view



Note: You can insert a different shape for the rubout area (polygon, circle, closed path) on the rubout layer via the **Insert** toolbar.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

2.4.4. Insulate Milling Tutorial

2.4.5. Customize Insulation and Rubout Job

CircuitCAM presents a list of pre-configured jobs in the **Jobs** view.

Let's examine how to calculate insulation channels and remove residual copper from the rubout area.

In the previous example we created a rubout are in the *Tutor2.cam* document. Let's proceed the work.

How to run Insulate and Rubout job:

- Click on the **Jobs** tab to open the **Jobs** view.
- Expand the **Scripts** tree branch.
- Select with your mouse the *Insulate_Top_Rubout_Bigger_Big_Primary* job.
- Click on the ➤ **Run** button in the toolbar of the **Jobs** view.

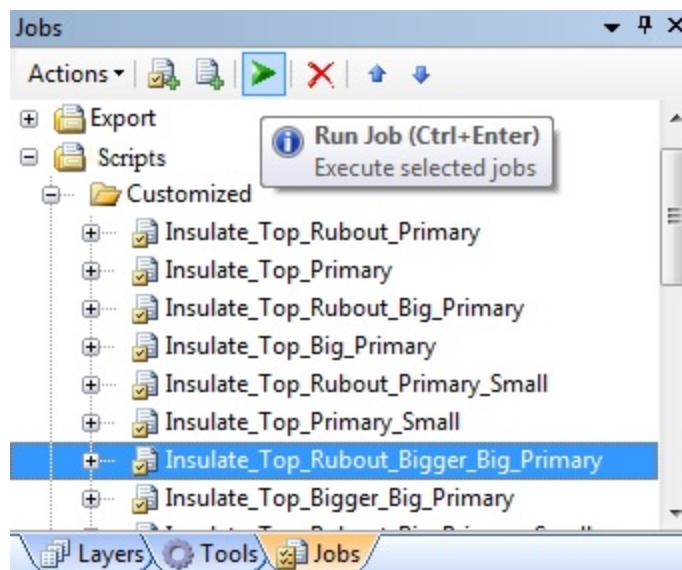


Figure 66. Jobs view, Run selected job

- The **Run job** dialogue window appears on the screen to set parameters for the job.

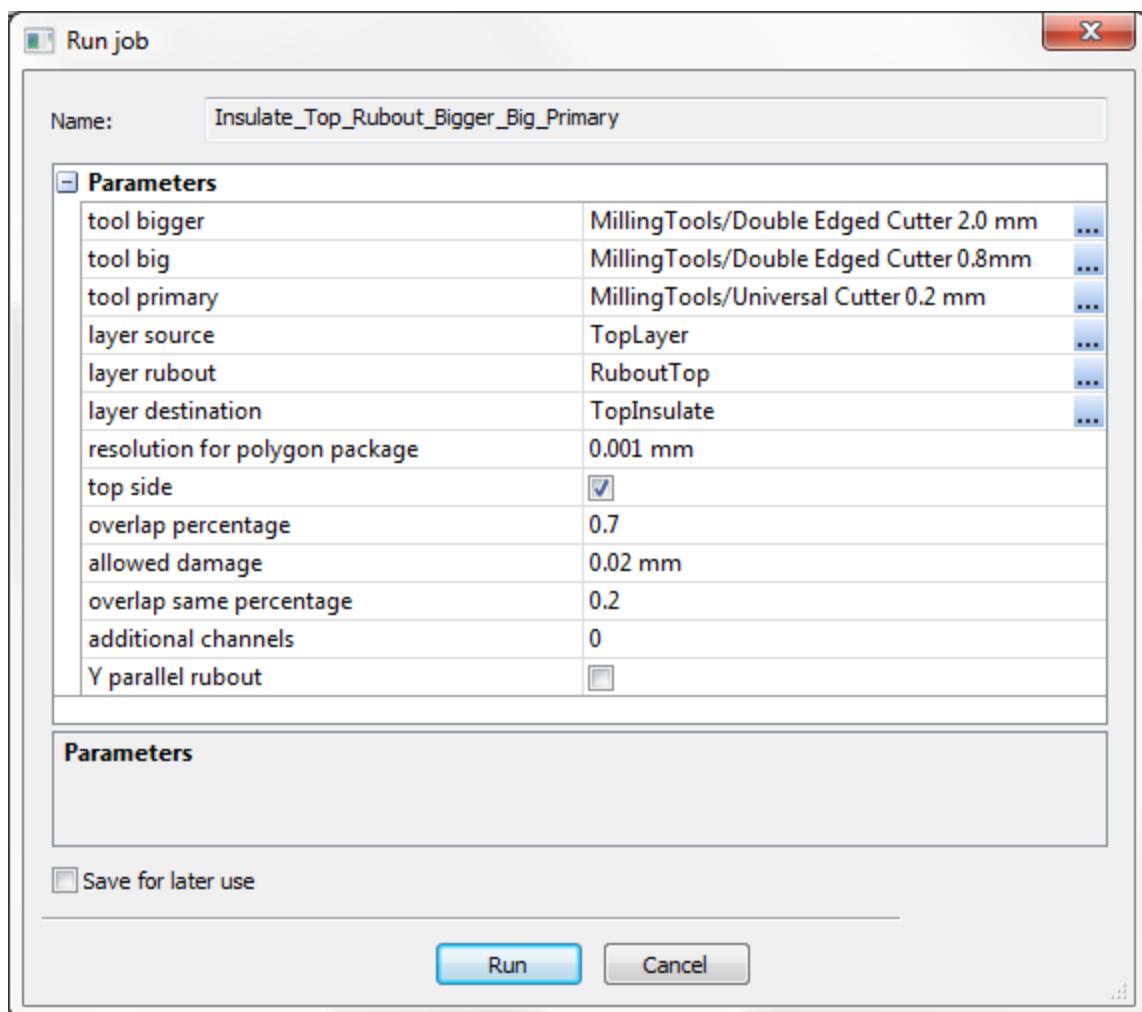


Figure 67. Run job dialogue window

- Click on the "..." button next to the **tool bigger** field. The **Tools** dialogue window appears on the screen to select an applicable tool. The currently selected tool is highlighted in the tree. Click on the **Cancel** button to close the window.



Note: You can select **tool big** and **tool primary** in the same way.

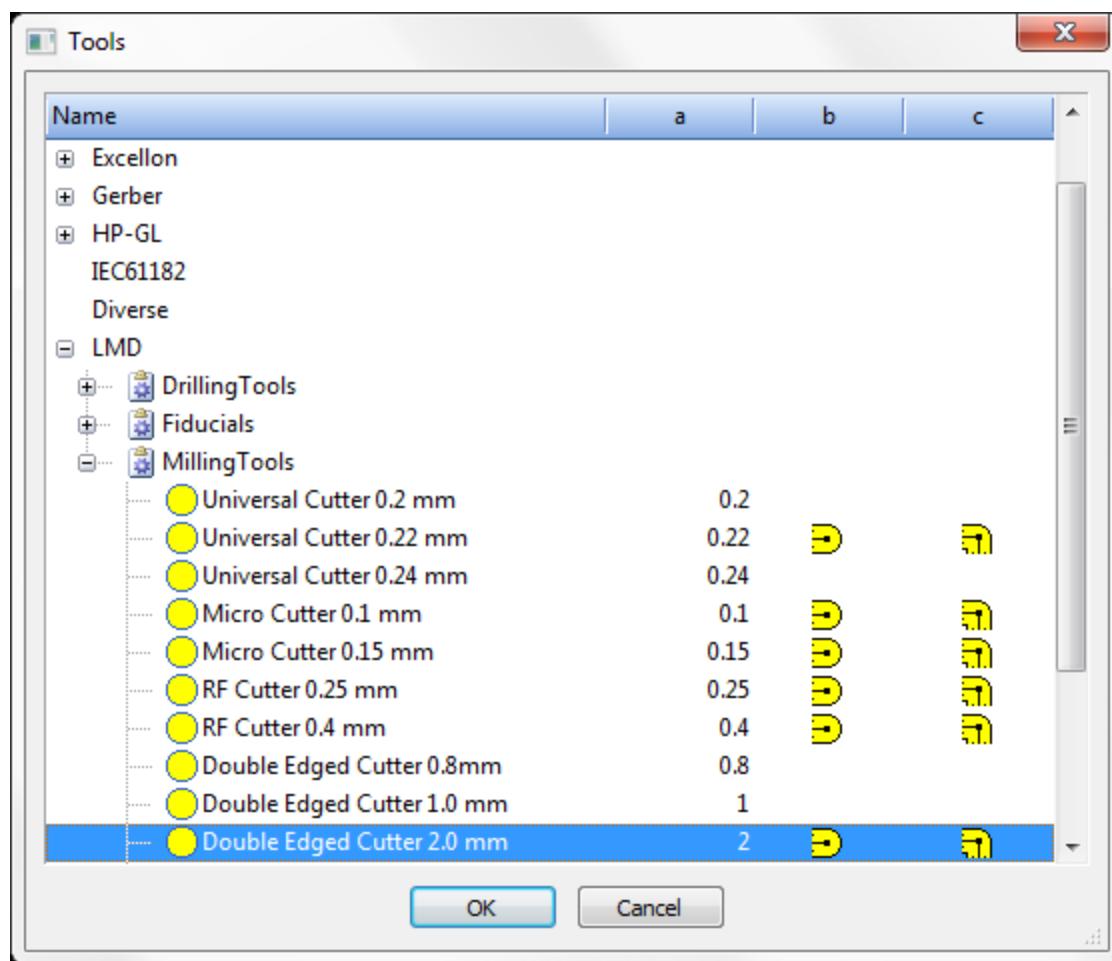


Figure 68. Tools dialogue window

- Click on the "..." button next to the **layer source** field. The **Layers** dialogue window appears on the screen to select an applicable source layer. The currently selected *TopLayer* is highlighted in the list. Click on the **Cancel** button to close the window.



Note: You can select **layer rubout** and **layer destination** in the same way.

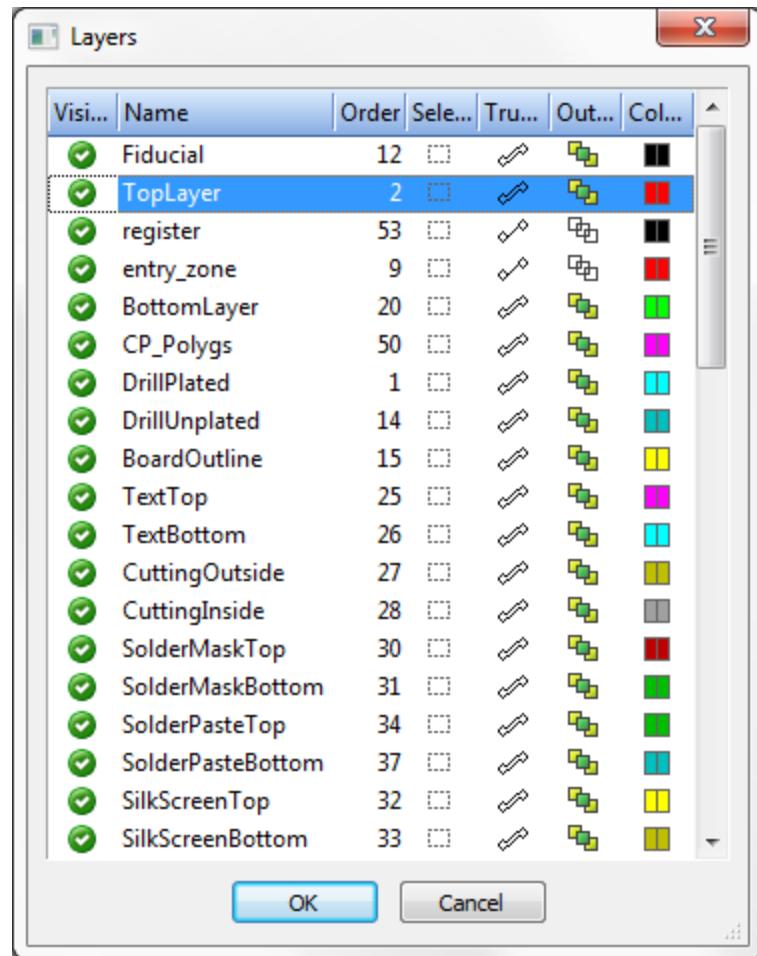


Figure 69. Layers dialogue window

- Click on the "..." button next to the **layer rubout** field. The **Layers** dialogue window appears on the screen to select an applicable source layer. The currently selected *RuboutTop* layer is highlighted in the list. Click on the **Cancel** button to close the window.
-  **Note:** You can remove the layer from the layer rubout field. In this case, the rubout function is excluded from the job.
- Click on the **Run** button on the bottom of the **Run job** dialogue window. The insulation is done for a whole graphical object and the rubout are is cleared.

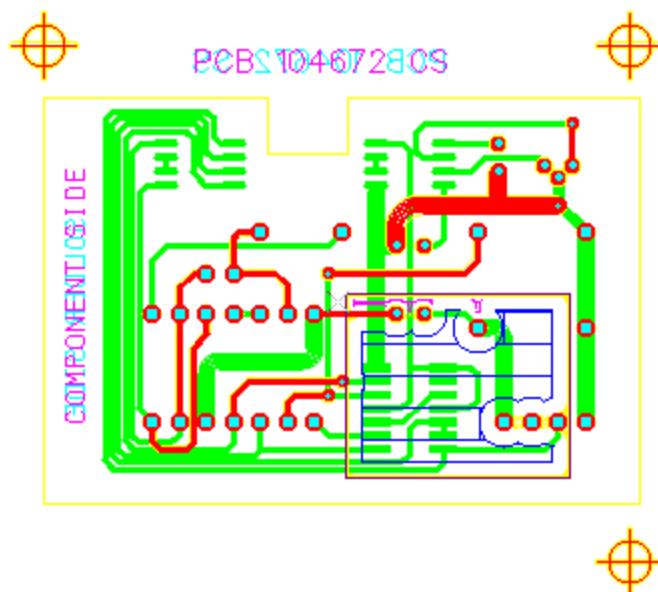
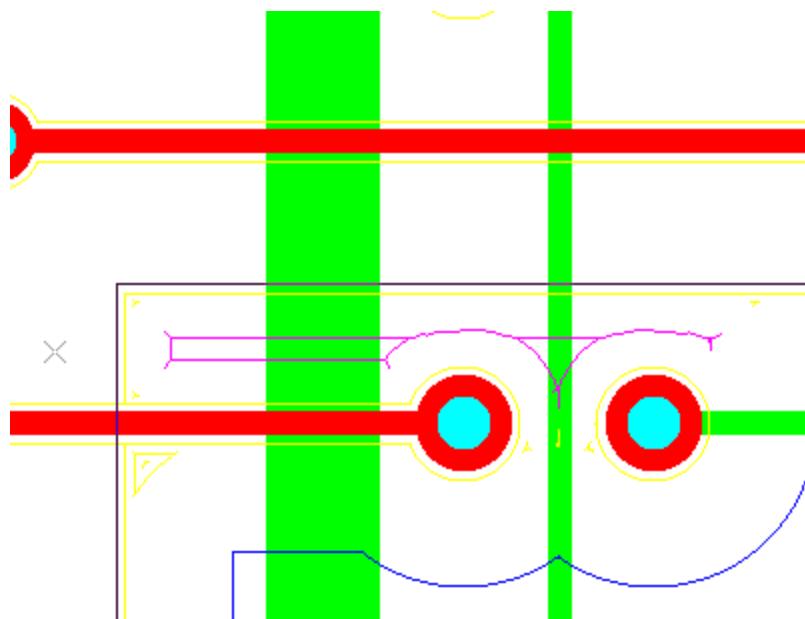


Figure 70. Insulation and rubout job result in Graphic view

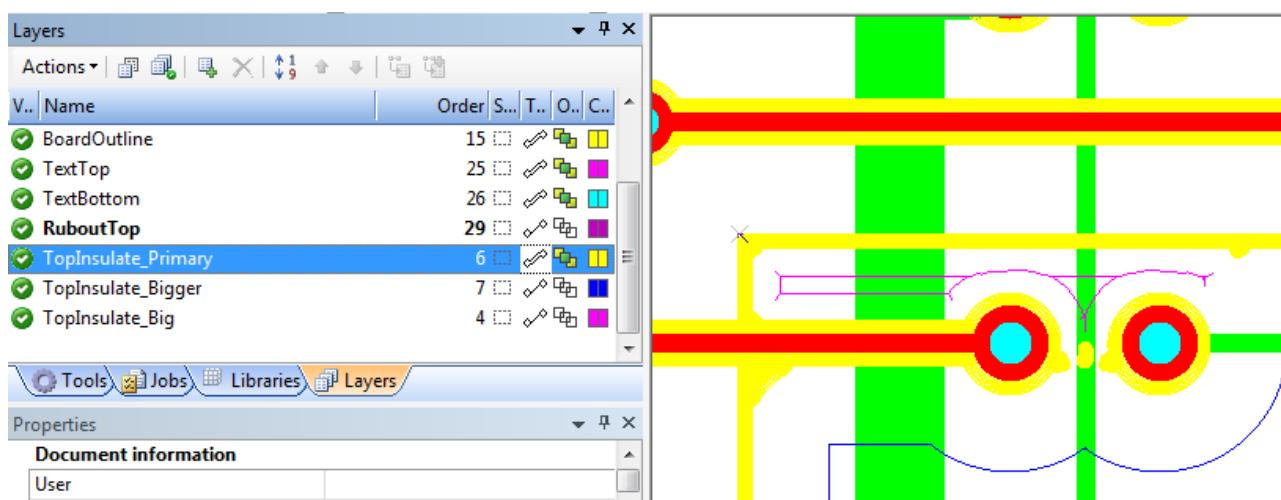
To view results, use the **Zoom Window** function, change the presentation of the layer.

- Click on the **Zoom Window** button in the **Zoom** toolbar.
- Draw with a mouse a rectangle in the **Graphic** view. The selected window is zoomed in the **Graphic** view.

**Figure 71. Zoomed window in Graphic view**

Let's change the insulation channel, made by a primary tool to be of the **True Width** and **Filled**. The current presentation is **Line Only** and **Outline**:

- Click on the **Layers** tab to open the **Layers** view.
- Select *TopInsulate_Primary* layer.
- Set **True Width** and **Fill** options *On*.

**Figure 72. TopInsulate_Primary layer insulation channels of true width and filled (yellow color)**

2.4.6. Laser Fill Tutorial

This functionality ensures that the defined area is completely exposed by a laser beam with a given diameter. In all cases a laser path, called "primary outline" is put along the outline of the area. Then the remaining area is then ISO filled by horizontal or vertical laser paths, these paths are either hatching or concentric.

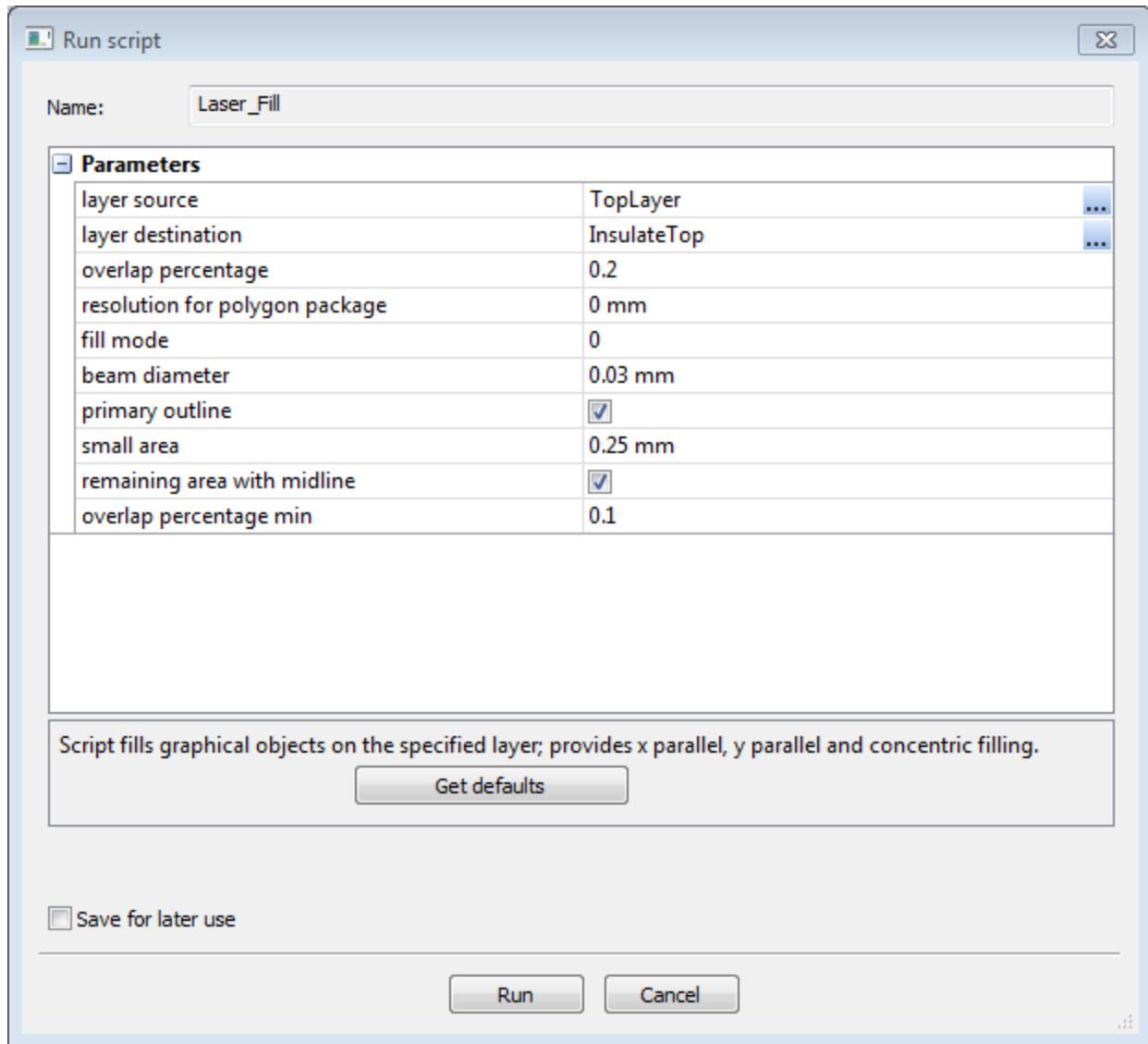


Figure 73. Laser Fill run job dialogue

Laser Fill job parameters

- **Layer source:** Specifies the layer where the graphical objects will be etched out of copper.
- **Layer destination:** The output layer where the fill lines are applied. This layer name will be combined to create two new layers; "xxx_FillO" and "xxx_FillX"
- **Overlap percentage:** Specifies the overlap between two fill lines, the value is a percentage of the beam diameter.
- **Resolution for polygon package:** Specifies the grid to be used.
- **Fill mode:** Specifies the strategy to be used in filling, either, "0: x-parallel", "1: y-parallel", or, "2: concentric".

- **Beam diameter:** Specifies the diameter of the laser beam to be used.
- **Primary outline:** If this box is checked, creates an outline path.
- **Small area:** If a fill area is smaller than the number specified here, concentric fill will be applied.
- **Remaining area with midline:** If this box is checked, midlines will be generated for the unfilled remaining areas.
- **Overlap percentage min:** Specifies the overlap between two filling lines. It is used only with the concentric fill mode. This value is a percentage from the beam diameter.

Example

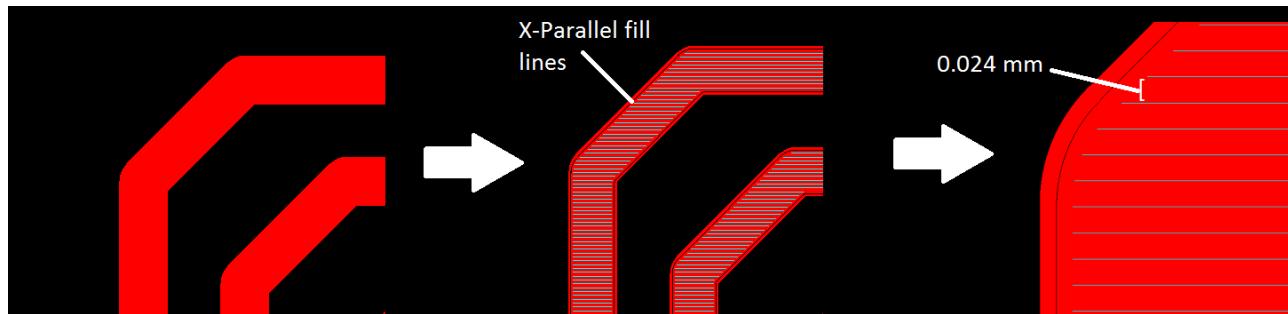


Figure 74. Running the Laser Fill script with default parameters

In the figure above is the results of **Laser Fill** script with the default parameters, the **Fill Mode** is set to **0**, which means **x-parallel** fill lines.

The **Laser Beam Diameter** is set to **0.03 mm**, and the **Overlap Percentage** is set to **20%**, thus the spacing between fill lines is **0.024 mm** (**0.03 minus 20% overlap**).

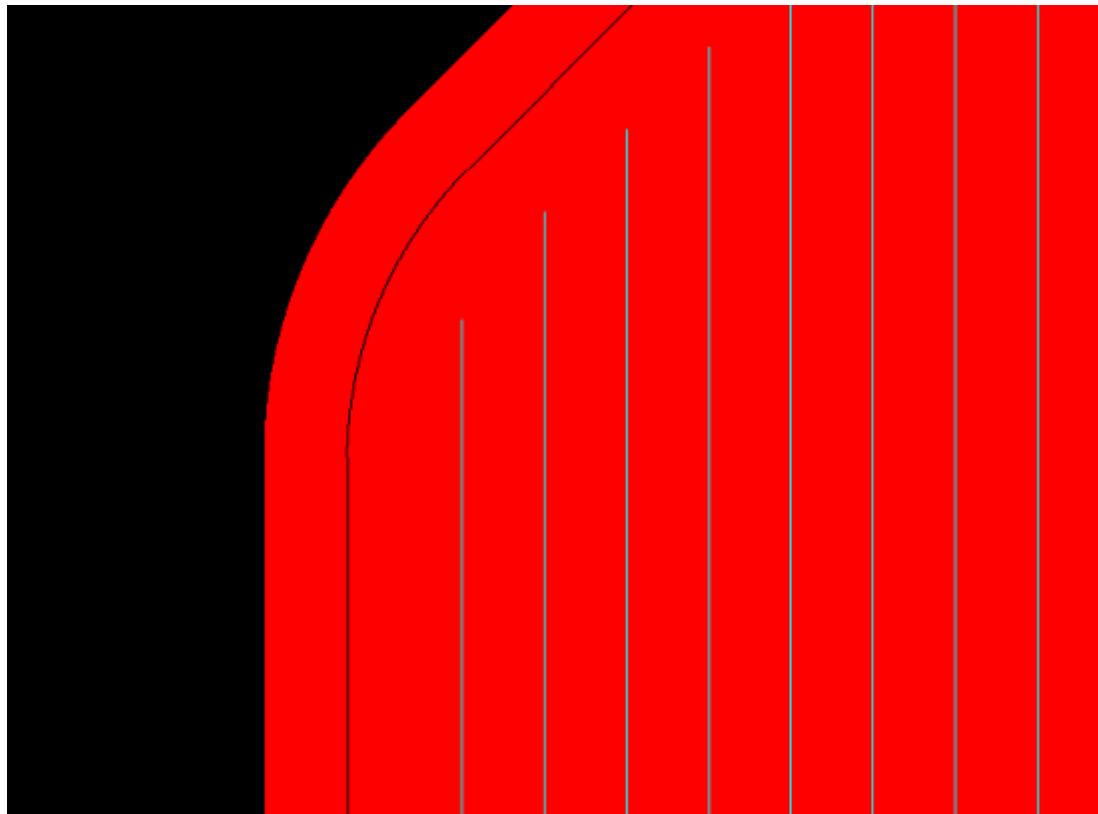


Figure 75. Running the Laser Fill script; with Y-Parallel Lines

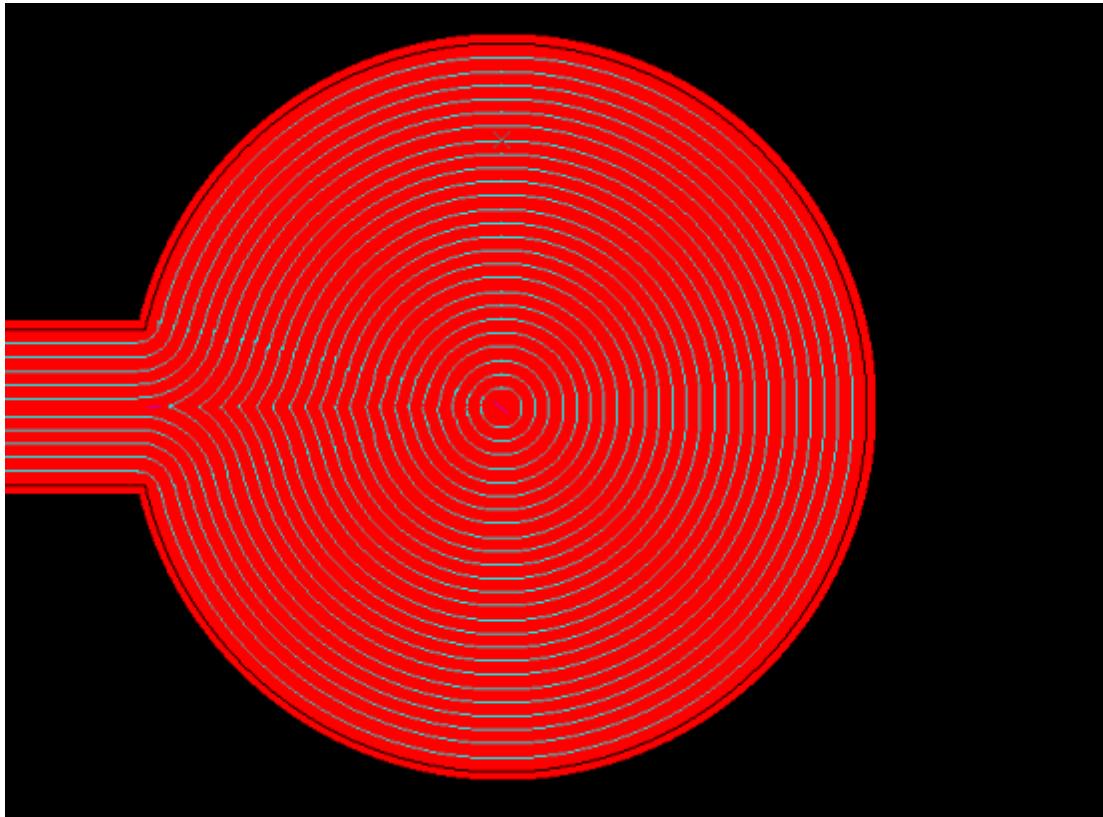


Figure 76. Running the Laser Fill script; with Concentric Lines

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

2.4.7. Laser Insulate Tutorial

CircuitCAM provides functionality to generate insulation paths using laser beams.

this functionality is intended to remove material outside of specified areas, the specified areas may either be insulated from the surrounding material, or the surrounding material may be completely removed.

In all cases a primary insulation channel is created around the specified material, taking into account the diameter of the laser beam generated, additional insulation channels can be generated with a specified overlap value, the remaining area is removed by horizontal or vertical hatching lines.

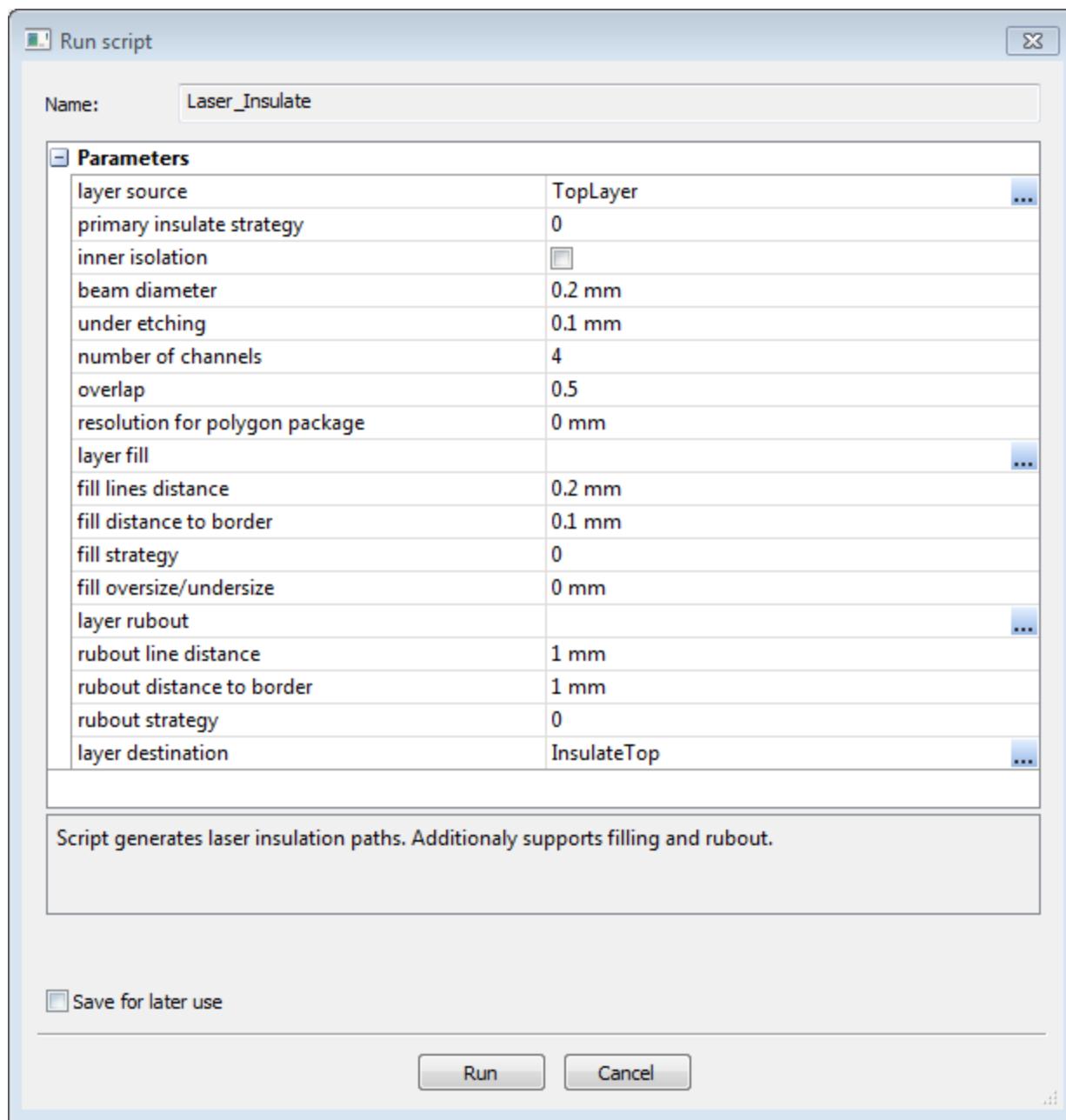


Figure 77. Laser Insulate run job dialogue

Laser Insulate job parameters

- **Layer source:** Specifies the layer to be insulated. The selected layer must exist and can not be empty.
- **Primary insulate strategy:** Specifies the strategy to be followed in the insulation process, either "0 - Simple", "1 - Simple with midline", or "2 - Independent oversize".
- **Inner isolation:** If this box is checked, an inner isolation channel is created.
- **Beam diameter:** Specifies the diameter of the laser beam to be used.
- **Under etching:** Specifies an additional distance between the first insulation channel and the border of area.
- **Number of channels:** Specifies the number of insulation channels. This number of channels is generated in addition with the primary insulation channel.
- **Overlap:** Specifies the overlap between insulation channels, in case of using only one insulation channel, the overlap value has no effect.
- **Resolution for polygon package:** Specifies the grid to be used.

- **Layer fill:** The graphical objects on this selected layer will be filled.
- **Fill lines distance:** Specifies the distance between x, y, or, x+y filling lines.
- **Fill distance to border:** Specifies the distance from the border of the area to the outline closed curve.
- **Fill Strategy:** Specifies the strategy to be used in filling, either, "0: x-parallel", "1: y-parallel", or, "2: x+y-parallel".
- **Fill oversize/undersize:** Specifies the overlap between the outline, and start/end points of hatch lines.
- **Layer rubout:** Graphical objects on this layer will define additional areas to be cleared by the isolation process.
- **Rubout line distance:** Specifies the distance between x, y, or, x+y hatch lines.
- **Rubout distance to border:** Specifies the distance from the border of the area to the start/end points of hatch lines. If the value is set to zero, the hatch lines start and end on the border of the area.
- **Rubout strategy:** Specifies the strategy to be used in creating the rubout area, either, "x-parallel", "y-parallel", or, "x+y-parallel".
- **Layer destination:** Specifies the main layer for isolation tracks, rubout, and filling lines. Additional layers are created in the same name with different endings to clarify various isolation layers.

Examples

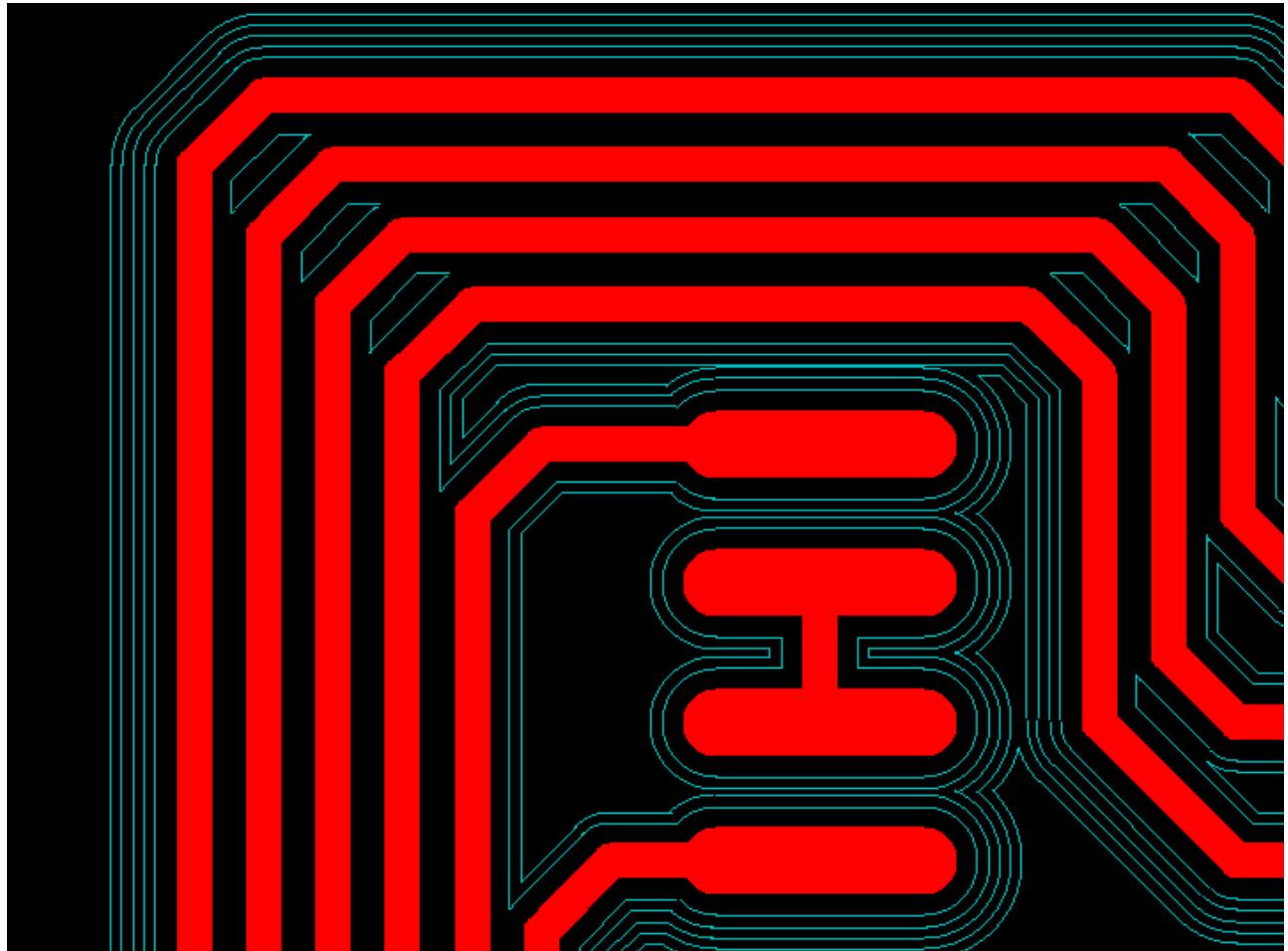


Figure 78. Laser Insulate Script with default values

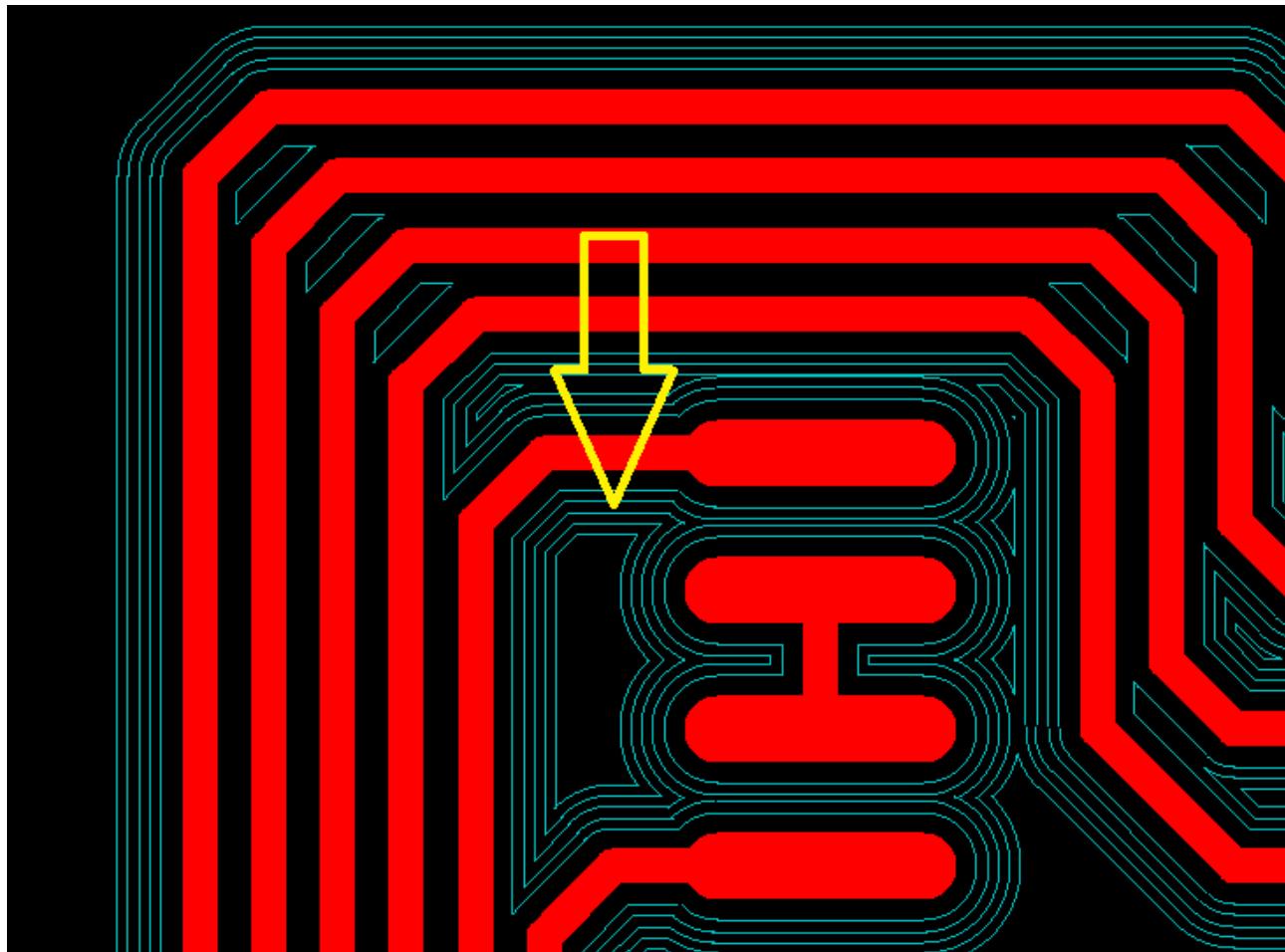


Figure 79. Laser Insulate Script with inner isolation flag on

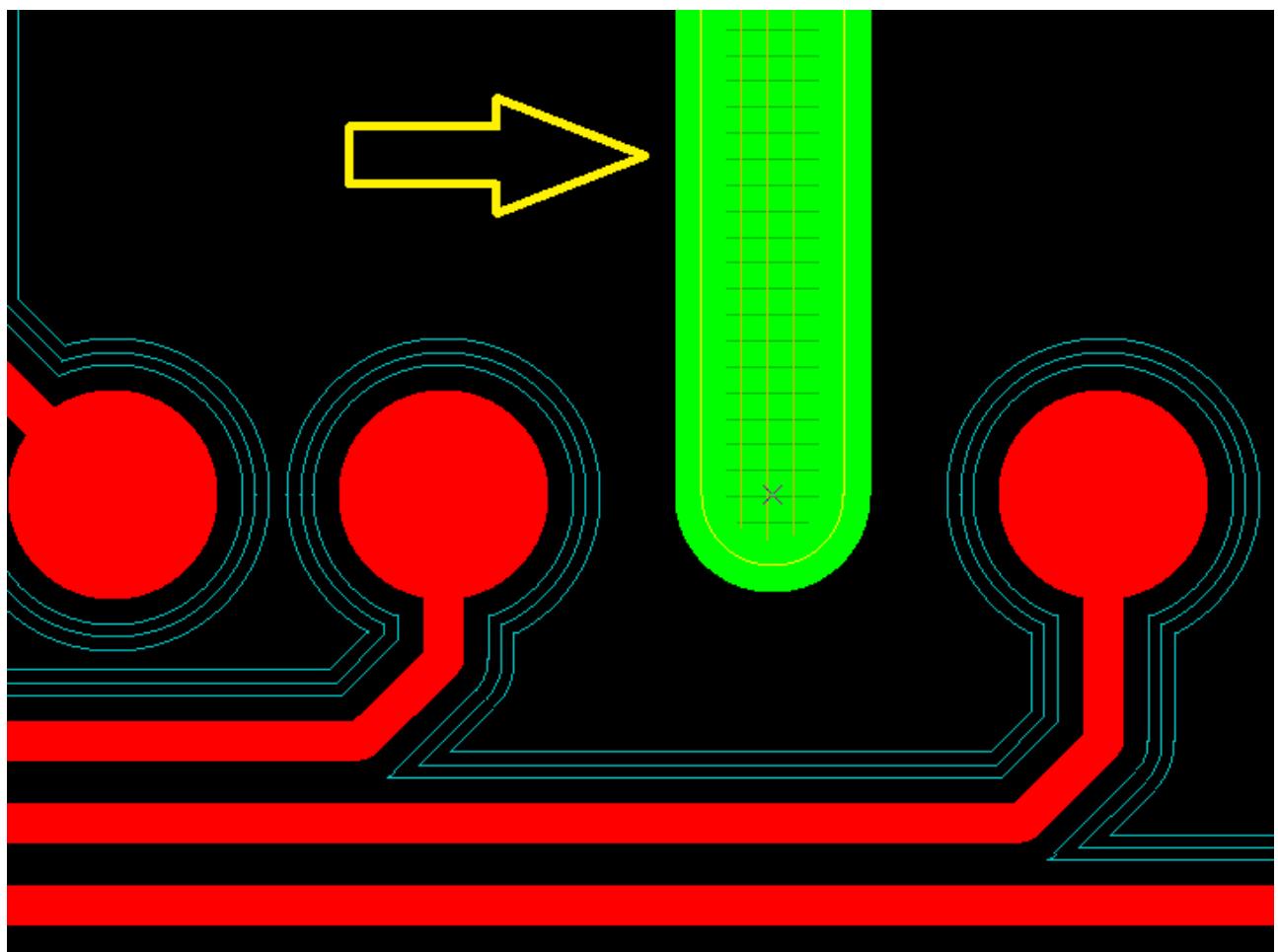


Figure 80. Laser Insulation Script with a defined layer fill

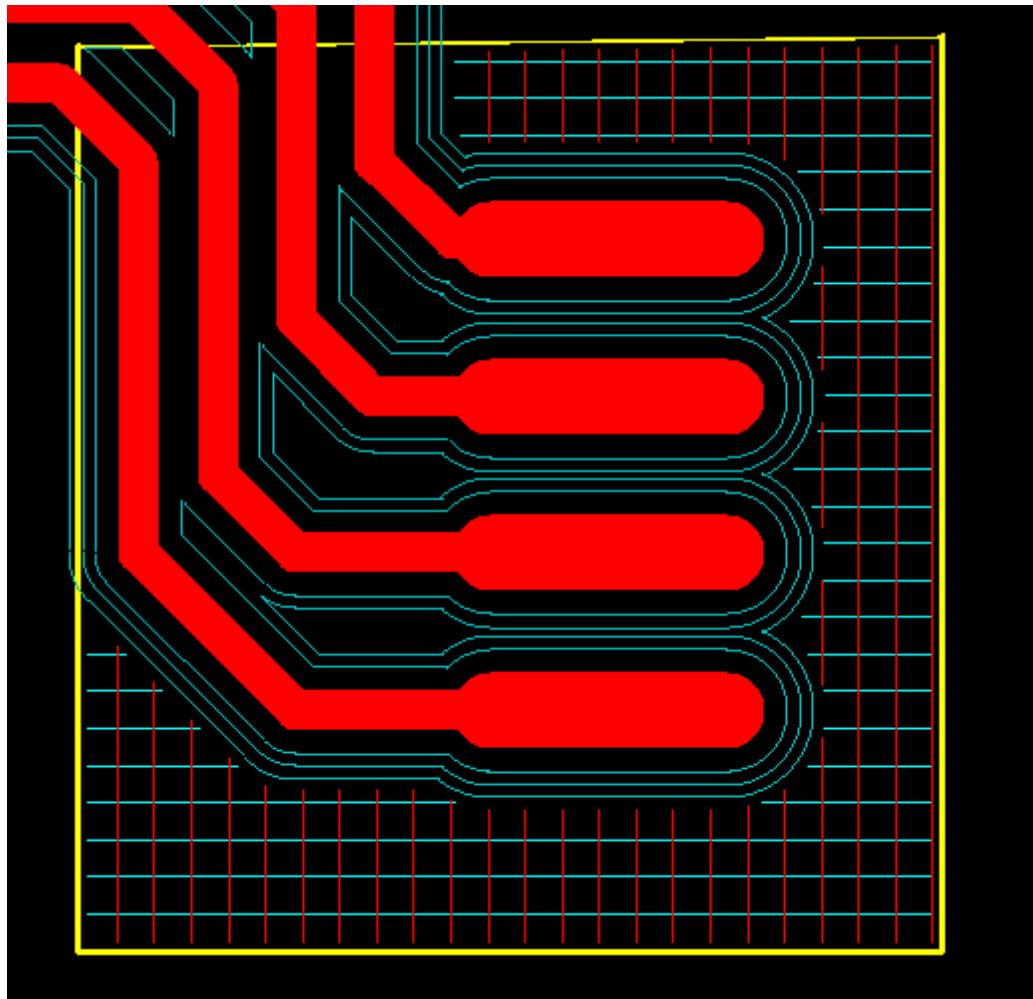


Figure 81. Laser Insulate Script with a defined layer rubout

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

2.4.8. SMT Stencils Tutorial

2.5. Processing NC Data

2.5.1. Export Production Data

How to export to Gerber:

- Select in the main CircuitCAM menu **File -> Export -> Gerber:**
 - Select **GerberXTop** to export production data to the file with an extension **".TOP"**.
Or
 - Select **GerberXBottom** to export production data to the file with an extension **".BOT"**.

CircuitCAM message window appears on the screen, where the destination path of the exported file is presented.

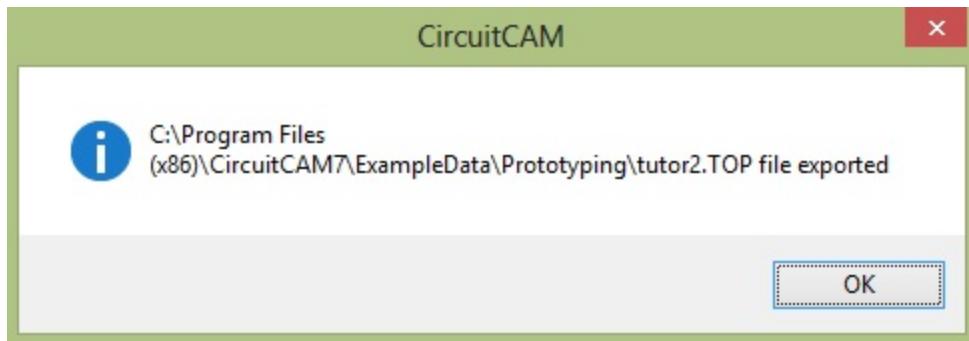


Figure 82. CircuitCAM message window with destination file path

How to export to Excellon:

- Select in the main *CircuitCAM* menu **File -> Export -> Excellon -> Excellon**. Production data are exported to the file with an extension ".**DRL**".

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

2.5.2. NC Mode Tutorial

2.5.3. Close CircuitCAM

How to exit *CircuitCAM*:

- Press on the keyboard shortcut **ALT + F4**.

Or

- Select in the main *CircuitCAM* menu **FILE -> EXIT**.

Or

- Click with a mouse on the **Close** button on the top right corner of the application window.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

2.6. Others

2.6.1. Create *CircuitCAM* Template File

In this example we will show how to create a template file with user-specific settings (aperture and tool list, layer name and color and etc.) and change the default template in settings.

How to create template file:

- Select in the main menu **File -> Recent Files -> Tutor1.cam**.



Note: In case the **Tutor1.cam** file is currently opened, skip this step.

- Ensure that all layers in the **Layers** panel are selectable. In case some of them are not selectable, click on the **All Visible and Selectable** button in the toolbar.

- Select in the main menu **Select** -> **All** to select all graphical objects.
- Press **Delete** to delete the selected objects.
- Select in the main menu **File** -> **Save As**. The **Save As** dialogue window appears on the screen:
 - Navigate to the *New_Template* directory.
 - Select **Template (*.cat)** in the **File Type** combo box.
 - Enter **Tutor1** into the **File Name** text field.
 - Click on **Save** button to save the template file and close the **Save As** dialogue window.
- Select in the main menu **File** -> **New**. The sub-menu appears on the screen, with the included **Tutor1** template file.

How to change default template:

- Select in the main menu **Settings** -> **General Settings**. The **General Settings** window appears on the screen.
- Click on the **Directories** branch of the tree on the left side of the window. The **Directories** panel is opened on the right side of the page.

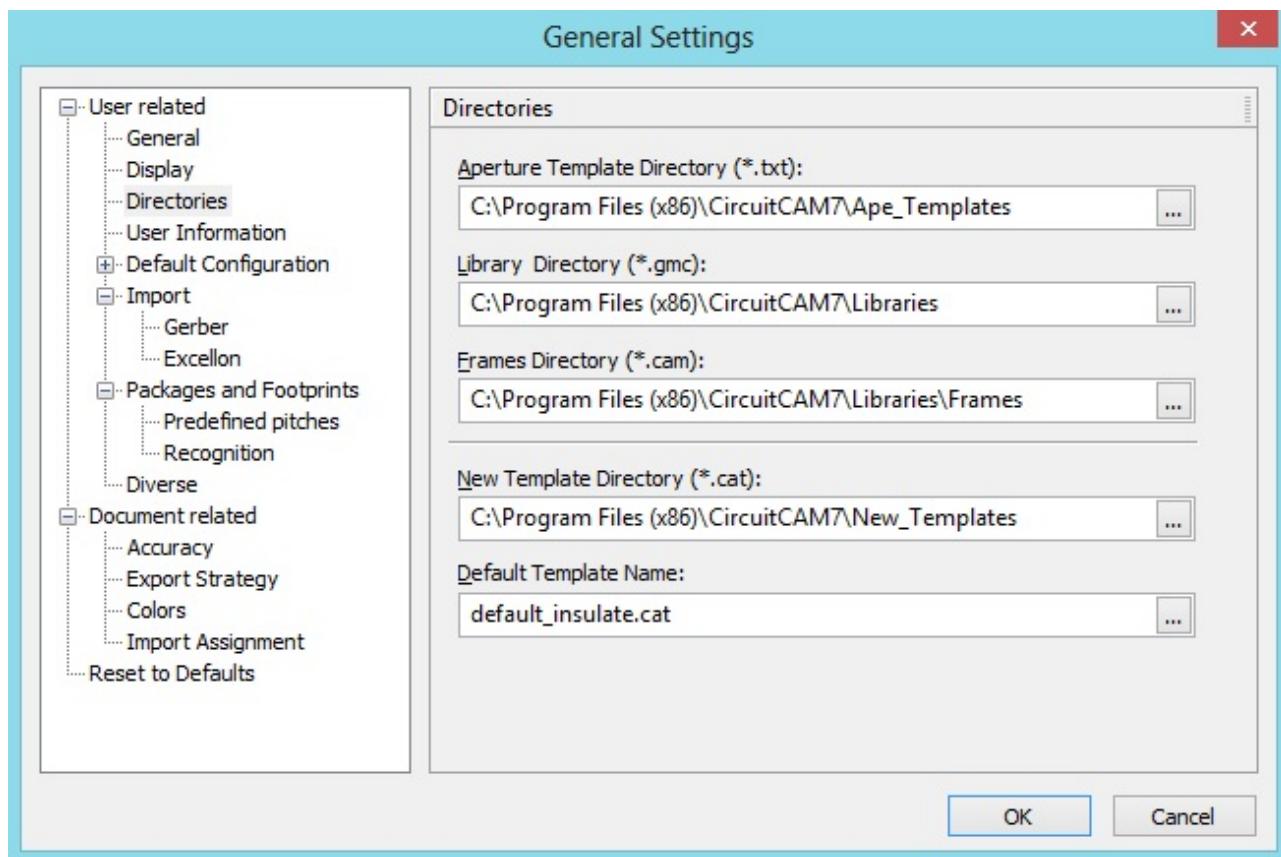


Figure 83. General settings window, Directories panel

- Click on the look up button next to the **Default Template Name** field. The **Select new template** dialogue window appears on the screen.
 - Select the file path for a new default template, e.g. "**Tutor1.cat**".
 - Click **Open** button.

- Click **Cancel** button to continue using of the **default_insulate.cat** as the default template.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

2.6.2. Using Import Assignments

To automate the import of production data, you can specify import assignments for production data with specified file extensions, e.g. "**drl**", "**sld**" and "**cmp**". After you define which file extension represents a particular type of production data, assign a layer and an aperture/tool list, you no longer need to specify this information in the future for the import purposes.



Note: if all production data files selected for import are covered by import assignments, import dialog box will not be shown. Otherwise

How to define import assignments:

- Click on **New** button in the **Standard** tool bar to open a new document based on a default template.
- Select in the main menu **Settings -> General Settings**. The **General Settings** window appears on the screen.
- Click on **Import Assignment** branch of the tree on the left side of the window. The **Import Assignment** panel is opened on the right side of the window.

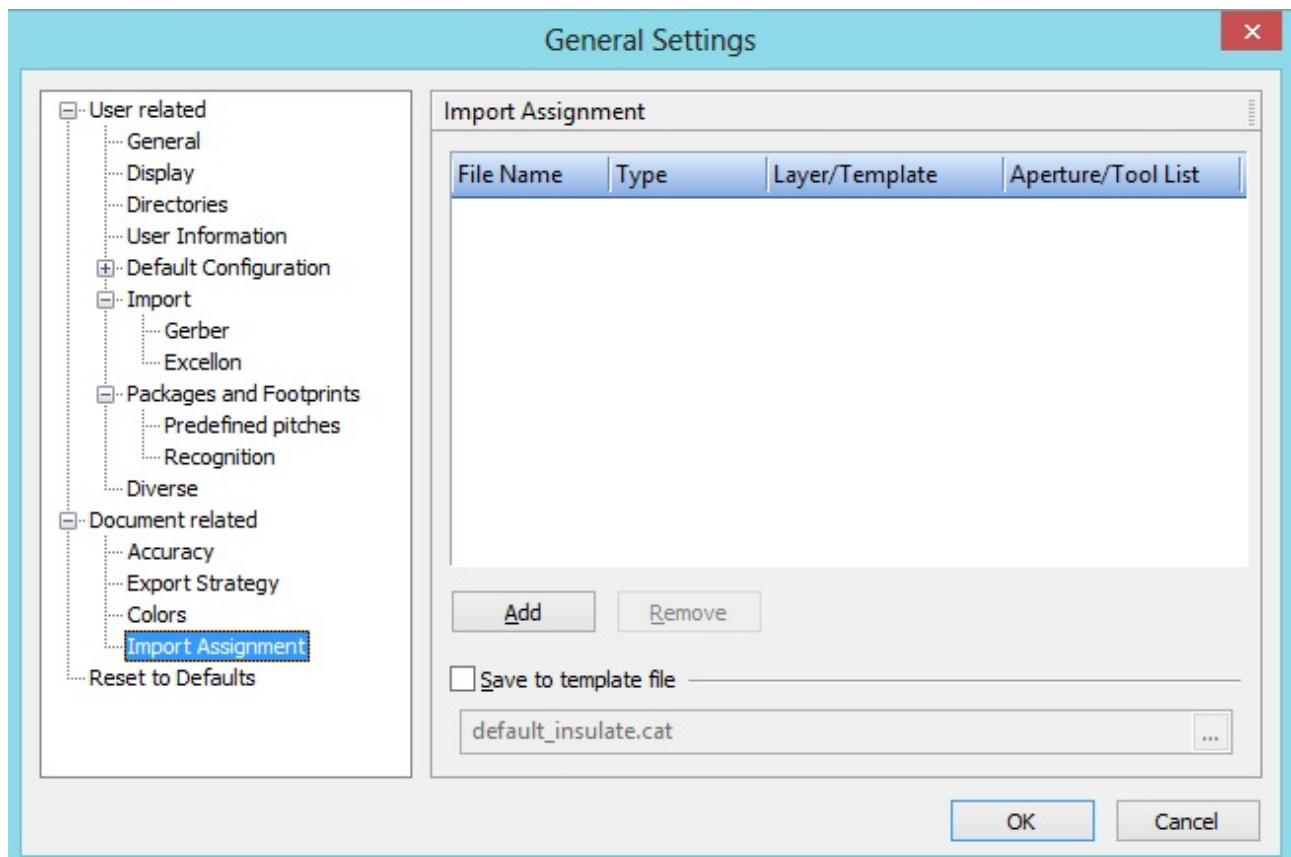


Figure 84. General Settings window, Import Assignment panel

In our example we use the double sided circuit board. We have to import an aperture list, tool list, two *Gerber* files for the conductor tracks and a file of holes.

Please find below the table with the corresponding extensions and assignments.

	File Specification	Aperture/Tool List	File Type	Layer/Template
Aperture list	*.whl	Apertures	Aperture/tool list	TutorApe.txt
Tool list	*.nct	Tools	Aperture/tool list	TutorTools.txt
Excellon data	*.drl	Tools	Excellon	DrillPlated
Gerber data, Component side	*.cmp	Apertures	Gerber	TopLayer
Gerber data, Solder side	*.sld	Apertures	Gerber	BottomLayer

- Click on **Add** button in the **Import Assignment** panel. A new empty row appears in the **Import Assignment** table.

Let's create the import assignment for an aperture list:

- Enter "***.whl**" in the **File Name** cell.
- Open the **Type** list and select "*Aperture Select*".
- Open the **Layer/Template** list and select "TutorApe.TXT".
- Click on the **Aperture/Tool list** cell and enter the name "*Apertures*".

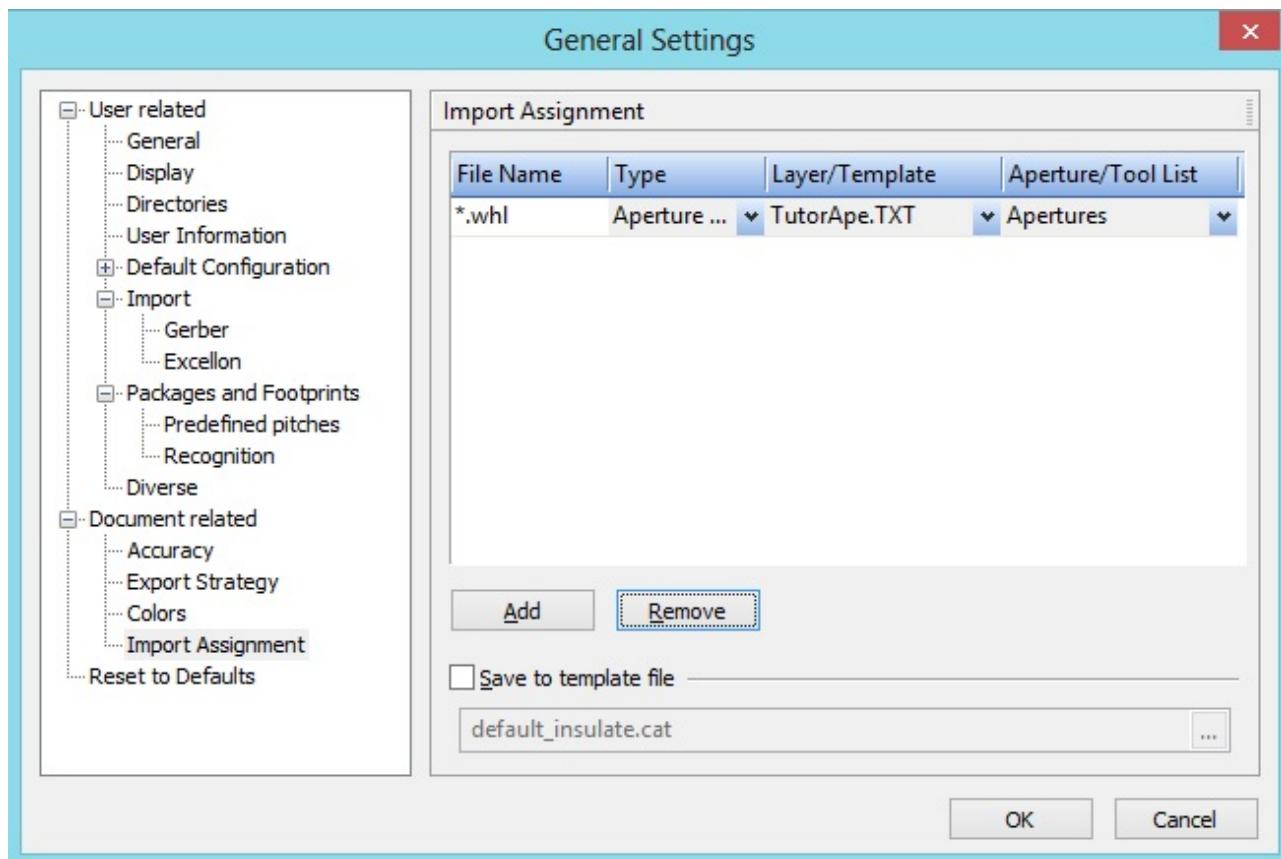


Figure 85. Assignment for aperture list

Let's create the import assignment for a tool list:

- Click on **Add** button.
- Enter "***.nct**" in the **File Name** cell.
- Open the **Type** list and select "*Aperture Select*".

- Open the **Layer/Template** list and select "TutorTools.txt".
- Click on the **Aperture/Tool list** cell and enter the name "Tools".

Create the import assignment for **Gerber** file, which contains layout data for the component side of the circuit board:

- Click on **Add** button.
- Enter "*.cmp" in the **File Name** cell.
- Open the **Type** list and select "Gerber".
- Open the **Layer/Template** list and select "TopLayer".
- Click on the **Aperture/Tool list** cell and enter the name "Apertures".

Create the import assignment for **Gerber** file, which contains layout data for the solder side of the circuit board:

- Click on **Add** button.
- Enter "*.sld" in the **File Name** cell.
- Open the **Type** list and select "Gerber".
- Open the **Layer/Template** list and select "BottomLayer".
- Click on the **Aperture/Tool list** cell and enter the name "Apertures".

Create the import assignment for **Excellon** file:

- Click on **Add** button.
- Enter "*.drl" in the **File Name** cell.
- Open the **Type** list and select "Excellon".
- Open the **Layer/Template** list and select "DrillPlated". The holes in the **Excellon** file should be through plated.
- Click on the **Aperture/Tool list** cell and enter the name "Tools".

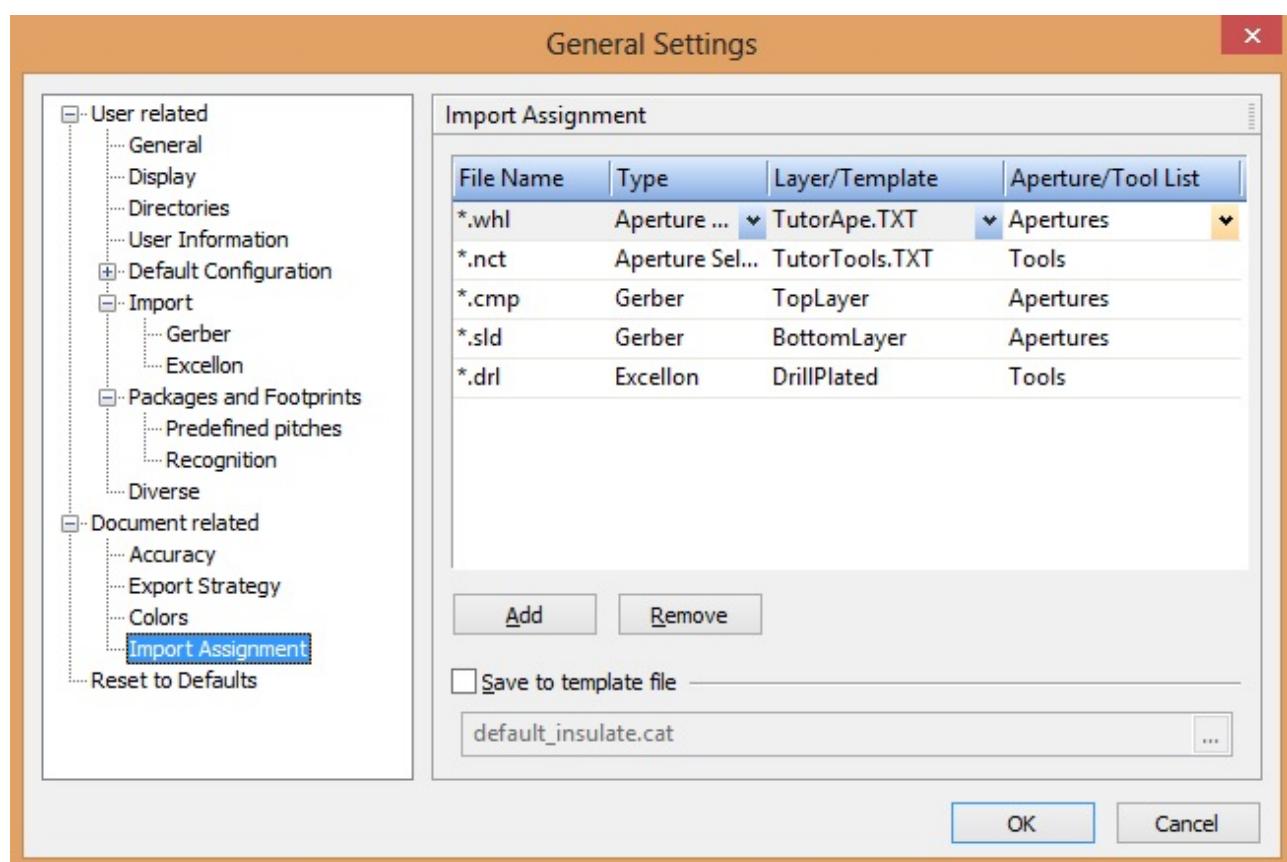


Figure 86. Import assignment DrillPlated

- Click on **Save to template file** check box to save import settings in the specified template file.
- Click on the lookup button next to the field below. The **Select new template** dialogue window appears on the screen.
- Define the path to the template file (i.e. "**tutor.cat**)
- Click **Open** button to confirm changes and close the dialogue window.
- Uncheck the **Save to template** file option, because will save the import settings in the current template file "**default_insulate.cat**".
- Click **OK** button to save import settings to the current template and close the **General Settings** window.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3. Reference Manual

3.1. Main User Interface

CircuitCAM is a so called single document interface application. This means that within one frame window only one **CAM** document is displayed. To handle several **CAM** documents simultaneously, several *CircuitCAM* instances must be opened from the Windows® system, each instance with its own frame windows. Copying data from one instance to another is possible.

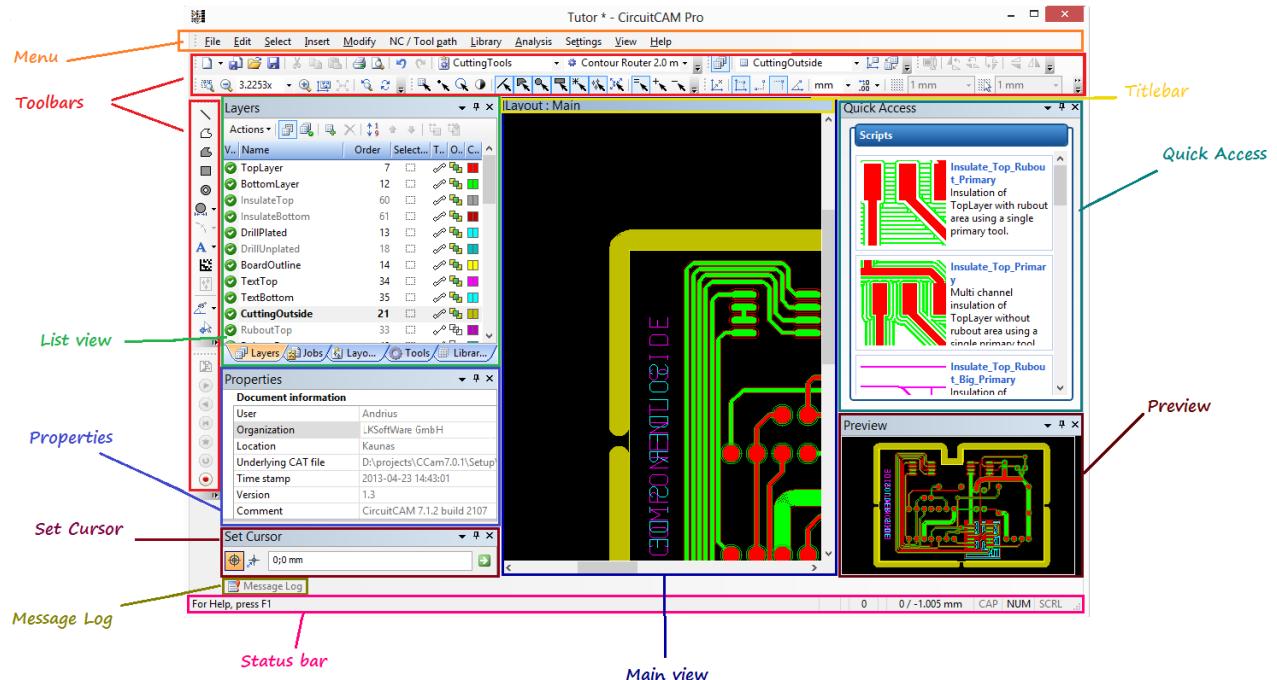


Figure 87. *CircuitCAM* frame window

The main *CircuitCAM* frame window contains:

- [Menu bar](#)
- [Several Toolbars](#)
- [Status bar](#)
- none, one, or several view windows.

Within the main working area, the following non-modal views may show up, depending on whether they are made active or hidden via their menu settings:

- [Graphic View](#)
- [List View](#)
- [Layers](#)
- [Tools](#)
- [Jobs](#)
- [Layouts](#)
- [Libraries](#)
- [Properties](#)
- [Preview](#)

- [Quick Access](#)
- [Message Log](#)
- [Set Cursor](#)
- [Set Anchor](#)
- [Geometry Manipulation Centre \(GMC\)](#)

Except for the Graphic view, all these views are either in the dockable or floating state, see [View arrangement](#). At most one of these views is active at a time, indicated by its title bar.

In contrast to the non-modal views listed above on which the user can work in parallel, the *CircuitCAM* dialogues are modal. After opening a dialogue it has to be closed, typically by **OK** or **Cancel**, before the user can proceed using other *CircuitCAM* functionalities.

The main *CircuitCAM* dialogues are:

- [File menu](#)
 - [Import](#)
 - [Printing and Printer Calibration](#)
 - [Statistics](#)
 - [Design Rule Check](#)
- [Insert menu](#)
 - [Insert Text / Barcode](#)
 - [Insert Data Matrix](#)
 - [Base Material / Fixture Holes](#)
 - [SMT Stencil Frame Generator](#)
- [Library menu](#)
 - [Create Package](#)
 - [Package Recognition](#)
 - [GMC Exchange](#)
 - [Import Libraries](#)
- [Tool path menu](#)
 - [Laser Scanner Fields](#)
 - [Contour Routing](#)
 - [Tool Compensation](#)
- [Modify menu](#)
 - [Size Modification](#)
 - [Step and Repeat](#)
 - [Create Bounding Shape](#)
 - [Create Outline of Shape](#)
 - [Perfect Shape](#)
 - [Perfect Shape Tolerance](#)
 - [Pitch of two Pads](#)
- [Settings menu](#)

- [Customize](#)
- [General Settings](#)
- [Help menu](#)
 - [CircuitCAM Activation Wizard](#)
 - [About CircuitCAM](#)

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.1.1. Menu

CircuitCAM user interface includes a menu bar on the top section of the window. Point with the mouse on an applicable menu item as in the picture below:



Figure 88. *CircuitCAM* Menu bar

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.1.1.1 File

The **File** menu is the most important starting point to create new, and open existing documents, save, import, export and print documents.

Icon	Function	Hotkey	Description	Further Info.
	New	Ctrl + N	Creates a new <i>CAM</i> document with the name "Untitled N" (N = 1,2,3...), that is based on a specified template file or the default template file. Typically the template files contains no graphic objects, but only layers, tools, etc.	See New
	Open	Ctrl + O	Opens an existing document (*.CAM or *.CAT).	See Open
	Synchronize		Updates the current CAM document according to a template file.	See Synchronize
	Close	Alt + F4	Closes the current active document if there are no changes made since the last document save. If the document was modified, <i>CircuitCAM</i> prompts to save or to discard the changes.	
	Import		Imports files into the current document.	See Import
	Export		Exports the production data.	See Export
	Save	Ctrl + S	Saves the current document using the existing file name. In case of a new <i>Untitled N</i> document, when pressing Save the Save As menu is acti-	See Save

Icon	Function	Hotkey	Description	Further Info.
			vated.	
	Save As		Saves the current document in a different location, <i>and/or</i> with a different file name <i>and/or</i> a different file format.	see Save as
	Print	Ctrl + P	Prints the active document. The CAM file name is printed on top of the page.	See Printing
	Print Preview		Displays the active document on the screen, as it would appear printed.	
	Print Setup		Selects a printer, a printer connection and printing options. For more information see your Windows® documentation.	
	Printer Calibration		Allows you to correct the aspect ratio of the printer in X and Y directions to achieve higher accuracy.	See Printer Calibration
	Recent files		Displays a list of your most recently accessed files, this list displays the last opened ten files.	
	Exit		Quit <i>CircuitCAM</i> with a prompt to save or discard any unsaved changes.	

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.1.1.2 Edit

The **Edit** menu offers the following commands:

Icon	Function	Hotkey	Description	Further Info.
	Undo	Ctrl + Z	Undoes the last three manipulations. There is a possibility to change the depth of Undo in General Settings dialogue, General panel.	
	Redo	Ctrl + Y	Reverses the actions of the last Undo operations.	
	Cut	Ctrl + X	Moves the selected objects to the standard clipboard.	
	Copy	Ctrl + C	Copies the selected objects to the standard clipboard.	
	Copy As EMF in		Copies Colored or Monochromatic graphics from <i>CircuitCAM</i> to be used in other Windows® applications. The <i>Enhanced Metafile</i> format is used.	
	Paste	Ctrl + V	Inserts the object(s) from the clipboard.	Procedure
	Remove	Delete	Deletes all selected objects.	• Select the

Icon	Function	Hotkey	Description	Further Info.
				<p>objects you want to delete.</p> <ul style="list-style-type: none"> Start the Delete operation (menu or keyboard). All selected objects are deleted. Thus no further objects are then selected.
Procedure				
	Move Selected	Space	<p>All objects in a selected group can be moved together. The relative positions and orientations between the objects are not changed. In this operation, the position of the active marker is reset, <i>i.e.</i>: all objects are moved by the same distance. This distance is obtained from the difference between the old and the new positions of the active marker.</p>	<ul style="list-style-type: none"> Select the objects you want to move. Start the Move operation (menu or keyboard).

Icon	Function	Hotkey	Description	Further Info.
				<ul style="list-style-type: none"> The selected objects follow the mouse movements virtually. Set a new position for the active marker by clicking the left mouse button. <p>The operation can be interrupted with the Cancel. The new position is on the selected cursor grid. All objects stay selected.</p>
	Duplicate Selected	Ctrl + Space	The Duplicate operation is very similar to the Move operation, only here a new copy is made instead of moving the original object.	
	Rotate Relative		See Rotation and mirroring function (relative mode) on page Rotation and Mirroring .	
	Set Flash Rotation Value		See Rotation and mirroring function (absolute mode) on page Rotation and Mirroring .	
	Set Cursor	Ctrl + Shift + C	This function simulates a left mouse click at the entered co-ordinates. It also activates the Set	See Set Cursor

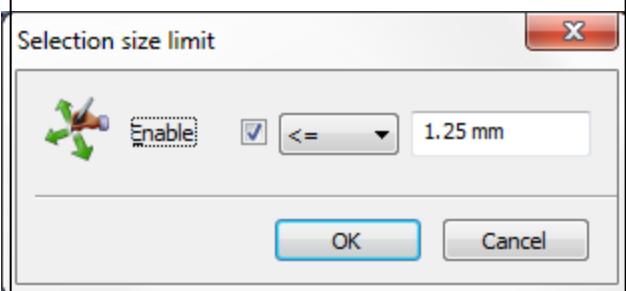
Icon	Function	Hotkey	Description	Further Info.
			Cursor panel, in case it is not visible on the screen.	
	Set Cursor to Zero	Ctrl + Shift + 0	Simulates a left mouse click at position (0,0).	
	Set Anchor Point	Ctrl + Shift + A	This function allows setting the anchor point to any other co-ordinates. It also activates the Set Anchor window, in case it is not visible on the screen.	See Set Anchor Point
	Anchor to Centre Point	Ctrl + Shift + Z	Calculates a bounding box of all selected objects and sets the anchor point to the centre of this bounding box.	The function is based on resolution of <i>Polygon Package</i> .
	Anchor to Mass Point	Ctrl + Shift + M	Calculates a mass point of all selected objects and sets the anchor point to this point.	The function is based on resolution of <i>Polygon Package</i> .
	Set Zero Point	Ctrl + Shift + N	A zero point (X/Y coordinate 0/0) can be set within the graphic or outside the layout. Set Zero Point function makes the current anchor point the new zero point (Origin point). So, all graphic objects in all layers are moved according new zero point. The relative position of graphic objects to each other stays unchanged.	
	Move Layer to Zero	Ctrl + Shift + L	The Move Layer to Zero function works in a similar way the Set Zero Point function does, but it sets the current anchor point to the new zero point (Origin point) only for one layer of graphic objects. The layer must be selected from the list of layers on the Layers list view.	
	Match up Layers		To execute this function: press the button, select a flash (source layer and its point) by a mouse click, move the cursor to another flash in the destination layer (a rubber band shows the road from the first point). This option works on flashes only (the nearest flash is selected).	

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.1.1.3 Select

Many operations, such as move, delete etc..., work only on selected objects. A single or group selection can be used. For more information on the selection status, see [Selection of graphic objects](#).

CircuitCAM has a number of functions and switches to influence the way in which objects are selected, and to determine what kinds of graphical objects may or may not be selected.

Icon	Function	Hotkey	Description	Further Info.
	Layer		Selects the entire active layer. The active layer is either selected from the Layers panel or by selecting a graphical object included in it. The layer must be visible and selectable. In case the <i>Plus/Minus</i> selection mode is set, the layer is added/removed to/from the existing selections.	
	Aper-ture/Tool		Selects all graphical objects which are created with the same active aperture or tool. The selection is only made from the visible and selectable layers. The active aperture or tool list is selected from the Tools panel or by selecting a graphical object. In case the <i>Plus/Minus</i> selection mode is set, the tools are added/removed to/from the existing selections.	
	All	Ctrl + A	Selects all graphical objects on all layers. The selection is only made from the visible and selectable layers.	
	Invert Selection		Inverts the current selection: deselects the selected objects and selects unselected objects.	
	Path		Enables/disables selection of paths.	
	Polygon		Enables/disables selection of polygons.	
	Circle		Enables/disables selection of circles.	
	Rectangle		Enables/disables selection of rectangles.	
	Pad/Hole		Enables/disables selection of flashes.	
	Sublayout		Enables/disables selection of sub-layouts.	
	Size Limit		<p>Enables/disables selection of objects that are bigger or smaller than a given value. This limit is applied on the bounding box of the objects (the bigger value in either X or Y direction) and works in combination with the current selection filters. The limit value is typed into the edit field</p>  <p>Figure 89. Selection size limit dialogue window The state (condition) is set from the combo box and can be: <, <=, >, >=, =. The size limit is used only when the Enable check box is ON.</p>	

Icon	Function	Hotkey	Description	Further Info.
=	New		Resets all previous selections in the event of a new selection (default setting).	
+	Plus		Adds a new selection to the existing selection.	
-	Minus		Removes an existing item from the current selection.	
	Sequence		<p>This operation changes the selection of objects. Properties of the selected object (type, current position) are shown in the <i>Status Bar</i>. Function has four operations:</p> <ol style="list-style-type: none"> 1. select the next object in the group; Ctrl + Shift + Right 2. select the previous object in the group; Ctrl + Shift + Left 3. select the first object in the group; Ctrl + Shift + Up 4. select the last object in the group. Ctrl + Shift + Down 	See Sequence , Hotkeys
	Pitch		Searches and selects the graphical objects, which have the same pitch, spacing and area.	See Pitch

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.1.1.4 Insert

Inserts a new graphic object directly to the currently active layer.

Icon	Function	Description	Further Info.
	Rubout Area	<p>Inserts a new rubout area.</p> <p> Top layer creates a rubout area on the top side of the circuit board;</p> <p> Bottom layer creates a rubout area on the bottom side of the circuit board;</p> <p> All layers – on both sides.</p>	See Rubout Area
	Open Path	Inserts a new path that consists of one or more lines or arcs segments.	See Open Path
	Closed Path	Inserts a new path. The <i>Closed Path</i> function is very similar to <i>Open Path</i> function except that the last path point will be joined to the start point. In the drawing procedure the <i>Open Path</i> function must be changed to the <i>Closed Path</i> function.	See Closed Path

Icon	Function	Description	Further Info.
	Polygon	Inserts a new polygon which is a filled area defined by an outline.	See Polygon
	Rectangle	Inserts a new rectangle. The new created shape is a filled area defined by an outline.	See Rectangle
	Dynamic Radius Circle	Inserts a circle which you can control its radius moving your mouse cursor. The new created shape is a filled area defined by an outline.	See Centre method on page Circle .
	Fixed Diameter Circle	Inserts a circle with a predefined fixed diameter. The new created shape is a filled area defined by an outline.	See Centre + Radius method on page Circle .
	Circle in Closed Path	Inserts a closed path circle with a fixed diameter. The new created shape is a closed path outline.	See Circle in closed path method on page Circle .
	Point on Arc	Inserts a new arc, where an additional point on the arc determines the circle sector. Only active while drawing.	See Arc
	Centre Point Arc	Inserts a new arc, where the circle sector is determined by defining the <i>Centre point</i> of the circle. Only active while drawing.	See Arc
	Pad	Inserts a new pad (flash).	See Pad
	Fiducial	Simplifies the process to insert fiducials: the new feature allows creating fiducials with just two mouse clicks. One click on the menu Insert ---> Fiducial , and another click on the graphic screen to place the fiducial. The diameter can be set in the Diverse panel of the General Settings . The fiducial will always be placed on the Fiducial layer. If such layer does not exist, the layer will be created.	
	Arbitrary Hole	Inserts a new drill hole. This function is very similar to the insert <i>Circle</i> function except that the drill hole is only executed on one of the following layers only: ■ Plated - the drill hole drawing is executed on the <i>Drill-Plated</i> layer ■ Unplated - the drill hole drawing is executed on the <i>DrillUnplated</i> layer	
	Insert Text	Inserts a new text: A Top... - inserts a text on the <i>TextTop</i> layer. A Bottom... - inserts a text on the <i>TextBottom</i> layer. A Active layer... - inserts a text on the currently active layer.	See Text
	Data matrix	Inserts a Data Matrix barcode.	See Data-

Icon	Function	Description	Further Info.
			Matrix
	Insert Sub-layout	Inserts a sub-layout (selected instance) in the current active layout.	
	Base material / fixture holes	Inserts a base material with or without fixture holes.	See Base material
	Stencil Frame Generator	Inserts a new Stencil Frame job	See "SMT Stencil Frame Generator"
	Cutout	Inserts a new polygon Cutout. The polygon Cutout consists at least of two figures. The first figure is always an outer figure (boundary), all others figures are inner figures (cutouts).	See Polygon with Cutouts
	Copper pouring	The function is rather similar to function for creating of a <i>Polygon with Cutouts</i> .	See Copper Pouring Note This function is only available in CircuitCAM variants STANDARD and PRO.
	Line mode 45°	Aligns the segments and arc segments so that the mouse cursor only moves by an angle of 45 degrees and increments. Only active while drawing.	See Line Drawing Modes
	Line mode 90°	Aligns the segments so that the mouse cursor only moves by an angle of 90 degrees and increments. Only active while drawing.	See Line Drawing Modes
	Select inserted object	If this option is <i>on</i> , the inserted(pasted) objects are selected in addition to the previous selected ones. If this option is <i>off</i> the inserted(pasted) objects are not selected or active.	

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.1.1.5 Modify

Icon	Function	Hotkey	Description	Further Info.
	Curve; A set of single object manipulation operations, including the following:			
	Add Point/Seg-	Ctrl + T	Creates and places a new point at the current anchor point position. A new segment is connected	See Add Point/Seg-

Icon	Function	Hotkey	Description	Further Info.
	Segment		to the selected path with line segments.	Segment
	Cut Point/Segment	Ctrl + U	Splits a path into two segments at the anchor point.	See Cut Point/Segment
	Delete Point/Segment	Ctrl + D	Deletes a point or a line segment at the current anchor point.	See Delete Point/Segment
	Move Point/Segment	Ctrl + M	Moves a point or a line segment at the current anchor point position.	See Move Point/Segment
	Create or Move Arc	Ctrl + E	Converts a line segment into an arc at the current anchor point position.	See Create or Move Arc
	Combine	Ctrl + B	Combines selected paths to a single path.	See Combine
	Close	Ctrl + Shift + Q	The selected open path becomes a closed path.	See Close
<hr/>				
	Path to Polygon	Ctrl + Shift + P	Converts the selected open path to a polygon. The path gets a new line segment to connect start and end points.	See Path to Polygon
	Combine to Polygon	Ctrl + K	Combines selected paths and converts them to a polygon.	See Combine to Polygon
	Convert to Polygon	Ctrl + Shift + Y	Converts any selected object to a polygon. Overlapping objects are converted to a single polygon.	See Convert to Polygon
	Convert to Closed Outline Path	Ctrl + Shift + W	Converts selected object to a closed path. Separate outlines will be generated for overlapping objects.	See Convert to Closed Outline Path
	Denest all Cutouts	Ctrl + Shift + I	Converts all negative cutouts for the selected polygon to positive objects.	See Denest Polygon Cutout
	Denest single Cutout	Shift + J	Converts a single selected negative cutout to a positive object.	See Denest Polygon Cutout Hole
	Convert to Circle	Ctrl + Shift + K	Converts the selected object to a circle with the same area.	See Convert to Circle
<hr/>				
	Draw to Flash	Ctrl + Shift + D	Converts selected objects to flashes. Overlapping objects are converted into a single flash.	See Draw to Flash
	Convert to One Flash	Ctrl + Shift + H	Converts selected objects into a single flash.	See Convert to One Flash
	Convert to		Converts selected objects into a single instance/lay-	See Con-

Icon	Function	Hotkey	Description	Further Info.
	Instance		out.	vert to Instance
Exchange With Flash; and it includes the following:				
	Compare Shape	Ctrl + Shift + S	Compares all objects with a shape of the selected flash. Replaces identical to the selected flash objects with a flash. The original objects are deleted and all new objects are made as selected.	See Compare Shape
	Compare Bounding Box	Ctrl + Shift + B	Compares all objects with a bounding box of the selected flash. The original objects are deleted and all new objects are made as selected.	See Compare Bounding Box
	Compare Outline	Ctrl + Shift + T	Compares all objects with an outline of the selected flash. The original objects are deleted and all new objects are made as selected.	See Compare Outline
<hr/>				
	Denest Flashes/Instances	Ctrl + Shift + E	Selected flashes/instances are replaced in a graphic window with their origin shapes (circle, rectangle, path and so on).	Note ¹
	Undo Footprint Occurrences		Selected footprint occurrences are restored in a graphic window with their origin shapes.	Note ²
<hr/>				
	Size Modification	Ctrl + Shift + U	Implements size manipulations on selected objects.	see Size manipulations dialogue window Note ³
	Step and Repeat	Ctrl + Shift + R	Repeats selected objects in a right-up direction, according to the settings in the dialogue window.	See Step and Repeat
	Create Bounding Shape	Ctrl + Shift + G	Creates a circular bounding shape one per each group of overlapped graphic objects.	See Create Bounding Shape
	Create Outline	Ctrl + Shift + O	Generates an outline on the edge of each object located on the Source layer.	See Create Outline
	Perfect Shape	Ctrl + Shift + V	Checks selected objects on a perfect rectangle, a rounded rectangle, a bevelled rectangle, a circle and an oval according to the given tolerance. If yes, converts the perfect shapes into flashes of mentioned types.	See Perfect Shape
<hr/>				
Boolean Operations; and it includes the following:				
	Union		Performs the set operation; Union	See Union
	Difference		Performs the set operation; Difference	See Difference

¹: This function is only available in the CircuitCAM variants STANDARD and PRO.

²: This function is only available in the CircuitCAM PRO variant.

³: This function is only available in the CircuitCAM PRO variant.

Icon	Function	Hotkey	Description	Further Info.
				reference
	Intersection		Performs the set operation; Intersection	See Intersection
	XOR		Performs the set operation; XOR	See XOR
	Paths on Area		Trim paths, leaving portions on a selected area.	See Paths on Area
	Paths off Area		Trim paths, leaving portions off a selected area.	See Paths off Area
	Item edit		Enabled only while graphically editing an aperture, tool, or layout. The graphical editing of this aperture, tool, or layout can be ended by either saving the changes or discarding them.	see also Tool Properties , Libraries , Active config-item .
	Geometry Manipulation Centre (GMC)		Perform various operations to change geometry of pads. There is a possibility to apply automatic exchange on the same pads. See Overview	Note ¹

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.1.1.6 Tool Path

Icon	Function	Hotkey	Description	Further Info.
	Laser Scanner		Generates a mesh of laser scanners.	
	Contour Routing		Calculates and creates an inner or outer milling contour.	See Contour Routing Note ²
	Create gap	Ctrl + G	Adds a gap at the active marker point.	See Create Gap
	Tool Compensation		Generates an offset tool path for the selected path.	See Tool Compensation
	Swap		Changes the order of object/group of objects. Move to next: Ctrl + J Move to previous: Ctrl + I Move to first: Ctrl + F	See Swap

¹: This function is only available in the CircuitCAM PRO variant.

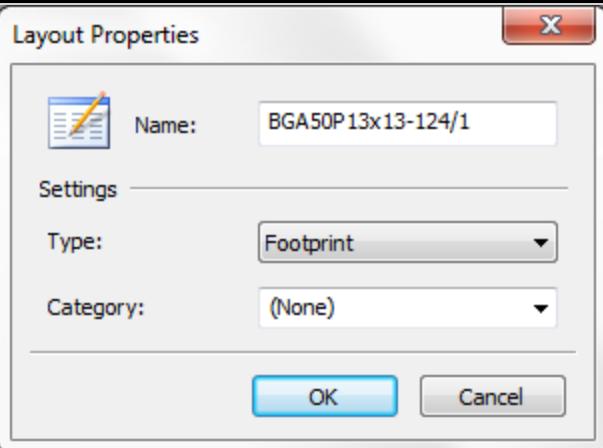
²: This function is not available in the CircuitCAM VIEWER variant.

Icon	Function	Hotkey	Description	Further Info.
			Move to last: Ctrl + L	
	Reverse	Ctrl + R	Reverses the selected open path.	
	Connect to NC Server		Connects to an NC server, whether local or remote server.	
	NC Continue		Continue in NC mode without discarding the previous changes.	
	NC Normal		Starts a new NC session with normal orientation.	
	NC Mirror-X		Starts a new NC session mirrored on X-axis.	
	NC Mirror-Y		Starts a new NC session mirrored on Y-axis.	
	End NC		Exits NC mode.	

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.1.1.7 Library

Icon	Function	Hotkey	Description	Further Info.
	Create Package		Allows the user to create a new package. The creation dialogue is very similar to Create Package from Selection, but the user himself specifies main package parameters - columns, rows and pitch.	
	Create Package from Selection		Allows the user to create a new package from selected pads.	See Create Package from selection
	Create Footprint from Selection		Allows the user to create a new footprint from selected pads. If more than one footprint library exists in specified library directory, the dialogue box with a list of all available footprint libraries will be displayed. If no libraries exist in the specified library directory, a user will be prompted to create a new footprint library. User needs to specify the Name for new footprint:	

Icon	Function	Hotkey	Description	Further Info.
			 <p>Figure 90. Layout Properties dialogue window This must be done carefully, because the applying of footprints for recognized packages are based on names checking.</p>	
	Package Recognition		Starts automatic recognition of packages according to the selected package library.	See Package Auto Recognition
	GMC Exchange		All enabled GMC shapes from GMC libraries are used for GMC Exchange. After the command is invoked, CircuitCAM starts a comparison process (similar as a mechanism, described in Geometry Manipulation Centre - see Compare): the original shape is compared with objects, placed in the layers, specified in the list.	
	Apply footprints		<p>All enabled footprints in Libraries view are used for apply footprints functionality.</p> <p>All layouts in the current CAM document are scanned. For each Package layout enabled footprints are checked. When footprint or footprints are found, the apply for corresponding Package layout is performed.</p>	See Apply footprint
	Import libraries		<p>CircuitCAM has a new internal data format for GMC libraries. If a user wants to use earlier created library, he will need to import (convert) them into a new format.</p> <p>The selected library(-ies) are copied into the libraries directory according to General settings. If libraries are in the old GMC format (pair of config items for original and modified shape), they are automatically converted into the new format (layout, "OriginalObj" and "ModifiedObj" layers).</p> <p>An extension of the converted file becomes "cam-gmc" (old library type was "gmc"), but a name of the library remains the same.</p>	

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.1.1.8 Analysis

Icon	Function	Hotkey	Description	Further Info.
	Measure	Ctrl + Shift+ X	Calculates a distance between the last selected Anchor Point and the current cursor position.	See Measure
	Pitch of Two Pads		Calculates the distance between the two centre points of two selected areas (pads).	See Pitch of Two Pads
	Area of Selected Objects		Calculates an area of selected objects.	See Area of selected objects
Design Rule Check, which includes:				
	Stencil DRC		Checks the stencil patterns according to the IPC7527 guidelines.	See Stencil Design Rule Check
	PCB DRC		<p>Tests the minimal distance of graphic objects of one layer or within two different layers. <i>CircuitCAM</i> supports the design rule check to test for a minimum distance. The following types of design rules are supported:</p> <ul style="list-style-type: none"> Intra Min Spacing: Tests the minimal distance of graphic objects within one layer. Inter Min Spacing: Tests the minimal distance of graphic objects within two different layers. <p>After performing these design rule checks, the violations of these rules are displayed as graphic object on the result layer.</p>	See PCB Design Rule Check
	Statistics		Shows statistics about the current document.	See Statistics

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.1.1.9 Settings

The **Settings** menu allows a user to customize display units, display/cursor grids, select a coordinate mode, and modify *CircuitCAM* features and options.

Icon	Function	Hotkey	Description	Further Info.
	Absolute Coordinates		Displays the mouse position in absolute coordinates.	
	Relative Coordinates		Displays the mouse position in coordinates relative to the Anchor Point.	
	Cartesian Coordinates		Displays the mouse position in Cartesian coordinates.	
	Polar Coordinates		Displays the mouse position in polar coordinates.	
	Big Cursor		Switches to big cursor; two perpendicular lines on the graphic area. The cursor type can be set in the General Settings dialogue window (Display panel).	
	Measurement Units		Changes the measurement units from the predefined list of units. The default value can be set in the General Settings dialogue window (General panel).	
	Decimal Places		Changes the number of decimal places that a unit is measured in. The default value can be set in the General Settings dialogue window (General panel).	
	Show Grid		Shows/hides the grid on the graphic window.	
	Grid Spacing		Changes the spacing between grid lines from the predefined list of values. This option only works when the grid is visible.	
	Snap to Grid		Turns on and off the snap to grid feature. The snap to grid feature explains as follows: When you draw, resize, or move a shape or an object, it will align or "snap to" the nearest point in the grid, even if the grid is not visible. If your shape or object does not move where you want, turn off the snap to grid option or choose a smaller scale grid.	
	Cursor Grid Spacing		Selects grid spacing for the positioning of the graphical objects, this option only works when the Snap to Grid feature is enabled.	
	Customize		Allows the user to modify menu options, the layout of toolbars, view and change the keyboard shortcuts mapping, menu settings, and general preferences.	See Customize
	General Settings		Shows and allows changing of the main user and document related preferences of <i>CircuitCAM</i> .	See General Settings

3.1.1.10 View

Controls how large or small the graphical representation appears on your screen. Allows showing/hiding of toolbars. Setting layers' properties. Showing/hiding various functions and indications windows.

Icon	Function	Hotkey	Description	Further Info.
	Zoom Window	Alt + draw box with mouse	Allows user to draw a rectangle around a specific area you wish to zoom into.	
	Zoom In	Page Up or Mouse Wheel Up	The graphical representation is shown enlarged by a factor of 2 around the screen centre point. By using keyboard function Page Up the current cursor position is used instead of the screen centre point. If the mouse cursor is outside the graphic view area, the last recorded mouse position will be used as zooming point.	Note ¹
	Zoom Out	Page Down or Mouse Wheel Down	The graphical representation is shown reduced by a factor of 2 around the screen centre point. By using keyboard function Page Down the current cursor position is used instead of the screen centre point. If the mouse cursor is outside the graphic view area, the last recorded mouse position will be used as zooming point.	Note ²
	Zoom scale		The graphical representation is shown zoomed by a predefined scale factor.	
	Zoom selected		Zooms in/out so that all selected graphical objects fits the screen. If no objects are selected, the function is disabled.	
	Overview	Home	All graphical objects are shown fitting the screen.	
	Previous	Insert	The previous view is returned. Up to twenty views are saved.	
	Refresh	F5	The graphical representation is completely redrawn again. This function is sometimes needed after deleting or other manipulations are made, to obtain a clear graphical representation once again.	
	Toolbars		CircuitCAM has a number of toolbars, which altogether include associated functions. Click on the toolbar you wish to display or hide.	See Toolbars

¹: The initial zoom-in point is the screen centre point and is valid until the mouse is moved. After the mouse cursor is moved, it will be considered the centre point of zooming. So, before moving the mouse, the user can press the PageUp key several times until the desired zoom level is reached.

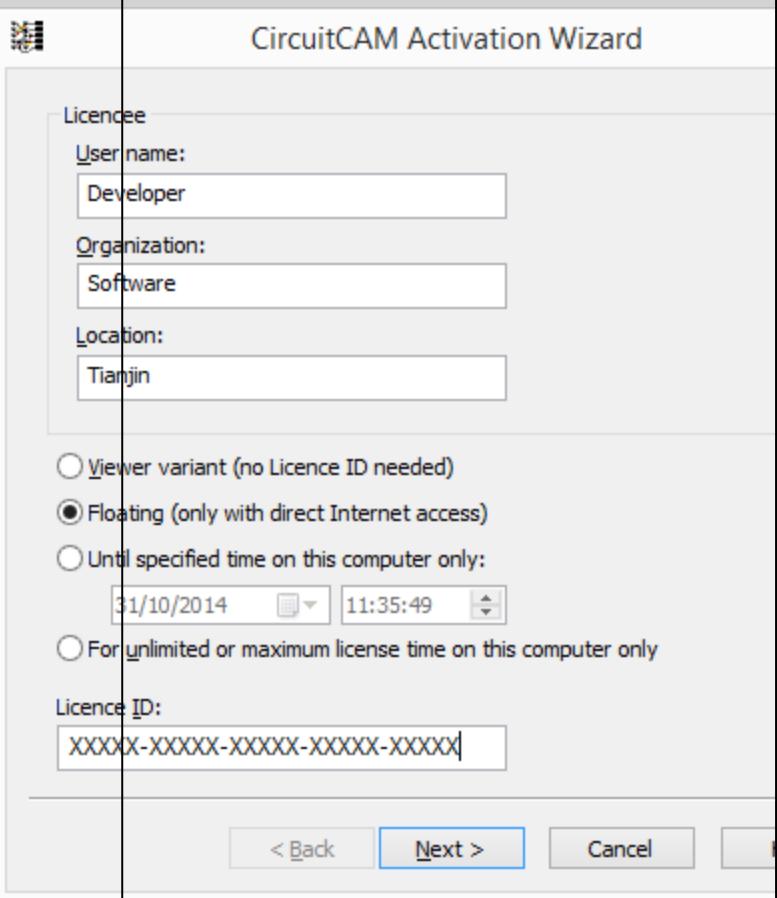
²: The initial zoom-out point is the screen centre point and is valid until the mouse is moved. After the mouse cursor is moved, it will be considered the centre point of zooming. So, before moving the mouse, the user can press the PageDown key several times until the desired zoom level is reached.

Icon	Function	Hotkey	Description	Further Info.
	Status Bar		The status bar at the bottom of the application window displays information regarding the object or function selected.	See Status bar
	Full Screen	Ctrl + Shift + Enter	Resizes the graphic representation of view mode to fit the full monitor size. To cancel the full screen mode, click on the Full screen button on the top-left corner of the window or use the keyboard combination.	
	Message Log		Activates Message Log panel, which is dedicated to view and handle various log/report messages generated by <i>CircuitCAM</i> after/while performing various operations.	See Message Log
	Layers		Activates the Layers panel which displays a list of the layers.	See Layers
	Tools		Activates the Tools panel which displays a list of available tools.	See Tools
	Jobs		Activates the Jobs panel which a list of available jobs.	See Jobs
	Layouts		Activates the Layouts panel which displays a list of available layouts.	See Layouts
	Libraries		Activates the Libraries panel, displaying a list of libraries according to Settings ---> General Settings ---> Directories ---> Library Directory settings.	See Libraries
	Properties	Ctrl + Q	Displays properties of the selected items.	See Properties
	Preview		Opens the Preview window, which is an overview of your whole graphical design.	See Preview
	Quick Access		Activates the Quick Access view which enables immediate access to a particular job.	See Jobs
	Auto Hide All		Enables Auto hide to all enabled/visible panels in <i>CircuitCAM</i> window.	

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.1.1.11 Help

Icon	Function	Hotkey	Description	Further Info.
	Help Topics	F1	Opens an online Help index and a table of contents for <i>CircuitCAM</i> . To use context-sensitive Help call the special cursor type  , which provides context-sensitive help for many <i>CircuitCAM</i> 's interface elements. This cursor type can be called with Shift + F1 or the  button	

Icon	Function	Hotkey	Description	Further Info.
			from the Main toolbar.	
			Opens the <i>CircuitCAM Activation wizard</i> dialogue window, where the user information is presented. See Wizard dialogue window	
	Activate Product		 <p>Figure 91. CircuitCAM Activation Wizard window</p>	
	About CircuitCAM		Displays the version number of <i>CircuitCAM</i> and our contact information. This window also displays the personal information of the registered owner.	

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.1.2. Toolbars

CircuitCAM has a number of toolbars which include buttons with associated functions. The showing/hiding of a toolbar is managed throughout the **Toolbars** submenu of the [View](#) menu. A user can move a toolbar to any applicable place of the window by using the mouse.

To move a toolbar:

- Place the mouse pointer on the area between the buttons on the toolbar or on the title bar of a floating toolbar.

- Drag the toolbar to a new position with the left mouse button pressed. The toolbar will automatically be docked at the edge of the window if you drag the toolbar to the edge of the application window.

3.1.2.1 Standard

The **Standard** toolbar provides standard functions such as open, save, print and list boxes for tool lists and apertures/tools. Almost all of these functions can be called from the [File](#) and [Edit](#) menu as well.



Figure 92. Standard toolbar

New	Create a new document.	See New
Import	Import production data into the active document.	See Import
Open	Open an existing document.	See Open
Save	Save the active document.	See Save
Cut	Cut the selection and put it into the Clipboard.	
Copy	Copy the selection and put it into the Clipboard.	
Paste	Insert the Clipboard contents.	
Print	Print the active document.	See Printing
Print preview	Display full pages.	
Undo	Undo the last actions. Maximum number of actions is defined in: General: Undo Depth in Settings -> General Settings menu.	
Redo	Reverts the previously undone action.	
ApeTutor	Active configuration (tool list) indicates the active configuration/tool list of the active config-item/aperture/tool.	
D10	Active config-item/aperture/tool indicates the active config-item / aperture / tool that is either the one of the selected graphic object, or that is used to create new graphic object. In the case that several graphical objects are selected based on different config-items the field is empty, when changing the active config-item, the actual selected graphic objects are changed for this. Note: By clicking right mouse button on the active config-item Aperture/Tool Properties dialogue will appear. See also Aperture/Tool	



3.1.2.2 Zoom

The **Zoom** toolbar provides the buttons to zoom in/out the graphic view and to set the layer properties. All functions can be called from the [View](#) menu as well.



Figure 93. Zoom toolbar

Zoom window	Fits the selected area to the screen.
Zoom out	Reduces the visible drawing by the increments of 50%
Zoom scale	Zoom the drawing to a predefined scale factor.
Zoom in	Enlarges the visible drawing by the increments of 200%
Overview	Fit the drawing to the screen.
Zoom selected	Fit selected objects to the screen.
Refresh	Refresh all views.

3.1.2.3 Select

The **Select** toolbar provides buttons to select graphical objects. All functions can be called from the [Select](#) menu as well.



Figure 94. Select toolbar

Select Layer	Select all objects of the active layer.	See Select layer
Select Aperture/Tool	Select all flashes with the same apertures/tools on all layers.	See Select aperture/tool
Select All	Select all graphical objects on all layers.	See Select all
Invert Selection	Select all not selected graphical objects and deselect previously selected objects.	See Invert Selection
Select Path	Paths may be selected if this function is enabled (framed menu icon means enable).	CTRL + click on a button enables selection of a particular object type. SHIFT + click on a button enables selection of all object types.
Select Polygon	Polygons may be selected if this function is enabled (framed menu icon means enable)	
Select Circle	Circles may be selected if this function is enabled (framed menu icon means enable)	
Select Rectangle	Rectangles may be selected if this function is enabled (framed menu icon means enable)	

	enable)	
* Select Pad/Hole	Pads/Holes may be selected if this function is enabled (framed menu icon means enable)	
* Select Sub-layout	Sub-layouts may be selected if this function is enabled (framed menu icon).	
* Size Limit...	Restrict size for selected objects.	See Select
= New Selection	Reset previous selections in the event of a new selection.	
+ Add to Selection	Add a new selection to the selection group.	
- Remove from Selection	Remove an existing selection from the group.	

3.1.2.4 Insert

The **Insert** toolbar provides buttons used to add graphical objects. All functions can be called from the [Insert](#) menu as well.



Figure 95. Insert toolbar

* Insert Open Path	Insert a new open path that consists of one or more lines or arcs segments.	See Open Path
* Insert Closed Path	Insert a new closed path (the last path point will be joined to the start point).	
* Insert Polygon	Insert a new polygon that is a filled area, defined by an outline.	See Polygon
* Insert Rectangle	Insert a new rectangle.	See Rectangle
* Insert Pad	Insert a new pad (flash).	See Pad
* Insert Circle	Insert a circle with a fixed diameter.	See Centre + Radius method on page Circle .
* Insert Circle	Insert a circle dynamically defining radius.	See Centre method on page Circle .
* Insert Circle	Insert a closed path circle with a fixed diameter.	See Circle in closed path method on page Circle .
* Insert	Insert a new arc, where the circle sector is determined by defin	See Arc

Arc	ing the centre point of the circle.	
Insert Arc	Insert a new arc, where an additional point on the arc determines the circle sector.	See Arc
Insert Text	Insert a new text.	See Text
Insert Data Matrix	Inserts a Data Matrix barcode.	See DataMatrix
Insert Sublayout	Inserts a sub-layout (selected instance) in the current active layout.	
Line Mode 45°	Aligns the segments and arc segments so that the mouse cursor only moves by an angle of 45 degrees and increments. Only active while drawing.	See Line Drawing Modes
Line Mode 90°	Aligns the segments so that the mouse cursor only moves by an angle of 90 degrees and increments. Only active while drawing.	See Line Drawing Modes
Select inserted object	If this option is <i>on</i> , the inserted(pasted) objects are selected in addition to the previous selected ones. If this option is <i>off</i> the inserted(pasted) objects are not selected or active.	

3.1.2.5 Layer

The **Layer** toolbar presents buttons to customize the display for layers.



Figure 96. Layer toolbar

Layers	Show the Layers list.	See Layers
Active Layer	Select a layer to activate.	
Move Layer to Zero	Move an active layer to zero.	
Match up Layers	When this button is enabled (framed icon means enabled), select a flash (source layer and its point) by a mouse click, move the cursor to another flash in the destination layer (rubber band shows the road from the first point). This option works on flashes only (the nearest flash is selected).	

3.1.2.6 Grid / Unit

The **Grid/Unit** toolbar provides buttons to customize; the coordinates, grid, cursor grid and measurement units. All functions can be called from the [Edit](#), [Modify](#) and [Display](#) menu as well.



Figure 97. Unit / Grid toolbar

 Set Zero Point	Set Zero Point	
 Absolute Coordinates	Mouse position in absolute coordinates.	
 Relative Coordinates	Mouse position in coordinates relative to the Anchor Point..	
 Cartesian Coordinates	Mouse position in Cartesian coordinates.	
 Polar Coordinates	Mouse position in polar coordinates.	
 Measurement Unit	Select a measurement unit.	
 Decimal Places	Change the number of decimal places that a unit is measured in.	
 Toggle Grid	Show/hide the grid on the graphic window.	
 Grid Spacing	Change the spacing between grid lines.	The field with the drop down list is activated after the Toggle Grid button is pressed.
 Snap to Grid	Turn on and off the snap to grid feature.	
 Cursor Grid Spacing	Select grid spacing for the positioning of the graphical objects.	The field with the drop down list is activated after the Snap to Grid button is pressed.
 Move Point/Segment	Move Point/Segment.	See Move Point/Segment
 Add Point/Segment	Add or Append.	See Add Point/Segment
 Cut Point/Segment	Cut into segments.	See Cut Point/Segment
 Delete Point/Segment	Delete Point/Segment.	See Delete Point/Segment
 Create Move Arc	Create or Move Arc.	See Create or Move Arc

3.1.2.7 Relative Rotations

The **Relative Rotations** toolbar provides buttons to rotate and mirror the selected objects around the anchor point. All functions can be called from the [Edit](#) menu as well.



Figure 98. Relative Rotations toolbar

 Value	Enter an angle for a relative rotation.
 90°	Rotation angle 90°
 180°	Rotation angle 180°
 270°	Rotation angle 270°
 Mirror horizontal	
 Mirror vertical	

3.1.2.8 Flash Rotations

The **Flash Rotations** toolbar provides buttons to rotate and mirror selected objects around its (0,0) points (location of flashes is not changed). All functions can be called from the [Edit](#) menu as well.



Figure 99. Flash Rotations toolbar

 Value	Enter an angle for a flash rotation.
 0°	Rotation angle 0°
 90°	Rotation angle 90°
 180°	Rotation angle 180°
 270°	Rotation angle 270°
 Mirror horizontal	
 Mirror horizontal + 90°	
 Mirror vertical	
 Mirror vertical + 90°	

3.1.2.9 Full Screen

The **Full screen** toolbar consists of one button, which switches on or off full screen mode (opens the graphic representation view to the full monitor size).



Figure 100. Full screen toolbar

3.1.2.10 GMC

Buttons used to perform various operations to change geometry of pads.



Figure 101. GMC toolbar

 GMC	Open the Geometry Manipulations Centre (More details; Geometry Manipulation Centre).	Button is activated if some objects are
---	---	---

		placed on <i>SolderPasteTop_Mark</i> layer.
▶ To Mark	Move selected objects from the <i>SolderPasteTop_Work</i> layer to the <i>SolderPasteTop_Mark</i> layer (only elements on the <i>_Mark</i> layer can be edited by the Geometry Manipulation Centre).	
◀ To Work	Move selected objects from the <i>SolderPasteTop_Mark</i> layer to the <i>SolderPasteTop_Work</i> layer.	
▣ To Mark	Move selected objects from the <i>SolderPasteTop_Modified</i> layer to the <i>SolderPasteTop_Mark</i> layer.	
★ To Ready	Move all objects from the <i>SolderPasteTop_Mark</i> and <i>SolderPasteTop_Modified</i> layers to the <i>SolderPasteTop_Ready</i> layer.	
◎ To Work	Move selected objects from the <i>SolderPasteTop_Ready</i> layer to the <i>SolderPasteTop_Work</i> layer.	
● Init	Copy all objects from <i>SolderPasteTop</i> layer to the <i>SolderPasteTop_Work</i> layer. All configurations used in the design are copied to a new configuration named <i>oldname_Work</i> . If <i>GerberDefault</i> was used in the design, a new configuration <i>GerberDefault_Work</i> is created.	

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.1.3. Status Bar

The status bar at the bottom of the application window, displays information regarding an object or a function selected.



Figure 102. Status bar

The status bar is divided into four information areas:

1. Textual information:
 - what is selected (nothing/a few objects/open path/closed path/ rectangle and etc.); if only one object is selected, the current position of the object (numbering) is shown at the beginning (#22);
 - in case of a path, the additional format (*total_number_of_points:current_number, Segment_type*) and the path width are shown.
2. Number of selected objects;
3. Current coordinates of the cursor;
4. Coordinates of the anchor point ×.



Note: Double click on this area to copy coordinates of the anchor point to Clipboard.

A user can change visibility of the status bar by selecting the **Status bar** entry from the **Toolbars** sub-menu of the **View** menu.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.1.4. View arrangement

A user has a possibility to change the layout of views in *CircuitCAM* according his needs:

- Dock view to editing frame,
- Move view outside the *CircuitCAM* window,
- Auto-hide views,
- Reset layout of views.

3.1.4.1 Dock View

View can be fastened to one side of *CircuitCAM* frame or to a location in the editing frame.

- Click with a mouse on a view title.
- Drag the view into the middle of *CircuitCAM* window. The guiding arrows appear on the screen.
- To position the view, move it over the guide arrows. The destination is as shaded.

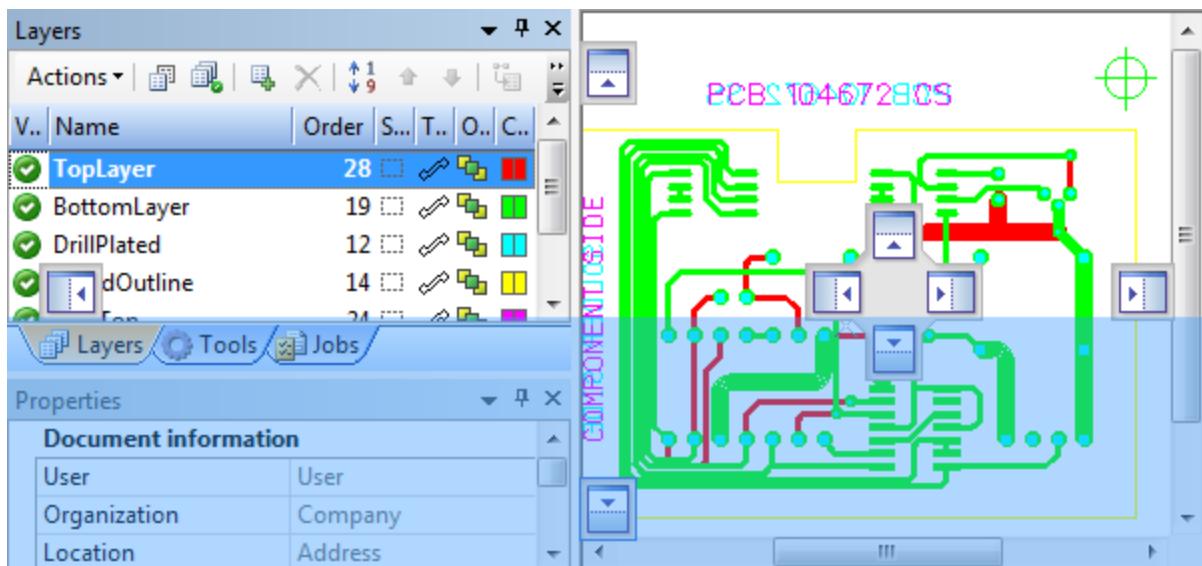


Figure 103. Guiding arrows and shaded designation area

- Release the mouse button, when the view is at the desired place.

3.1.4.2 Move View Outside CircuitCAM Window

- Click with a mouse on a view title.
- Drag the view outside the *CircuitCAM* window.
- Release the mouse button.



Note: To return the view back to the initial position, double click on the title of the view.

3.1.4.3 Auto-hide View

When a view is auto-hidden, its name and icon are displayed on a tab at the edge of the *CircuitCAM* frame.

- Select a view to auto-hide.
- Select the **Autohide** item in the view menu or press the button with a pushpin icon.

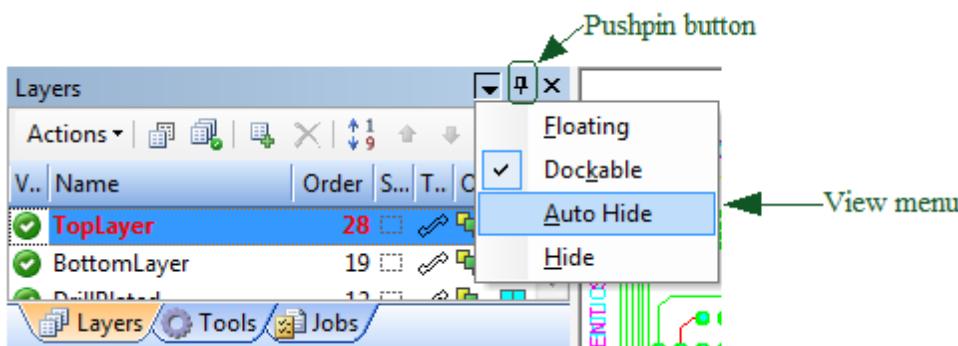


Figure 104. Auto Hide item in view menu and pushpin button

Tab of auto-hidden view is displayed at the edge of the *CircuitCAM* frame. To use the window again, move the mouse pointer over the tab so that it slides back into view.

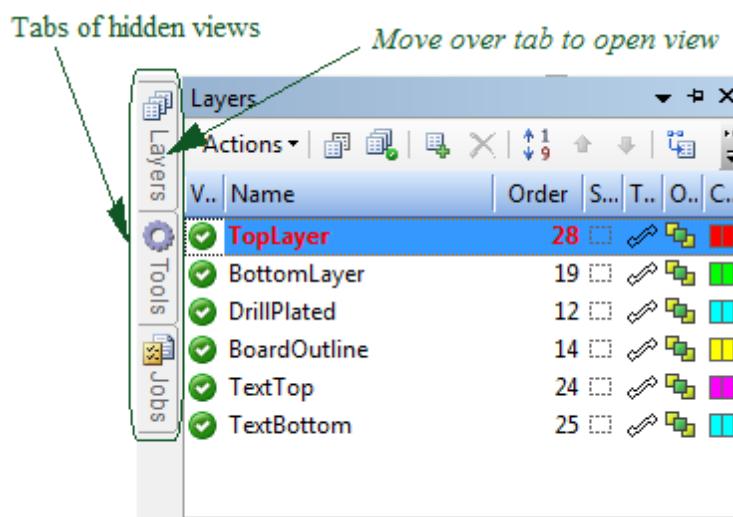


Figure 105. Tabs of hidden views

To turn off auto-hide mode, perform the steps listed above.

3.1.4.4 Reset Layout

To reset views to the default location, select the **Settings** menu, **General Settings -> Reset to Defaults** the click the checkbox of **Docking Panes**.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.1.5. Message Log

The **Message Log** window displays various log/report messages generated by *CircuitCAM* after performing various operations. The text of the message log can be selected (**Ctrl + point on the line** with a mouse), copied (button), removed (button) and saved (button).

The message log window:

- Shows a log of functions such as import of the data;

For the import example shown below, the log indicates the path of the imported file and the file type (e.g. Gerber, Excellon, G-Code,...etc), also shows the used layers, configuration and tool/aperture list, together with the number and length of usages of each tool.

Message Log		
Tool/Aperature	Flash	Path
D10	48	0
D11	0	523
D13	150	0
D15	40	0
D18	0	4
D113	0	100
D118	0	280
D121	0	112
D122	0	12
D123	0	0

Figure 106. Message log view, import log

- Shows a log of production data export;

For the export example shown below, the log indicates the used layers, phases, objects, number of objects, their perimeter, moves, tool usage ,the number of tools used, flashes, paths, segments and length and at the bottom of the log export directory, job type and execution time are stated.

Message Log		
Tool/Aperature	Flash	Path

Layer = BoardOutline, Phase =		
Directly realized objects: Closed Shapes, Rectangles, Circles (n...		
No. of objects 0		
Perimeter [mm] = 0		
Moves [mm] = 0		
Tool usage :		
No.	Flashes	Paths
3	0	1
C:\Program Files (x86)\CircuitCAM 6.4\ExampleData\Prototyping\Tu...		
=====		
Execution of job GerberXTop was finished in 00:00:03		

Figure 107. Message Log view, export log

- Displays characteristics/errors of others *CircuitCAM* functions (e.g., time which was used for import);
- Displays error messages of many others *CircuitCAM* functions.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.1.6. General Settings

The **General Settings** dialogue window allows the user to change main user and document related preferences of *CircuitCAM*. This dialogue window can be called from the **Settings** menu.

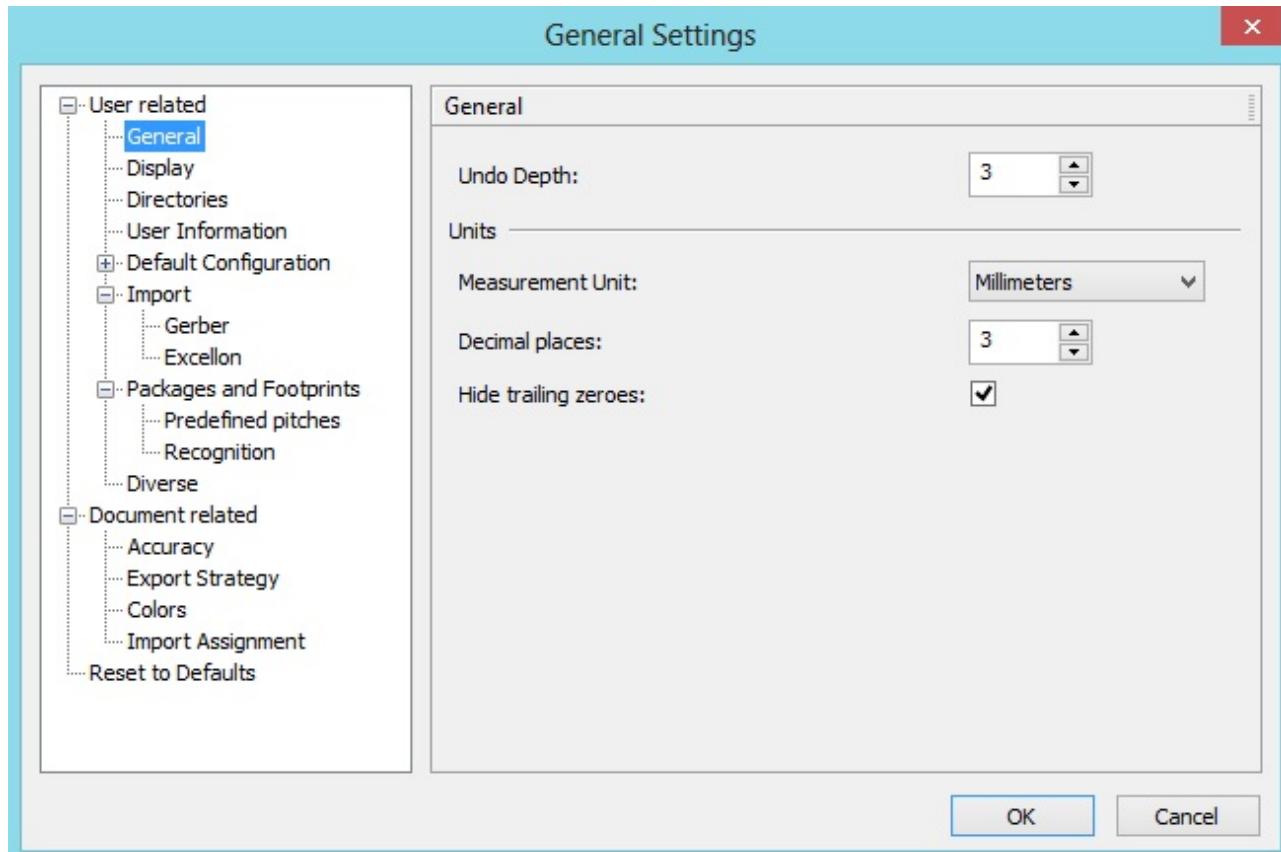


Figure 108. General Settings dialogue window

3.1.6.1 User Related

The **User related** section allows the user to customize *CircuitCAM* software.

General Panel

- **Undo Depth:** configures the size of the command stack to return an object to the state it was in before the last action.
- **Measurement Unit:** defines the measurement units from the predefined list of units (drop down list).



Figure 109. Measurement unit drop down list

- **Decimal places:** specifies a number of decimal places that a unit is measured in (use up and down arrows next to the field to increase and decrease numbers).
- **Hide trailing zeroes:** if *On*, trailing zeroes are hidden.

Display Panel

Helps to customize display settings:

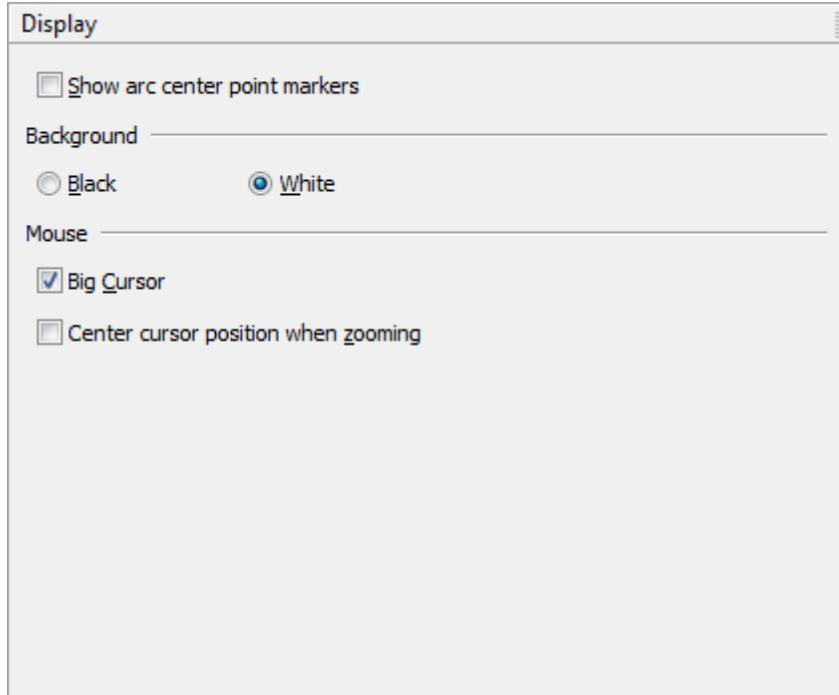


Figure 110. Display panel

- **Show arc centre point markers:** if *On*, the arc centre point marker appears on the screen after the arc is selected.
- **Background** of the graphic area can be set to **Black** or **White** by setting *On* a particular check box.
- **Big Cursor:** if *On*, the mouse cursor appears as two perpendicular lines on the graphic area.
- **Centre cursor position when zooming:** if *On*, the zoom centre is a cursor position.

Directories Panel

Helps to customize folders for template, translation files:

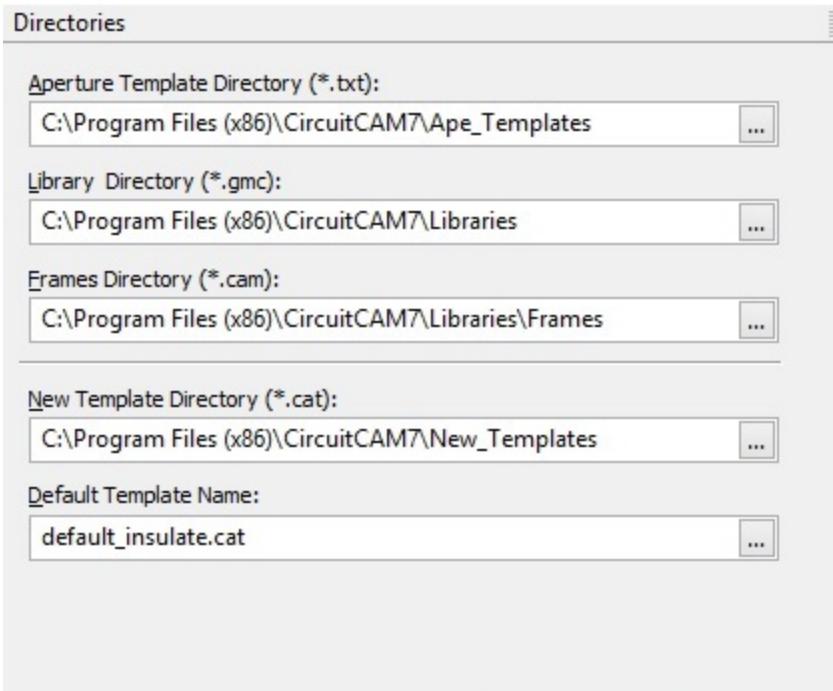


Figure 111. Directories panel

- **Aperture Template Directory (*.txt):** for the aperture template (translation).
- **Library Directory (*.gmc):** for the library directory for GMC, Library exchange and Package recognition.
- **Frame Directory (*.cam):** for the Frame Generator **Graphic from File** mode.
- **New Template Directory (*.cat):** for the *CircuitCAM* template [CAT](#) files.
- **Default Template Name:** The default template is used when *CircuitCAM* is started with no specified CAM document. In this case the document is named "Untitled N" (N = 1,2,3...) and based on the default template. Also when creating a new CAM document without specifying a template file the default template is used. During installation the default template is set to default_insulate.cat. User can specify another template file as needed.

User Information Panel

Allows a user to personalize *CircuitCAM* software.

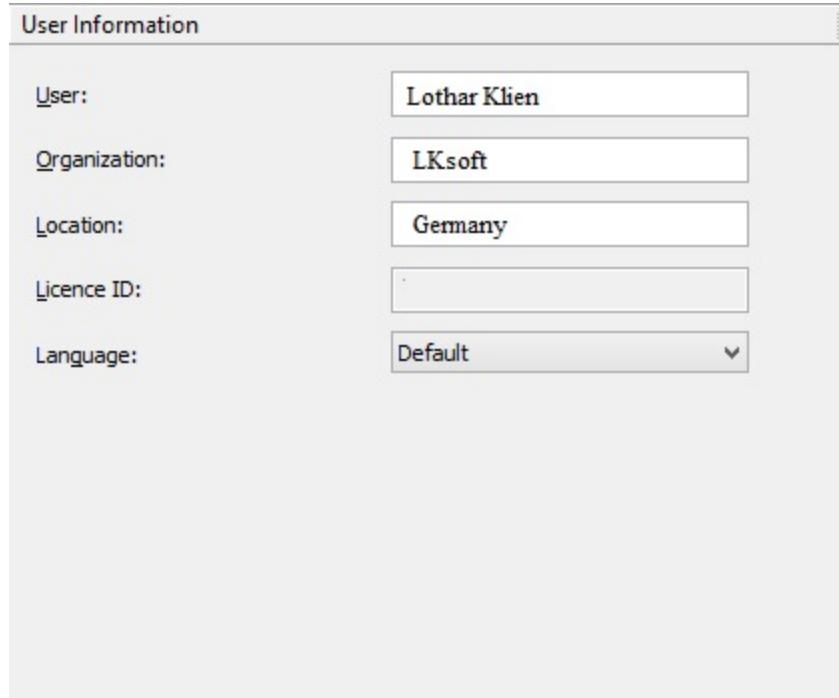


Figure 112. User Information panel

- **User:** Specifies the user details (first name, last name).
- **Organization:** Specifies details of the user's organization.
- **Location:** Specifies a location of the organization.
- **Enabling Number (Licence ID):** Contains the enabling number, which is entered together with the serial number, during the installation process.
- **Language:** Specifies a language for *CircuitCAM* software. User can either select one of already existing languages from the list or adapt *CircuitCAM* to his vernacular. For more details see Language support.

Default Configuration

Default configurations/tool list are needed during export, in the case that no specific configuration/tool list is defined, see [export task](#) under "create new aperture/tool list". The information provided in the default configurations is identical to the information under [tool list](#) properties. The default properties are defined for each format independently.

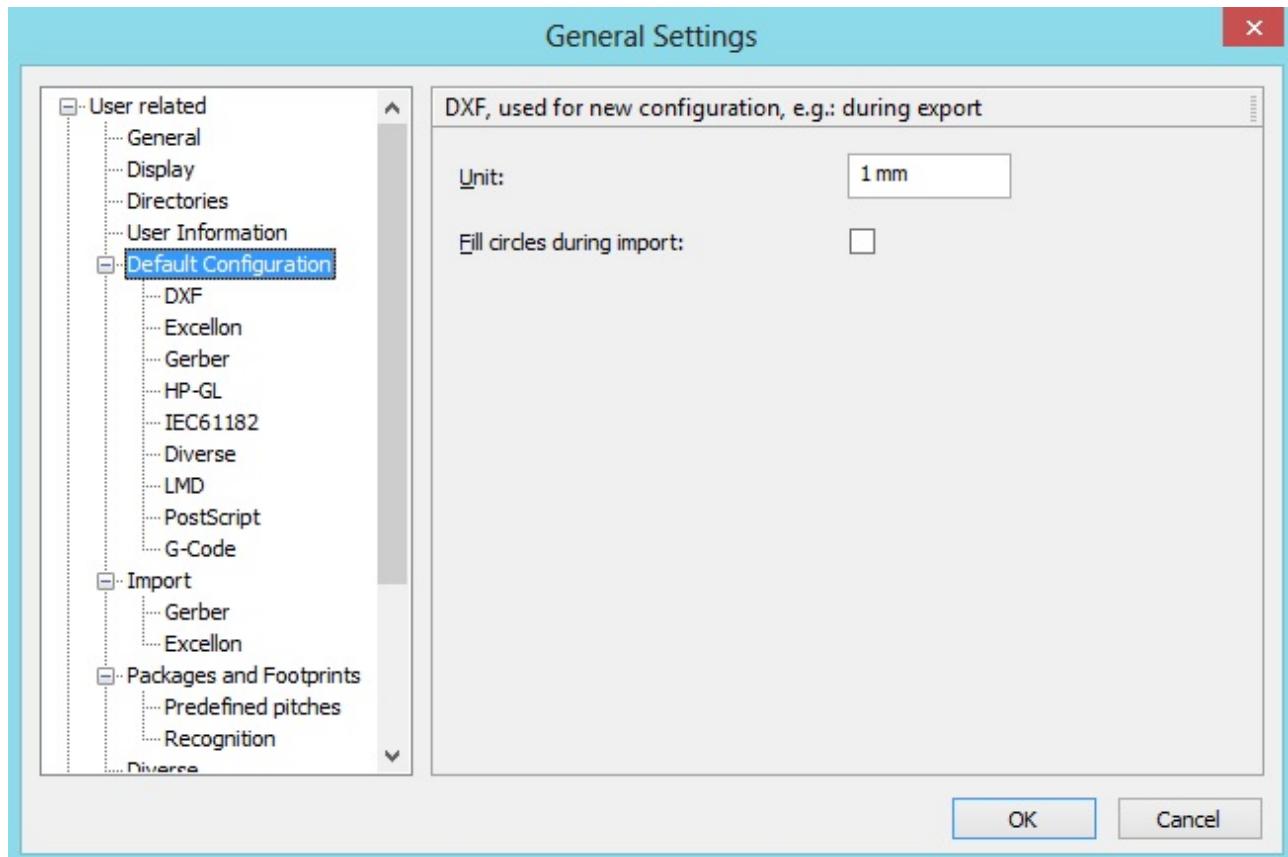


Figure 113. Default Configuration dialogue window

Gerber Import Panel

When *CircuitCAM* is started, **Gerber Import** options are taken from the registry. Every time during opening a new document (**File -> New**), the **Gerber Import** options are read from the corresponding CAT file (if these options are already in this CAT file). The user can change these options either in the **General Settings/Gerber Import** by writing them into the CAT file or in the **Import** dialogue window without writing these updates into CAT file. In both cases the current values will be saved into the registry while closing *CircuitCAM*.

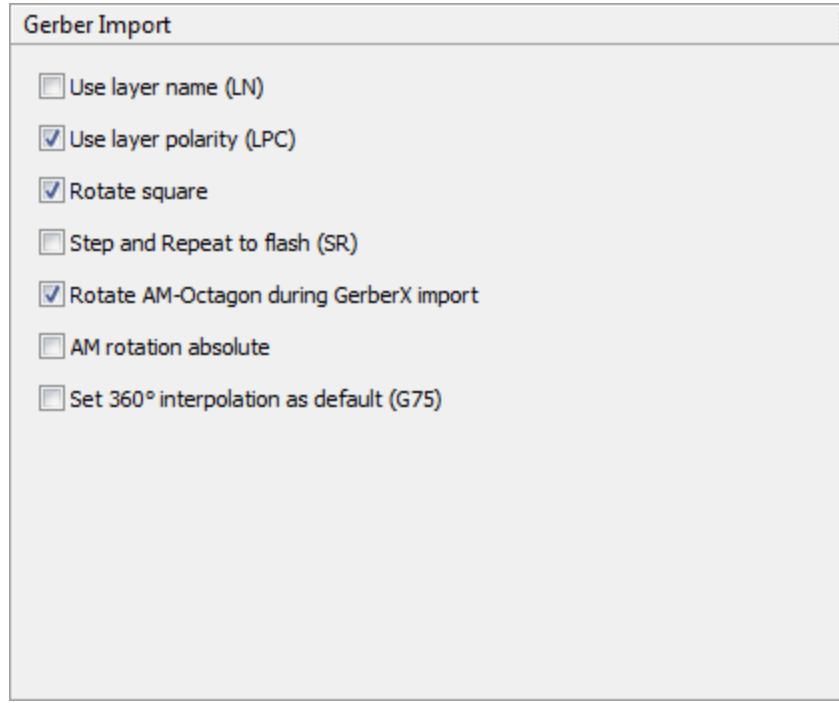


Figure 114. General Import panel

- **Use layer name (LN):** if *Off*, always use the layer names from LN commands (despite the fact it was used only once);
- **Use layer polarity (LPC):** a possibility to ignore LPC/LPD commands.
- **Rotate square:** *On* = old mode, when aperture is rotated during a drawing process; *Off* = path is drawn as a polygon and its width depends on a rotation angle (the aperture always remains in the zero rotation).
- **Step and Repeat to flash (SR):** if *On*, repeated group is Special flash; *Off* - is similar to the denested flash.
- **Rotate AM-Octagon during GERBER import:** if *On*, an octagon is rotated by 22.5 degrees to compare with others *Aperture Macro* (AM) primitives of type **Polygon**. It means that AM octagon is converted into *Aperture Definition* (AD) regular octagon.
- **AM rotate absolute:** if *Off*, *CircuitCAM* rotates every AM primitive around the point 0/0. If *On*, *CircuitCAM* rotates every AM primitive around its reference point according to this table (primitive Reference point): 2/20 - start point, 21 - centre point, 22 - lower left point, 4 - start point, 5,6 and 7 - centre point.
- **Set 360° interpolation as default (G75):** with *Off* the quadrant circular interpolation is used.

Excellon Import Panel

This panel allows a user to modify import parameters for *Excellon* file and save modifications in the current template file.

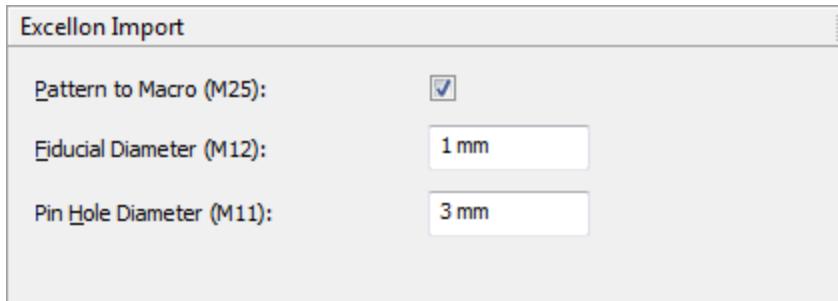


Figure 115. Excelon Import panel

- **Pattern to Macro (M25):**
- **Fiducial Diameter (M12):**
- **Pin Hole Diameter (M11):**

Predefined Pitches Panel

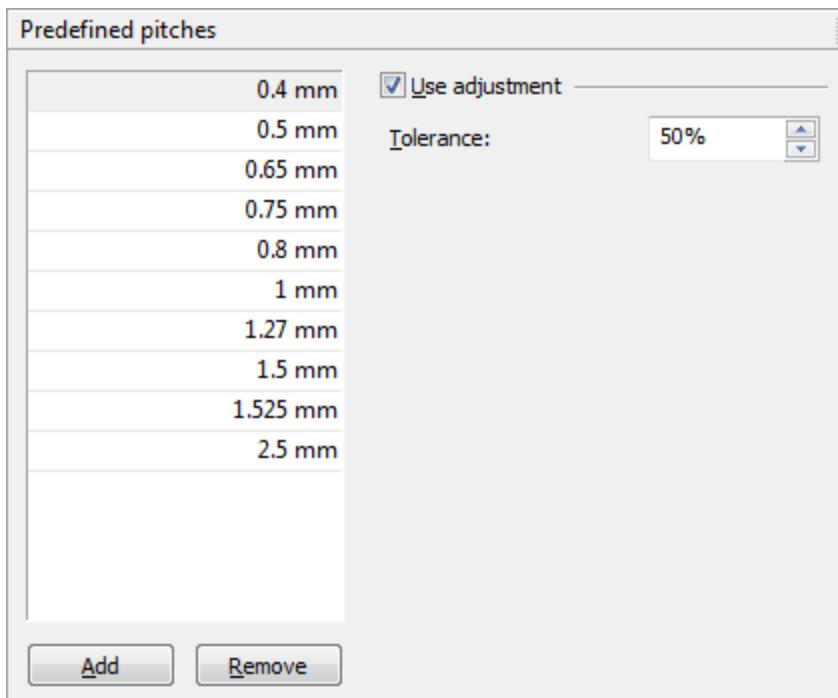


Figure 116. Predefined pitches panel

The combo box, that lists some pre-defined values of typical pitches. Some default values are presented in the list, but a user can enter own pitches as well.

Recognition Panel

This panel helps to control the applying of footprints after package recognition procedure.

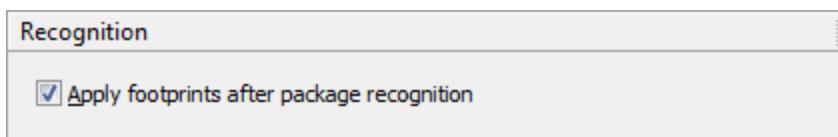


Figure 117. Recognition panel

Apply footprints after package recognition: if ON, *CircuitCAM* will perform apply footprints after each package is recognised.

Diverse Panel

Allows specification of the following parameters:

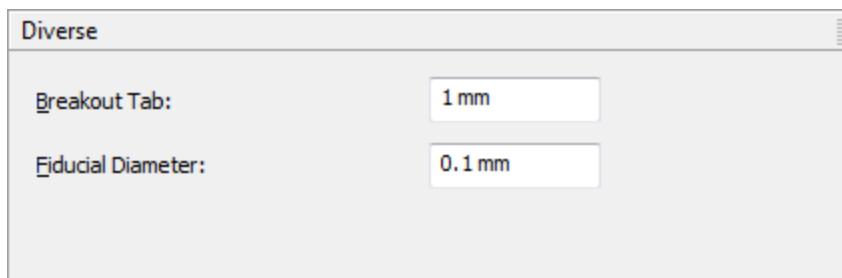


Figure 118. Diverse panel

- **Breakout Tab:** for the [Create Gap](#) function.
- **Fiducial Diameter.**

3.1.6.2 Document Related

The **Document Related** section helps to customize document parameters. If the option **Save to default template file** (*default_insulate.cat*) is set *On*, the modified settings will be saved to the selected template file.

Accuracy Panel

This panel helps to change the calculation parameters:

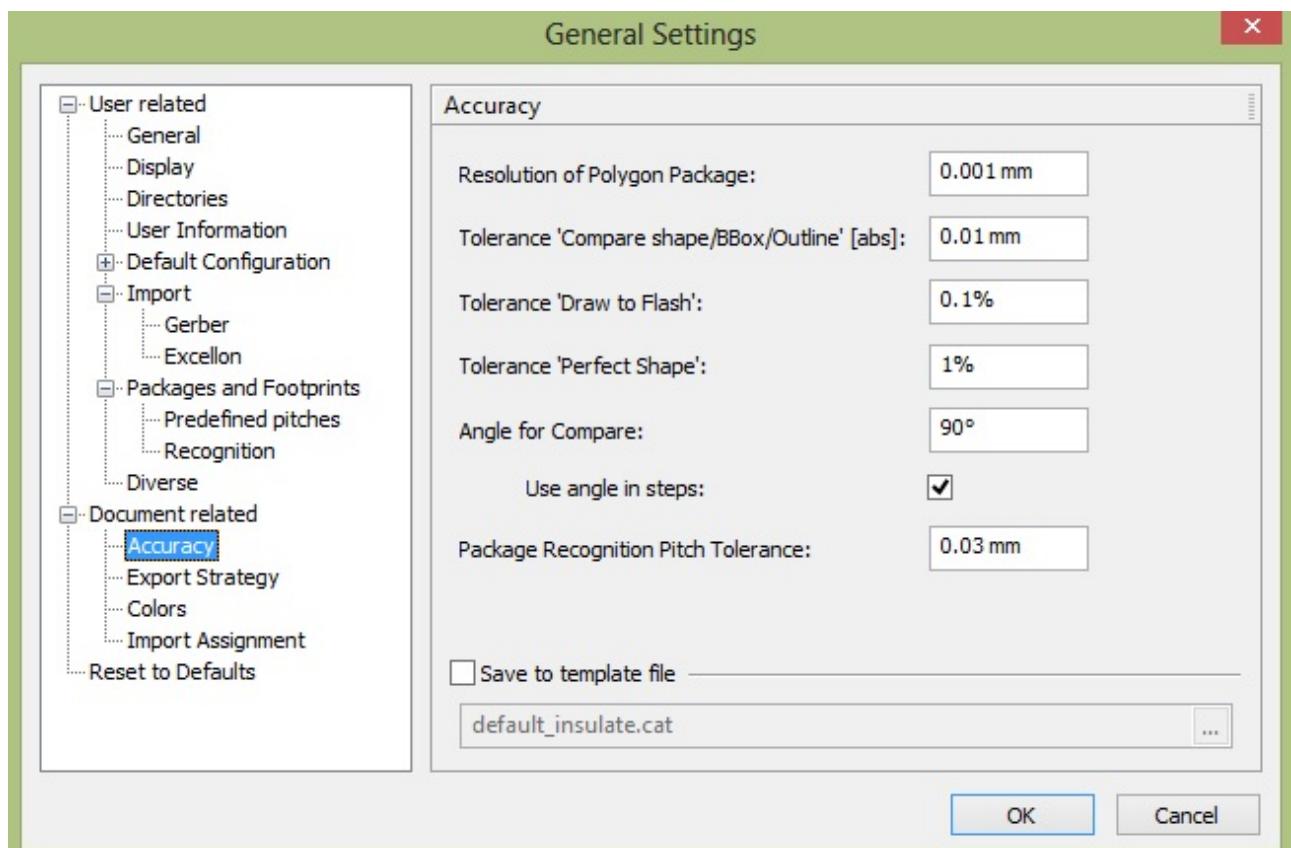


Figure 119. Accuracy panel

- **Resolution of Polygon Package.**
- **Tolerance 'Compare shape/BBox/Outline' [abs]:** for the [Compare Shape](#), [Compare Bounding Box](#) and [Compare Outline](#) functions.
- **Tolerance 'Draw to Flash' [%]:** for the [Compare Outline](#) and [Draw to Flash](#) functions.
- **Tolerance 'Perfect Shape' [%]:** for any testing on perfect shape function ([Shape Manipulations](#), [Draw to Flash](#), Convert selected shapes to polygons in GMC) .
- **Angle for Compare:** specifies either current compare angle (when flag **Use angle in steps** is **Off**) or rotation step (when **Use angle in steps** is **On**).
- **Package Recognition Pitch Tolerance:** for Package Recognition function.
- **Save to template file :** By default the generated file will be saved to the default template in the [Directory panel](#) that was created. (when flag **Save to template file** is **On** it will save to another *.CAM file) .

Export Strategy Panel

The panel allows specifying of a sorting strategy for a different type of packages during export. The following sorting types are available: XParallel, YParallel, XMeander, YMeander, SpiralInward, SpiralOutward, Local (sort shortest distance, but within the footprintOccurrence) and Global.

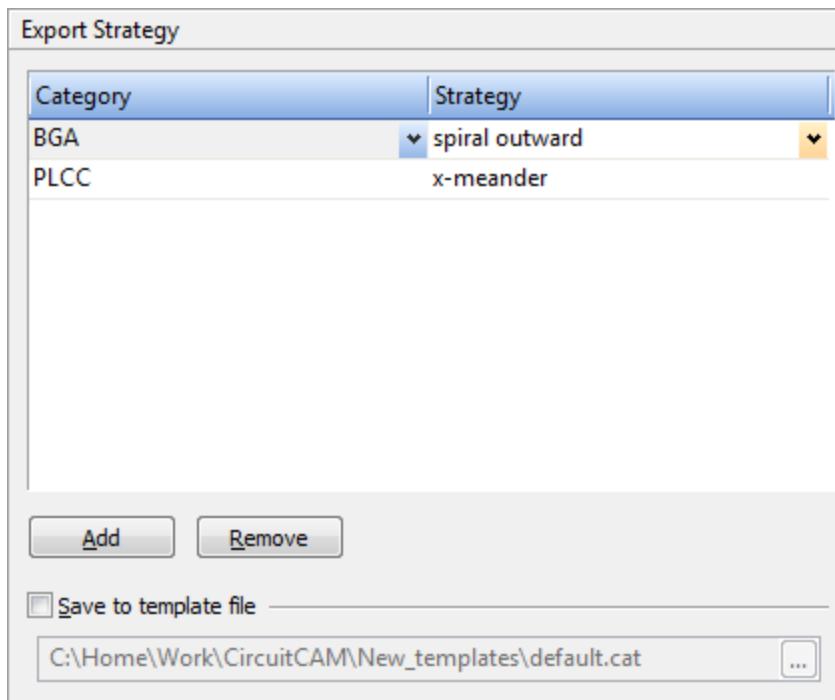


Figure 120. Export Strategy panel

- **Category:** specifies the type of a package.
- **Strategy:** specifies the sorting types.
- **Add:** a button to add a new table entry.
- **Remove:** a button to remove the selected table entry.

Colors Panel

The panel allows modifying of colors for selected and not selected objects.

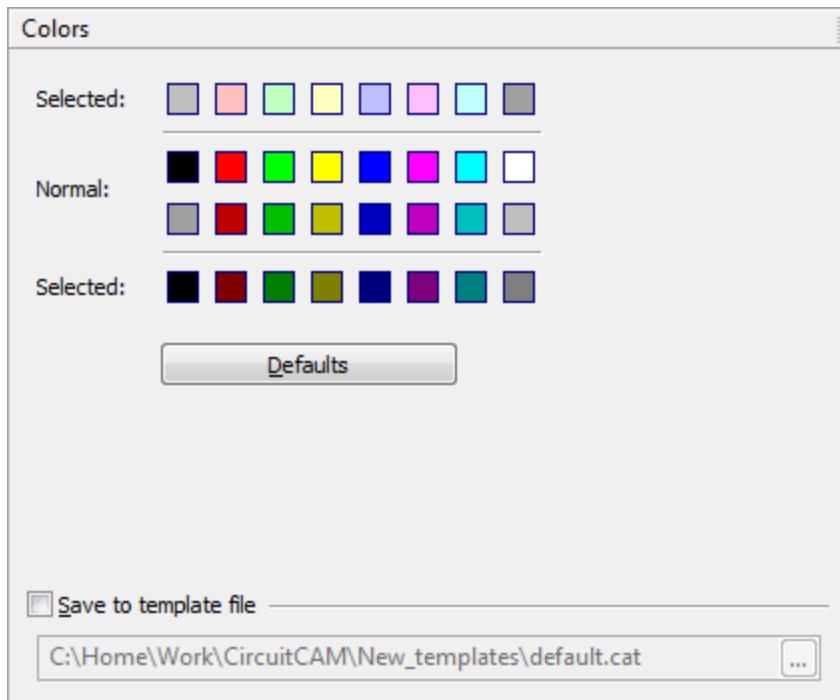


Figure 121. Colors panel

- **Defaults:** a button to restore colors to defaults.

Import Assignment Panel

If the production data are used with the specified file extensions, e.g.: *drl*, *sld*, *cmp*, *CircuitCAM* provides a possibility to add an import assignment to automate and simplify the import.

These import assignments can also be effectively used by the script task "Import"(see "[Script Tasks](#)") inside script job named "Default"(for details check "[Batch and Background Processing](#)").

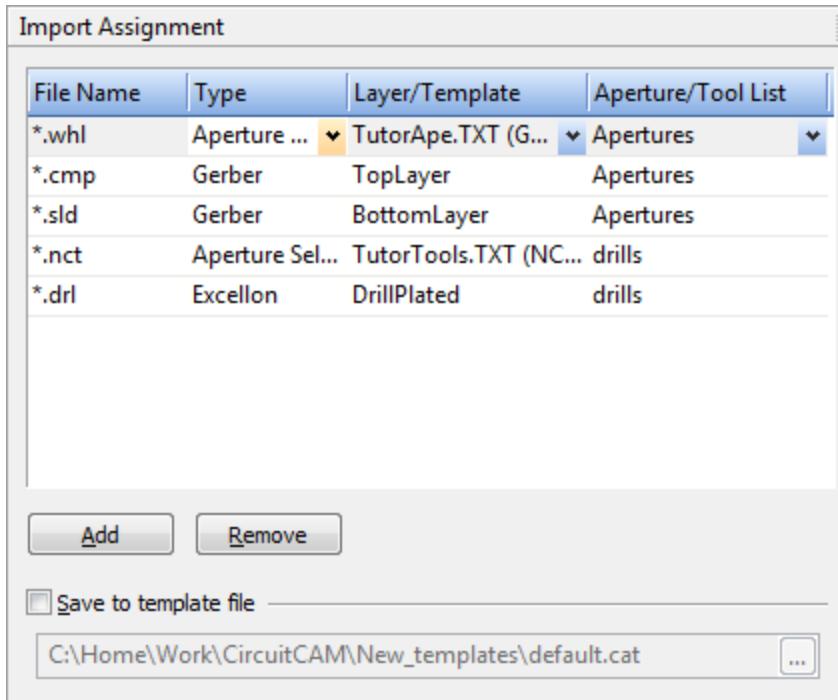


Figure 122. Import Assignment panel

- **Add** button: creates an auto row in the table, where new entries can be made.
- **File Name:** specifies a name or an extension of the file to be imported. See table below, how the import assignment should work (* - any character (one or many), ? - any character (one)):

File name	File-spec	show Import Dialogue
aaa.gbx	*.gbx	No
aaa.gbx	???.gbx	No
aaa.gbx	aaa.gbx	No
aaa.gbx	aaaa.gbx	Yes
aaa.gbx	bbb.gbx	Yes
aaa.gbx	aba.gbx	Yes
aaa.gbx	?a?.gbx	No
aaa.gbx	?b?.gbx	Yes
aaa.gbx	????.gbx	Yes
aaa.gbx	a*.*	No
aaa.gbx	a*.gb?	No
aaa.gbx	a*.g*	No
aaa.gbx	a*.g?	Yes
aaa.gbx	.gbx	Yes
aaa.gbx	*.?bx	No
aaa.gbx	aaa.gbr	Yes

- **Type:** specifies a type of the file.
- **Layer/Template:** specifies the matching layer or template for the specified file.
- **Aperture/Tool List:** specifies a name of the list to store the apertures or tools in *CircuitCAM*.
- An incorrect or redundant import assignment can be removed from the table by using **Remove** button.

3.1.6.3 Reset to Defaults

This panel allows a user to reset modified settings to *CircuitCAM* defaults.

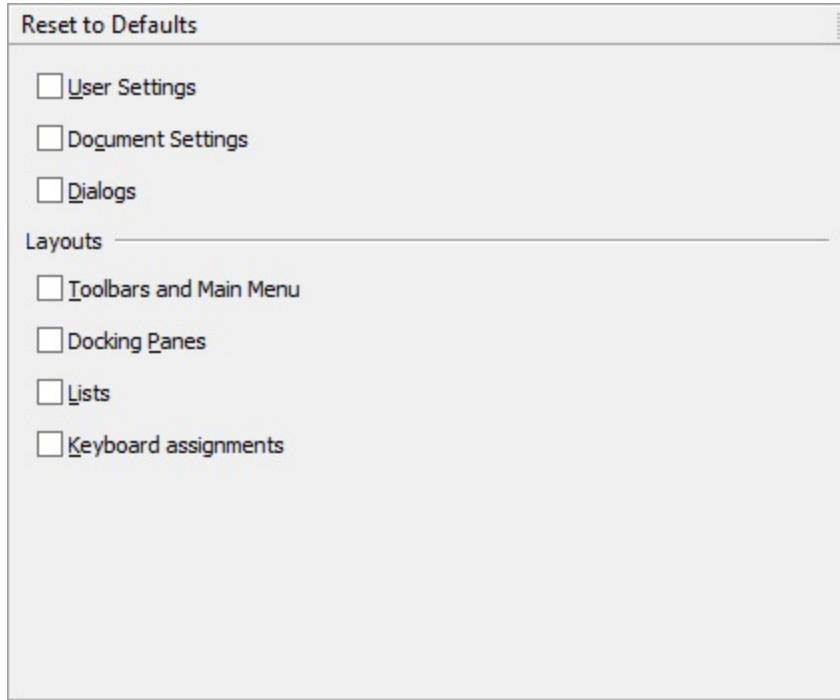


Figure 123. Reset to Defaults panel

The following settings can be updated, if the option is set *On*:

- User Settings.
- Document Settings.
- Dialogues.
- Toolbars and Main Menu.
- Docking Panes.
- Lists.
- Keyboard assignments

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.1.7. Customize

The **Customize** dialogue window helps a user to modify menu options, toolbars layout, the keyboard mapping, menu settings, and general preferences. The **Close** button quits the customizing.

To open the **Customize** dialogue window right click on any toolbar and choose **Customize**

3.1.7.1 Toolbars

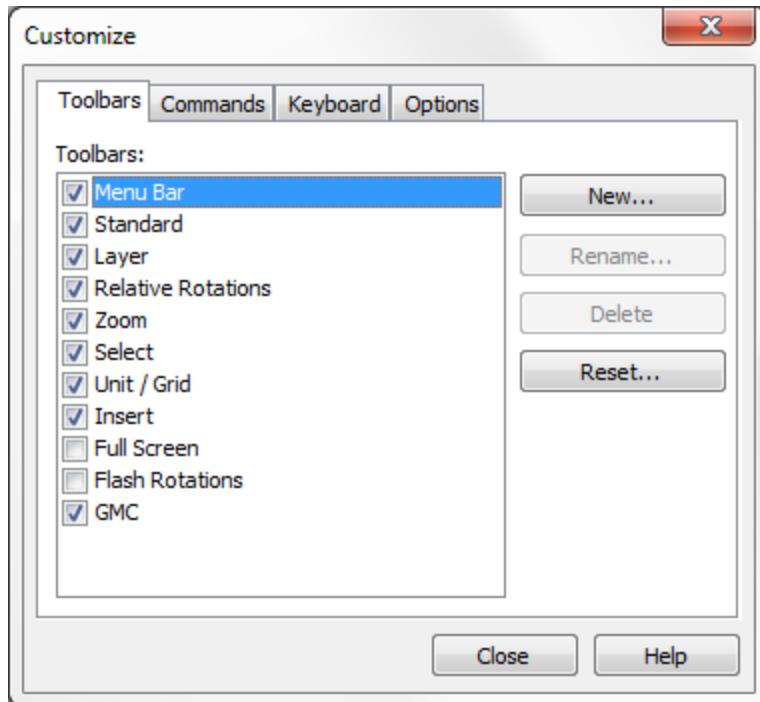


Figure 124. Customize dialogue window, Toolbars panel

- **Toolbars:** a list of available toolbars. If flag is set *On*, the particular toolbar appears on the *CircuitCAM* main window.
- **New button:** invokes the **New Toolbar** dialogue window to create a new toolbar:

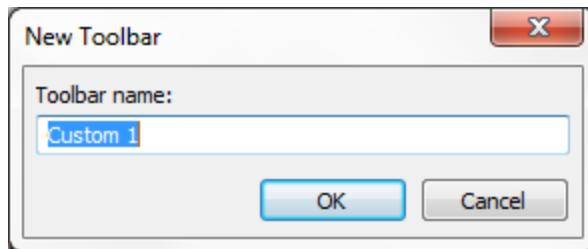


Figure 125. New Toolbar dialogue window

- A name of the toolbar should be entered into **Toolbar name** field. After the **OK** is pressed, the new toolbar appears in the **Toolbars** list.
- **Rename button:** invokes the **Rename Toolbar** dialogue window to rename the selected toolbar:

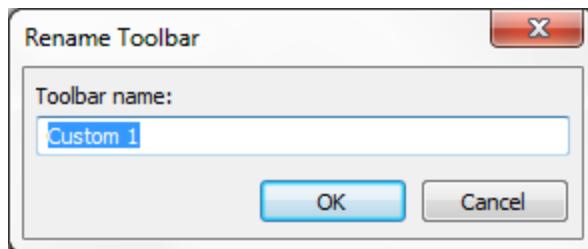


Figure 126. Rename Toolbar dialogue window

- A new name of the select toolbar should be specified into the **Toolbar name** field. The changes are confirmed with the **OK** button.

- **Delete** button: removes the selected toolbar.
- **Reset...** button: resets the default toolbar settings to the selected toolbar.



Note: For a predefined toolbars, such as **Standard**, **Layer** and etc., **Rename** and **Delete** buttons are disabled. The modifications can be applied only for the user created toolbars.

3.1.7.2 Commands

A new command can be added to the existing toolbar, menu or submenu.

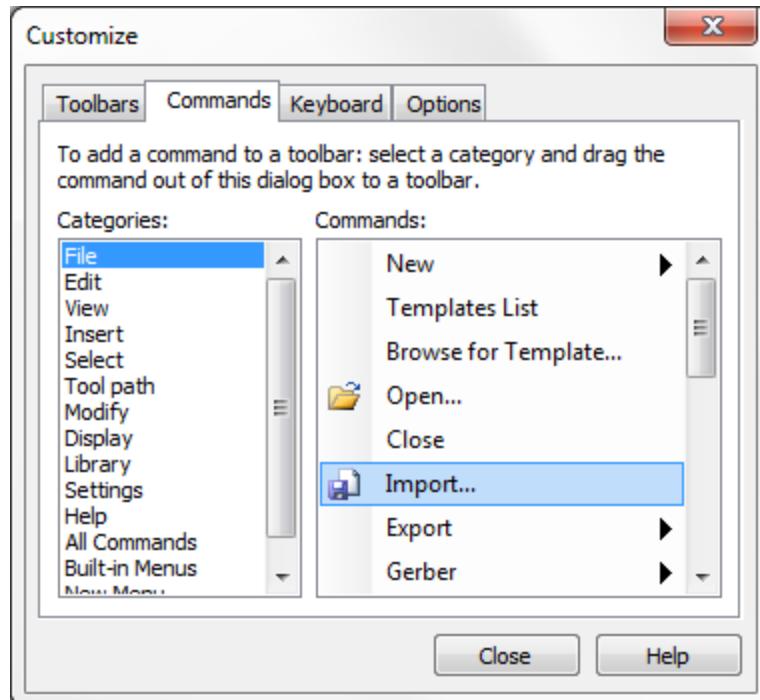


Figure 127. Customize dialogue window, Commands panel

- **Categories:** a list of the available categories.
- **Commands:** a list of available commands for the selected *Category*.

To add a command to a toolbar/menu: select a category and drag the command out of the dialogue window to the preferable toolbar.

3.1.7.3 Keyboard

Allows a user to create, modify and load from the file a set of function key assignments for **A-Z**, **1-9**, **F1-F12** and their combinations with **Alt**, **Shift**, **Ctrl**, **Alt+Shift**, **Alt+Ctrl**, **Shift+Ctrl** and **Alt+Shift+Ctrl**. Created keyboard shortcuts are not case sensitive, e.g.: Characters 'a' or 'A' are the same 'A'. Created shortcuts are stored in XML files. The **Keyboard** panel has following controls:

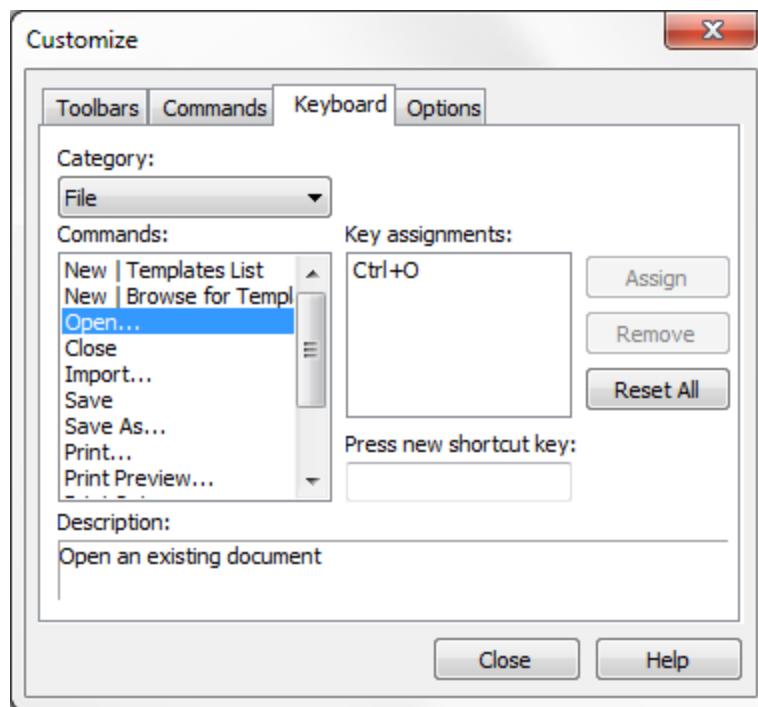


Figure 128. Customize dialogue window, Keyboard panel

- **Category:** a list of the available categories.
- **Commands:** a list of commands for which the shortcut can be created.
- **Key assignments:** shows already assigned shortcuts for the selected *Command*.
- **Press new shortcut key:** Click **Assign** to add the definition from the **Press new shortcut key** field to the **Key assignments** field.
- **Description:** displays a brief description of the currently selected command.
- **Assign** button: assigns the new shortcut. It automatically appears in the **Key assignments** list.
- **Remove** button: removes the selected shortcut. Select a key assignment from **Key assignments** field and click **Remove** to delete it.
- **Reset All** button: loads the default shortcut file.

3.1.7.4 Options

The **Options** panel has the following controls:

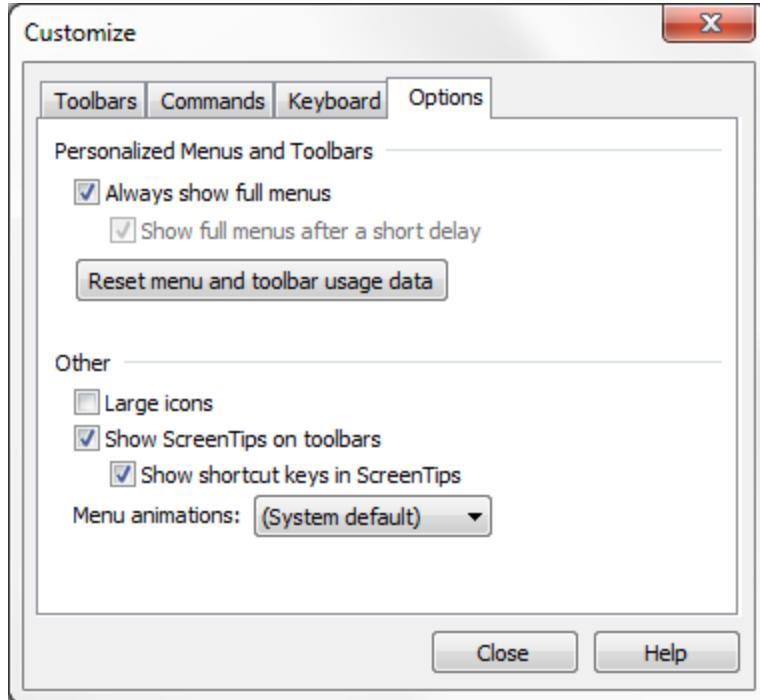


Figure 129. Customize dialogue window, Options panel

Personalized Menus and Toolbars group

- **Always show full menus:** If *On*, full menus are displayed.
- **Show full menus after a short delay:** If *On*, the selection of the commands is displayed in menus. However, after a short delay, the full menu appears.
- **Reset menu and toolbar usage data:** Restores the default set of visible commands on the menus. A click on the button does not undo any customizations you have done.
- **Large icons:** If *On*, enlarges the icons on the screen to make them easier to see or to access.
- **Show Screen Tips on toolbars:** If *On*, displays a short help text when you place the mouse pointer over a toolbar button.
- **Show shortcut keys in Screen Tips:** If *On*, the possible keyboard shortcut is displayed in addition to the short help text.
- Use the **Menu Animations** drop-down menu to select the type of menu animations. The available options are 'System default', 'Random', 'None', 'Unfold', 'Slide', and 'Fade'.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.1.8. Online Help

CircuitCAM provides a context sensitive help and an online help.

The context-sensitive help can be called directly with **F1**.

The online help is presented in a separate window, where you can find a **Help content** and a **Help index** to find an applicable topic or term quickly.

To invoke the online help, select in the main menu **Help ---> Help Topics**.

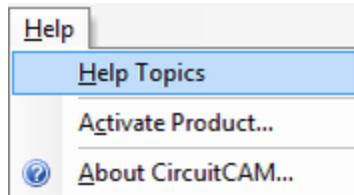


Figure 130. Help -> Help Topics menu

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.1.9. Preview / World View

The preview displays either the graphics of the selected tool in the **Tools** view, or the selected library item in the **Libraries** view, if these views have the focus. Otherwise it displays the world view of the current layout.

The **Preview** window is set **Default** to appear in the bottom right corner of the window, to switch **Preview** on/off select in the main menu **View -> Preview**.



Note: The same window is used for *world view* and *preview*. All depends on the currently focused context.

By clicking on the mouse in the preview area, world view window will focus on the clicked area, if the mouse is still clicked on and dragged all over the preview the main view will move and be dragged as the preview area.

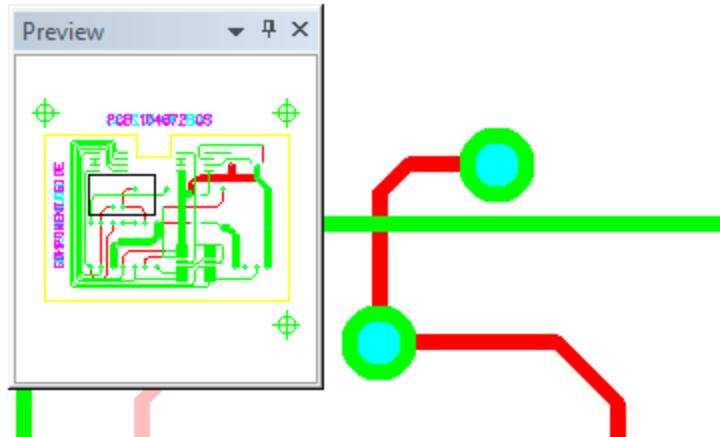


Figure 131. World view of Graphic view

For example, the preview of a tool is visible if the tool is focused in the **Tools** view.

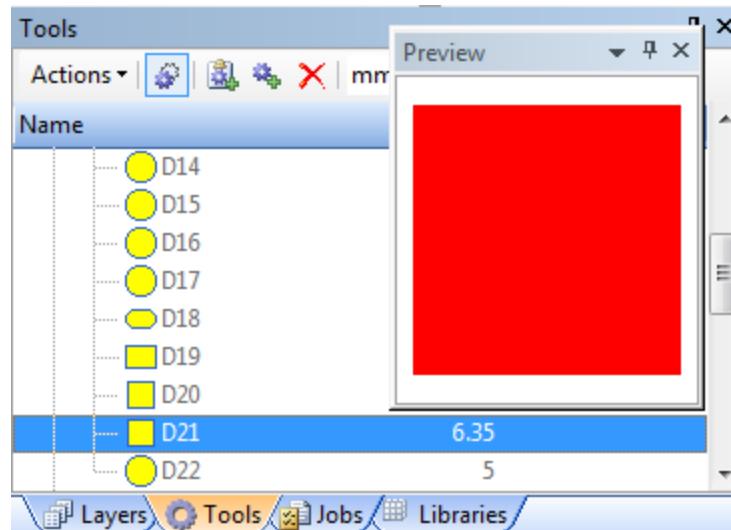


Figure 132. Preview of Tool

This **Preview** window can be used to explore libraries content as well. The **Preview** window will display graphical content of a library, if the library item in **Library** view is focused.

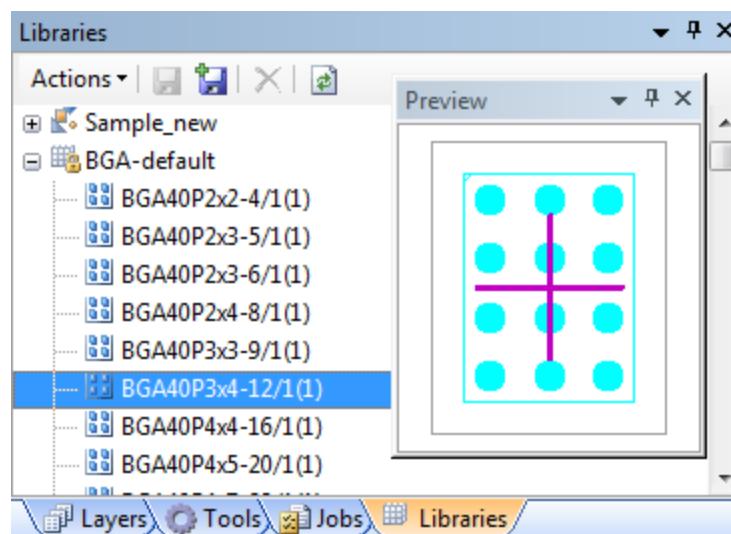


Figure 133. Preview of Library content (BGA package)

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.1.10. Handling of Esc Key

The **Esc** key can be used for cancelling or interrupting operations executed in *CircuitCAM*. Additionally, the **Esc** key is used to bring the focus back to the **Graphic** view. Depending on what is currently selected, escape works in a multistage process; so several **Esc** key presses might be needed. In case that a graphical object is currently created or edited, alternatively the right mouse button can be used to end this operation.

Example 1

The focus is in the **Layers** view. There are more than one selected layer in the **Layers** view and a graphic object (i.e.: path) is selected in the **Graphic** view.

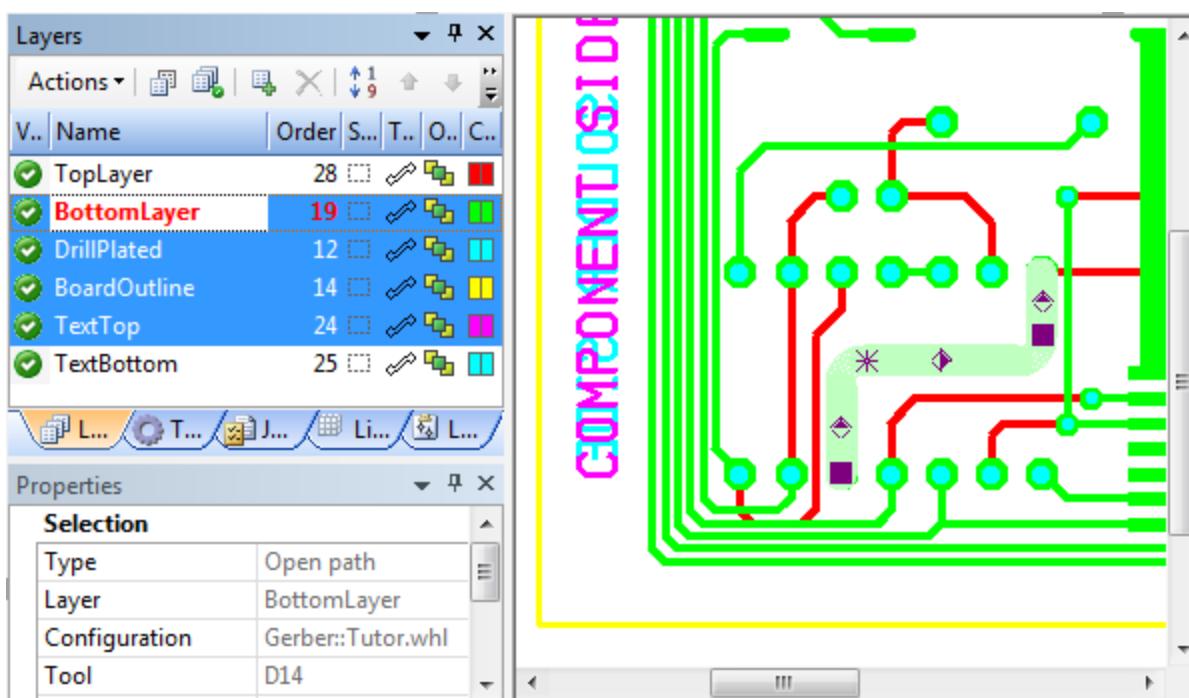


Figure 134. Selected layers in **Layers** view, highlighted path in **Graphic** view

1st Esc: All layers are deselected in the **Layers** view, except the focused one (i.e. "*BottomLayer*").

2nd Esc: Focus switches from the **Layers** view to the **Graphic** view.

3rd Esc: The path is unselected in the **Graphic** view.

Example 2

The focus is in the **Layers** view. There is one selected layer in the **Layers** view and a graphic object (i.e.: path) is selected in the **Graphic** view.

1st Esc: Focus switches from the **Layers** view to the **Graphic** view.

2nd Esc: The path is unselected in the **Graphic** view.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.2. CAM and CAT Document Handling

CircuitCAM documents are stored in files that are based on the [CircuitCAM format](#) with the file-name extension "***.cam**"; we also call them CAM documents. The same file format is also used for:

- Template files with the file-name extension "***.cat**" and stored in the "*New_Templates*" installation directory.
- Library files with a file-name extension in the form "***.cat-xxx**" where "xxx" indicates the type of library, *e.g.:* "***.cat-pkg**" for the package library. These files are stored in the "*Libraries*" installation directory.

When starting *CircuitCAM* without any file specification, or when using the **New** submenu, a new **CAM** document is created with the name "**Untitled N**", where N is the sequenced number of the document (1, 2, 3 ...).

The document name is always displayed in the title bar. If an asterisk "*" is displayed in the title bar after the name of the document, this means that there are unsaved changes in the current document/project.

CAM and CAT documents can be handled through various operations:

- [New](#),
- [Open](#),
- [Save](#),
- [Print](#),
- [Synchronize](#),
- [Libraries](#).



Note: It is recommended to frequently save your modified documents, especially after major modifications or before taking a break from work.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.2.1. Open

Opens an existing *CircuitCAM* document/project/footprint/package or library.

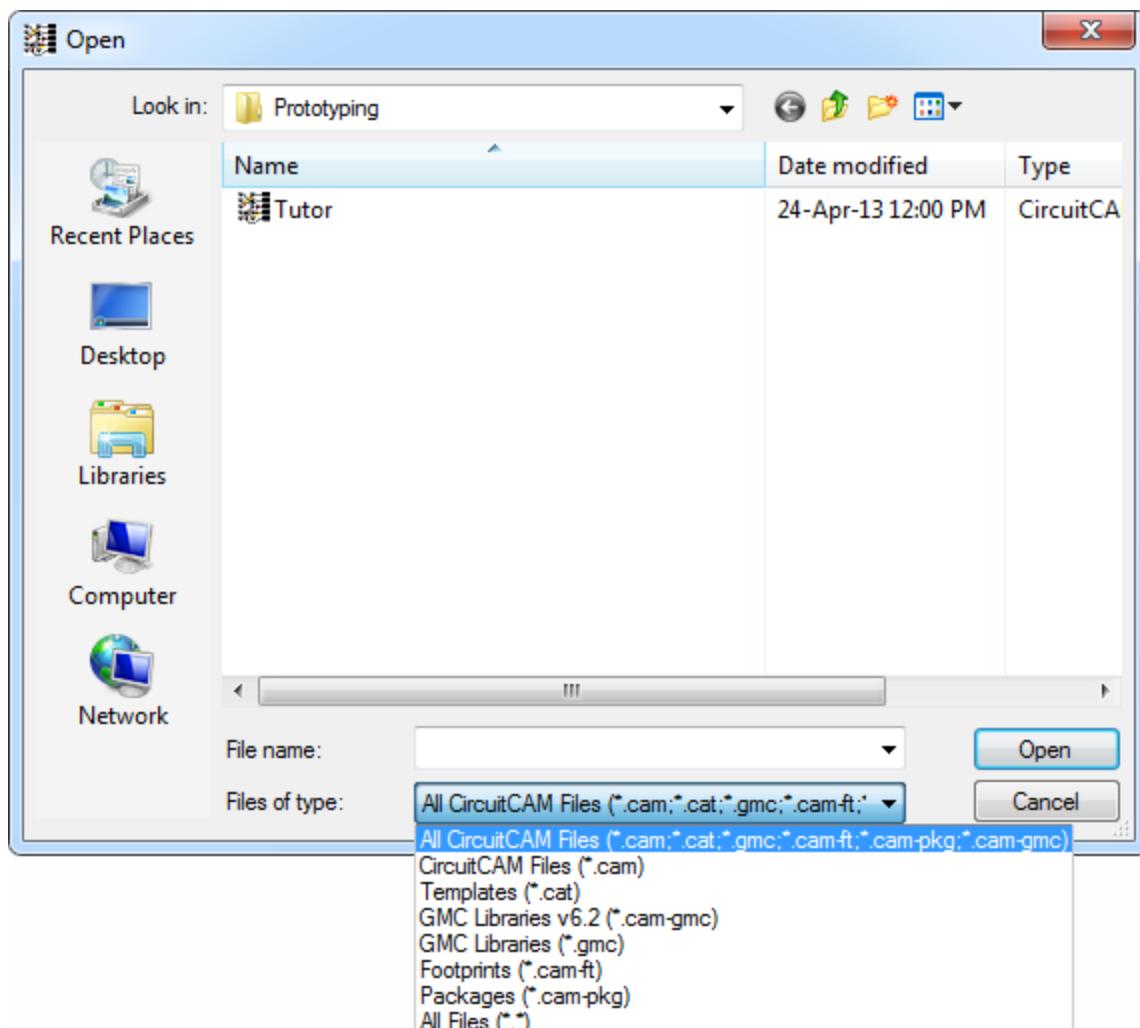
You can open your documents in one of the following ways;

- By selecting **Open** from the **File** menu.
- Pressing the keyboard shortcut **Ctrl + O**.
- Clicking the open icon in your standard toolbar.
or
- By double clicking an existing **CAM** file through your file browser.

After doing any of the first three steps, a browsing window will appear allowing you to browse in your computer choosing the file to open.

The "*Files of type*" filter controls which file formats to be shown; **CAM files, templates, GMC libraries, footprints, packages, or all files *.***. By default, the filter is set to show all files that are compatible with *CircuitCAM* file formats.

Figure 135. Open file dialogue



Note: You can see a list of your recently opened files and documents in the **Recent Files** submenu under the **File** menu.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.2.2. Synchronize

With synchronize, the currently displayed CAM document is modified in such a way that it becomes compatible with the selected CAT (template) file. This is done by copying layers, tools and jobs from the selected template file to the currently active document.

In addition the selected CAT file is stored as the Underlying CAT File of this document.

The CAM document can either be synchronized with one of the default template files, or it can be synchronized with a user created template.



Note: To add new template file(s) to *CircuitCAM*, please follow the following procedures:

- Using the file browser navigate to the place of the new template file.
- Make a copy of the new template file and paste it in the **New_Templates** in the main *CircuitCAM* installation directory, this folder contains all the template files that *CircuitCAM* uses.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.2.3. New

CircuitCAM offers two ways of creating a new document:

1. Use the document based on the default template file "**default_insulate.cat**" which is selected automatically in *CircuitCAM* when the program starts.
2. Create a new document by selecting a pre-configured template file from the **New** submenu under the **File** menu.

If there are no ***.CAT** files an internal empty template is used.



Note: New documents are always named "**Untitled**" followed by sequenced numbers for multiple documents.

You can create new documents in one of the following ways;

- by selecting **New** from the **File** menu.
- pressing the keyboard shortcut **Ctrl + N**.
- clicking the open icon in your standard toolbar.



Note: Pressing the keyboard shortcut **Ctrl + N** or clicking in the toolbar, will start a new document using the default template file.

New Templates

CircuitCAM allows users to create a template file with user-specific settings, which are comparable to "**format templates**" files in a word processor.

Template files allow the users to save time and implement standards more effectively. Commonly-used lists may be saved as a template. For example, if the same aperture list and tool list are used frequently, the user can create a template file that contains both lists. All current settings, such as names and colors of the layers, are also saved in this template file. This template may be used as the basis for other circuit board designs. Thus, there is no longer a need to import these lists again and again.

CircuitCAM template files have the same build and internal format as **CAM**, but they have the extension "**CAT**" so that they can be distinguished from each other.

To create a new template file;

- Create and/or select the wanted jobs, scripts, tools, layers, colors....etc
- Under the **File** menu, select **Save As**, and from the filter menu, select **Template (*.cat)**, the desired template name is entered.



Note: The saved templates can be synchronized into any other *CAM* document, see [Synchronize](#).

For more information see [CAT](#).

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.2.4. Import

One of the *CircuitCAM* functionalities is the importing of production data (apertures, circuit board or drilling data). To perform this, select **Import** from the **File** menu. The **Select files for import** dialogue window is displayed.

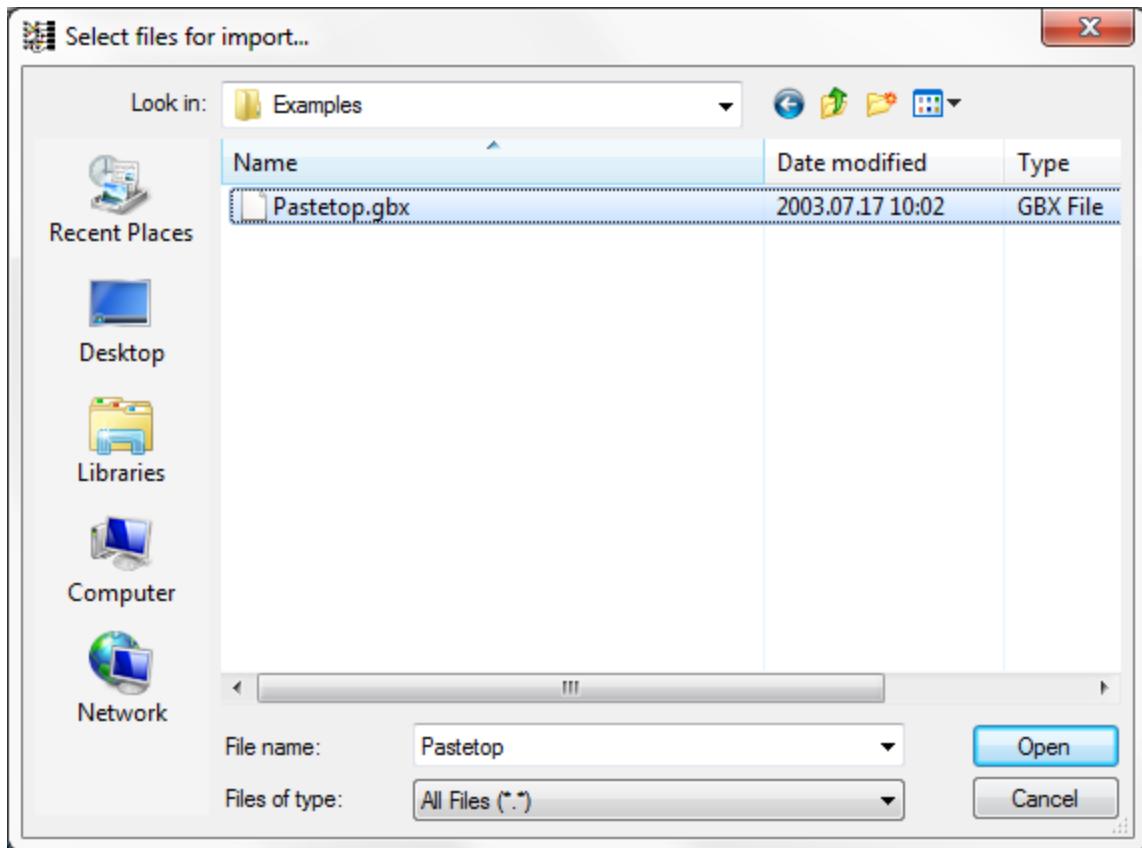


Figure 136. Select files for import... dialogue window

After the selection is made (one or multiple files) and the **Open** button is clicked, the following **Import** dialogue window is displayed.

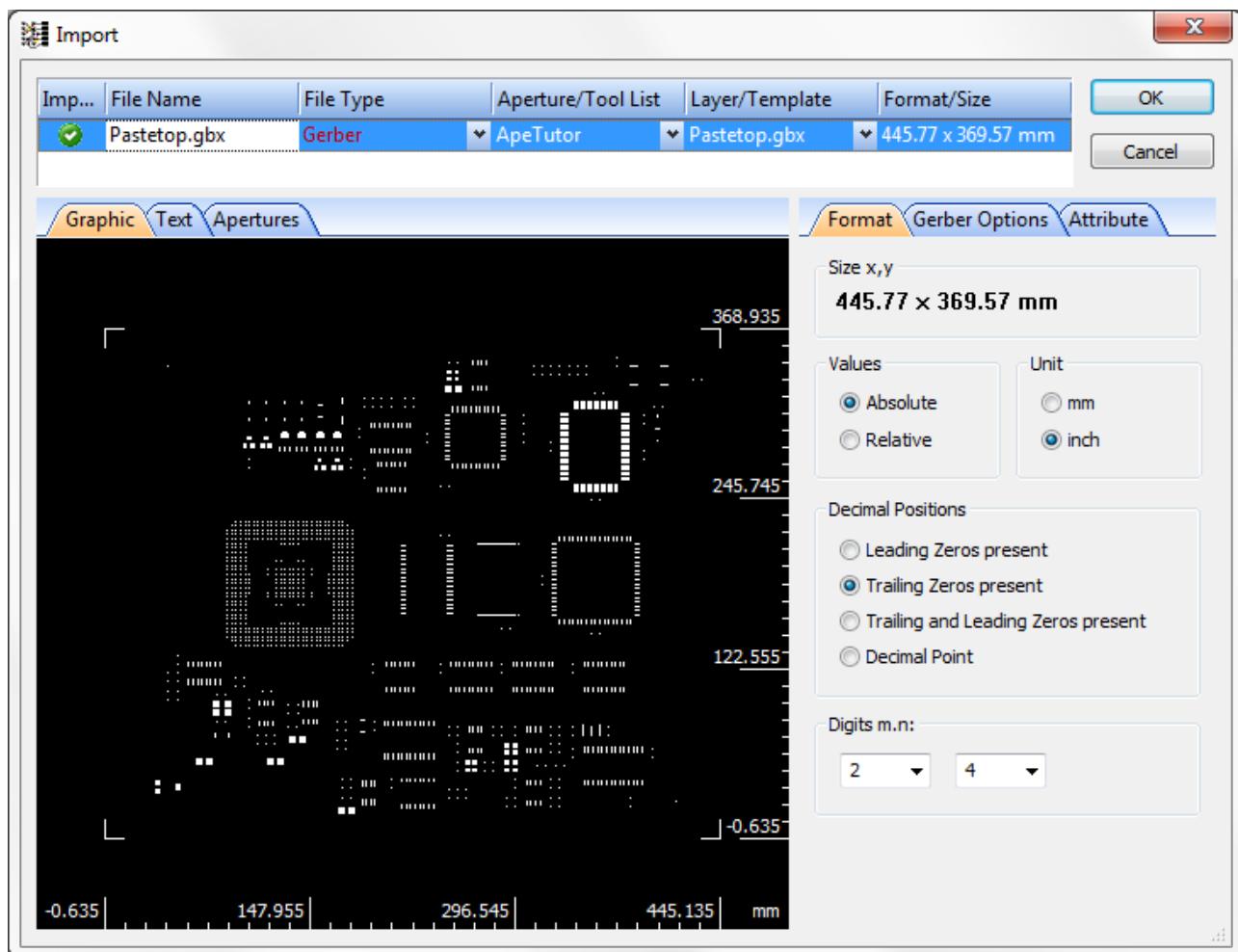


Figure 137. Import dialogue window

- **Import:** if is selected, import is enabled; if is selected, import is disabled (the import state can be changed by clicking on the icon).
- **File Name:** indicates the name of the file to be imported.
- **File-Type:** specifies the type of the file. The file is recognized by *CircuitCAM* and its type is set automatically. In case the file was not recognized, or the user wants to change its file type, the selection of the file type can be done manually from the type list. *Aperture/Tool auto* is shown if an aperture or tool list was recognized and just one matching template was found. If more than one matching template was found - *Aperture/Tool select* appears in the cell.



Note: In case of *IEC-1182/IPC-35x* file type, the file is automatically imported without showing the import dialogue window. The *IecDefault* configuration is used.

- **Aperture/Tool List:** if the file was recognized as an aperture or tool list, **Aperture/Tool List** shows the name of the list to store the apertures or tools in *CircuitCAM*. By default a file name is used as an aperture or tool list name.
If the file was recognized as a data format, *Aperture/Tool List* shows a name of the list to be used to import the data file.
- **Layer/Template:** shows the first matching template, if the file was recognized as an aperture or tool list. Another template can be selected from the list box, if the first one does not fully match. Templates are sorted by decreasing number of recognized apertures.

If the file was recognized as a data format, **Layer/Template** shows the layer name to store the data in *CircuitCAM*. The layer can be selected from the list box, according to the content of the file. By default the file name is used as the layer name.

- **Format/Size:** if the file was recognized as a list of an aperture or a tool list, it shows the format settings. If the file was recognized as a data format, max dimensions are shown.

Graphic Panel

Shows the resulting graphical view, which depends on the currently selected parameters.

Text Panel

Presents the textual file contents.

Apertures Panel

Provides a list of apertures (tools) of the configuration, which was selected in **Aperture/Tool List** list box.

Name	M.	a	b	c
D10	*	0.1 mm		
D11	*	1 mm		
D12	*	0.3 mm		
D13	*	1.1 mm		
D14	*	1.5 mm		
D15	*	1.3 mm		
D16	*	1.6 mm		
D17	*	0.05 mm		
D18	**	2.5 mm	0.6 mm	
D19	**	2.5 mm	0.6 mm	
D20	*	6.35 mm		
D21	*	6.35 mm		
D22	**	5 mm		
D113	**			

Figure 138. Apertures panel

- **Name:** specifies the shape and the name of an aperture.
- **Mode:** specifies the using mode of an aperture.
- **a, b, c, ..., j:** these fields describe dimensions of the shape. E.g.: **a** value means diameter of a circle or width of a rectangle/oval; **b** – height of a rectangle; **c** – a bevel or a radius for a bevel/rounded rectangle.

For more detailed description, please refer [Tools](#) section.

Format Panel

The **Format** panel contains the following information:

- **Size X,Y:** indicates the size (bounding box) of the imported board.
- **Values:** coordinates can be in an absolute or relative (incremental) form.

- **Unit:** shows a unit (mm or inch), which is used to represent coordinates of all graphical objects, drawn by using tools in the current configuration.
- **Decimal position group:** since the decimal point for coordinate values is usually not given in the *Gerber/Excellon/Sieb&Meyer* format. The position of the decimal point has to be determined by other methods. For this purpose, the **m.n** digits, and the selection from the decimal group are combined.



Note: For different *File-Types* this group has different fields. For *NC_Drill* an additional **Decimal Point** field is provided, *HP-GL* and *Barco* types have no **Digits m.n** and **Decimal position** groups at all. *DXF* type has a new **DXF –Import: Field Circles** flag to fill an imported circle (otherwise the circle is assumed as a circle in Closed-path, i.e.: a contour with zero path Width).

- **Digits m.n:** these values are used if no decimal point format is set. "m" indicates the number of digits before the decimal point and "n" the number of digits after the decimal point.

Gerber Options Panel

The **Gerber Options** panel gives possibility to set some input options for *Gerber* and *GerberX* formats (for other format types the button is invisible):

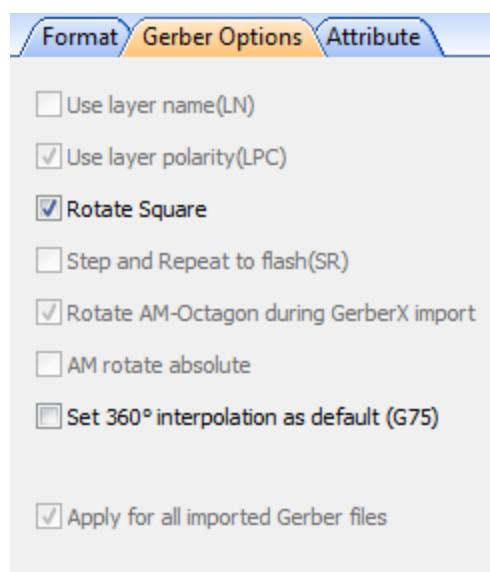


Figure 139. Gerber Options panel

1. **Use layer name (LN):** if *On*, the names of the layer from LN commands will be used always (despite that it was used only once);
2. **Use layer polarity (LPC):** a possibility to ignore LPC/LPD commands.
3. **Rotate square:** if *On* = aperture is rotated during drawing; if *Off* = a path is drawn as a polygon and its width depends on a rotation angle - the aperture always remains in a zero rotation.
4. **Step and Repeat to flash (SR):** if *On*, the repeated group is Special flash; if *Off* - is similar to the denested flash.
5. **Rotate AM-Octagon during GERBER import:** if *On*, an octagon is rotated by 22.5 degrees to compare with others *Aperture Macro* (AM) primitives of type **Polygon**. It means that AM octagon is converted into *Aperture Definition* (AD) regular octagon.
6. **AM rotate absolute:** if *Off*, *CircuitCAM* rotates every AM primitive around the point 0/0. If *On*, *CircuitCAM* rotates every AM primitive around its reference point

according to this table (primitive Reference point): 2/20 - start point, 21 - centre point, 22 - lower left point, 4 - start point, 5,6 and 7 - centre point.

7. Set 360 interpolation as default (G75): with *Off* the quadrant circular interpolation is used.



Note: Default values after *CircuitCAM* installation are: Use layer polarity (LPC) = On, Rotate square = On and Rotate AM-Octagon during GERBER import = On. Others flags are in Off state.

The **Apply for all imported Gerber files** option enables a user to set the selected options to all imported Gerber files.

Attributes Panel

The **Attribute** panel shows a shape and a size of an aperture/tool:

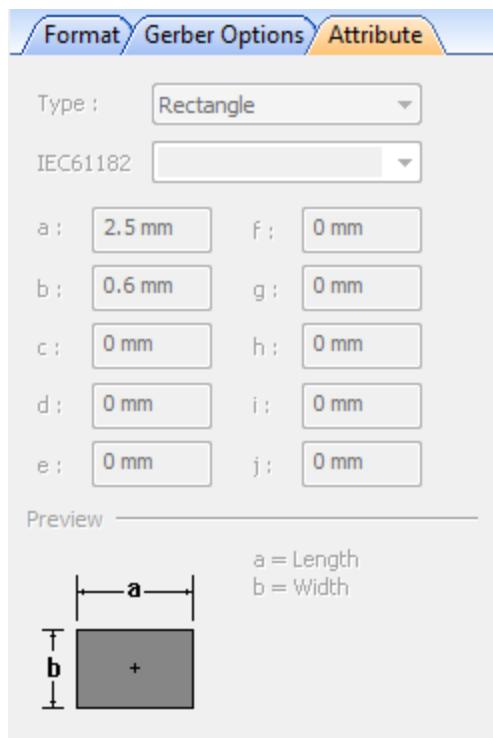


Figure 140. Attribute panel

- **Type:** specifies a shape of an aperture/tool.
- **IEC61182:** specifies a shape from the special shape library.
- **a, b, c, ..., j:** specifies dimension values of the aperture/tool.
- **Preview:** shows a graphical view of the selected type and explains the meaning of fields (*a/b/c/.../i/j*) for this type.
- **OK** button runs an import operation. It is disabled if at least one obligatory assignment is not done.

This dialogue window has one interesting feature – it allows direct editing of any aperture while importing an aperture/tool list, and UNDEFINED (with ? icon in **Apertures** panel) aperture while importing data file, in the **Attribute** panel.

3.2.5. Saving your data.

CircuitCAM allows the user to save documents and projects in binary formatted files. These files will store all the CAM data inside them.

Save

Saves the current document using the existing file name. In case of a new "*UntitledN*" document, when pressing **Save** the **Save As** menu is activated instead.

Save As

This function, in contrast to **Save**, allows the user to define a file name, type and directory. The standard Windows **Save As** dialogue window appears, in which the user can choose the appropriate data for the saved file. The data can be saved in all formats supported by CircuitCAM; *CAM files, templates, GMC libraries, footprints, packages, or all files* *.*.

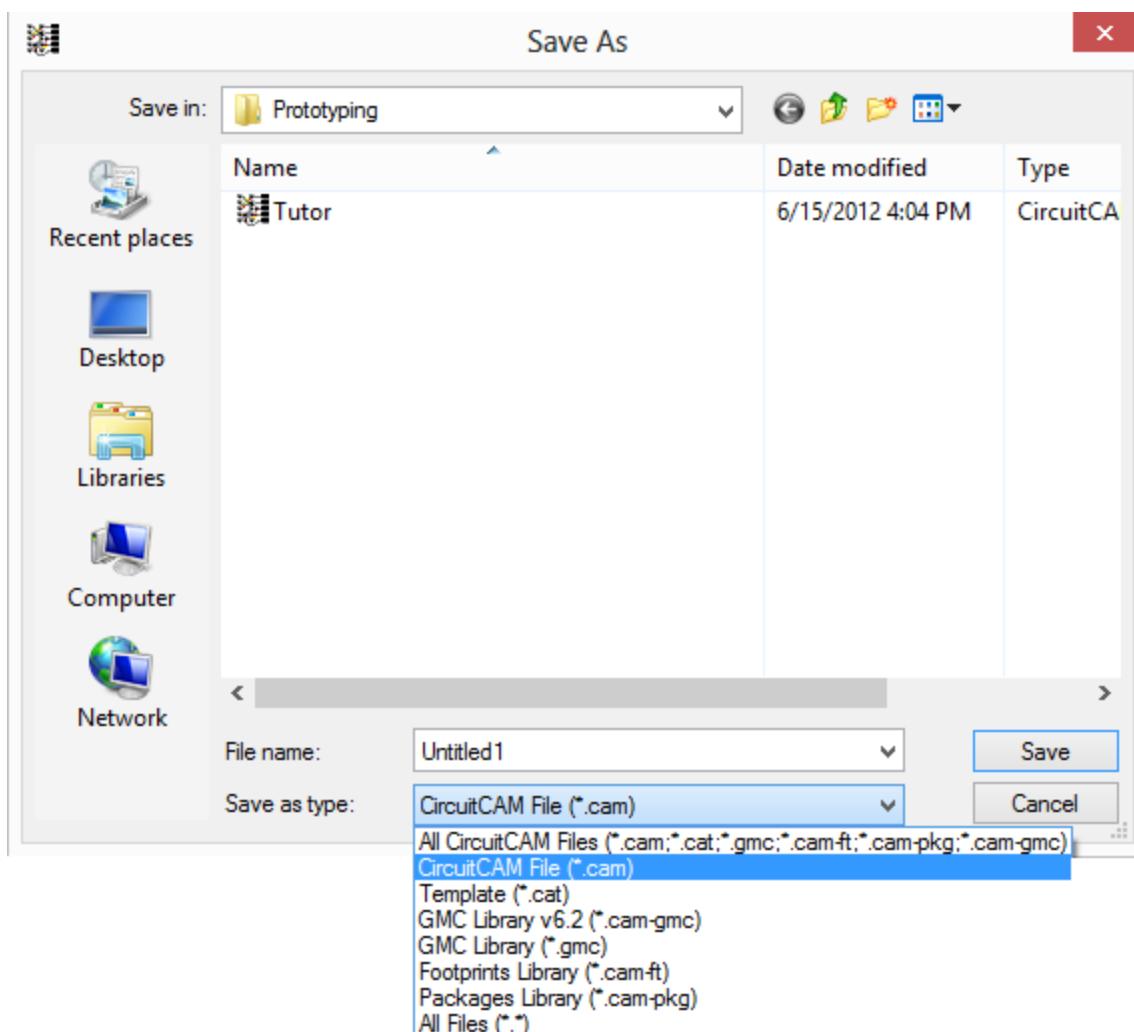


Figure 141. Save As dialogue window



Note: If an asterisk "*" is displayed in the title bar after the name of the document, this means that there are unsaved changes in the current document/project.

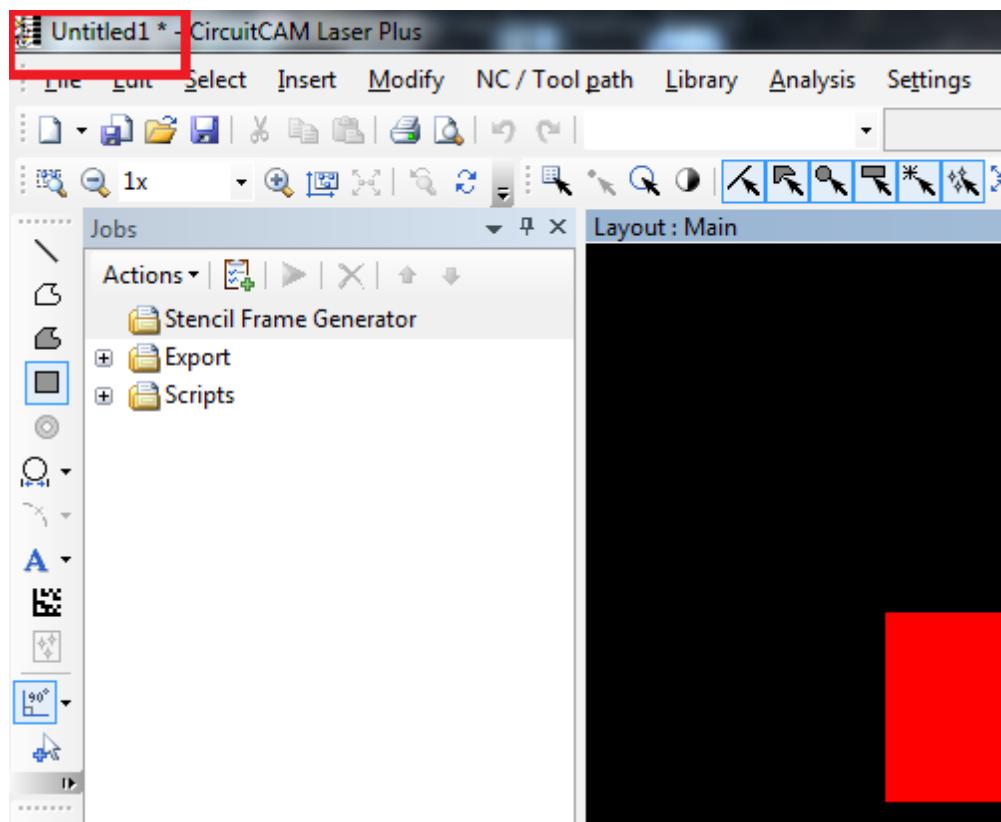


Figure 142. If the CAM document contains any unsaved changes, an asterisk "*" appears next to the document name



Note: For .CAM document properties see [Properties](#).

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.2.6. Export

Allows the user to export production data into files in the common *CAM* formats *Gerber*, *Excelon*, *HP-GL* and others.

Export is organized by **Export Jobs** that are a collection of **Export Tasks**. Export jobs have a name and are stored in the local *CAM* file, so that they can be re-executed at any time. Default export jobs are provided in the *CAT* files. An Export Task specifies a certain layer that to be exported by using a specific configuration/tool list. User can create or modify Export jobs or export tasks.

To see the available export jobs in the local *CAM* file, select **Export** from the **File** menu and select an export job from one of these groups:

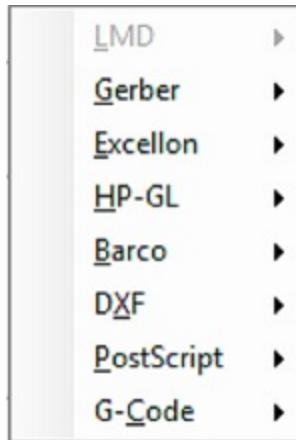


Figure 143. Export menu content

When executing an Export Job, a statistic overview of the used config-items/tools and further information is recorded in the [Message Log](#).

When exporting in the basic Gerber format, an aperture file is needed in addition to the Gerber file because the Gerber file is not able to contain the size and shape of the used apertures. An aperture file is generated in the same directory of the Gerber file and the same file name with an additional extension ".rep", See [Example of Aperture List after Gerber File Export](#).



Note: The **VIEWER** variant of *CircuitCAM* does not allow any export jobs.

Further details on Export:

- [Export Job](#)
- [Export Task](#)
- [Export Layout Instance](#)
- [Example of Aperture List after Gerber File Export](#)

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.2.7. Export Layout Instance

With export layouts instance a customizable ASCII file is created, which lists all layout instances of the current layout; typically the **main** layout of the current document. The generated ASCII file is organized in such a way that it can be imported into typical spreadsheet applications, such as Microsoft Office, Open Office or Google Doc. The ASCII file consists of a header, that contains CAM file, user, organization, time and licence information. This is followed by one line for each layout instance, consisting of the name of the layout instance, the name of the layout, its category and group, the coordinates and rotations.



Note: This capability is available with *CircuitCAM PRO* only.
The **Layout instances** export job has additional **Settings** button to specify/modify export format:

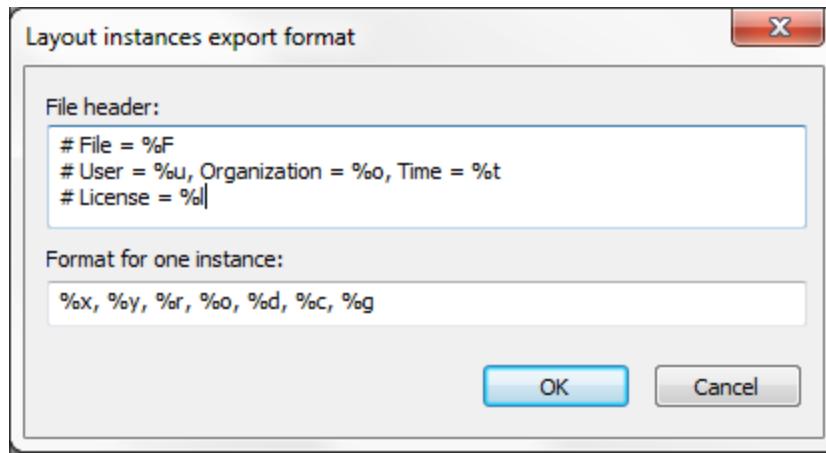


Figure 144. Layout instances export format dialogue window

The configurable format for each **layout occurrence**:

%x - Value of the x coordinate of the layout occurrence,

%y - Value of the y coordinate of the layout occurrence,

%r - Rotation in degrees,

%o - Name of the layout occurrence,

%d - Name of the layout definition,

%c - Name of the category of the layout definition,

%g - Name of the lowest level group that contains the layout definition.

The **default** configuration string for layout occurrences is:

"**%x, %y, %r, %o, %d, %c, %g**"

In addition, a user may specify a multi-header format **for a file**:

%f - Name of the CAM-file,

%F - Name of the CAM-file including full path,

%u - Name of *CircuitCAM* user,

%o - Name of *CircuitCAM* organization,

%l - *CircuitCAM* licence id,

%t - Timestamp in YYYY-MM-DD HH:MM.SS ISO standard (ISO 8601).

The **default** configuration for the header is:

"**# File = %F**

User = "%u", Organization = "%o", Time = %d

Licence = %l"

The export file example with default form settings:

Instances file generated by CircuitCAM 6.3.0.1304

File = D:\Footprint recognition\Sample1.cam

User = RA, Organization = KTU, Time = 2011-01-26 12:07:09

Licence = 11223344

179.55, 128.299, 0.00, I1, BGA80P14x14-179/1(1).1, SolderPasteTop, BGA

222.271, 167.545, 0.00, I2, BGA80P14x14-179/1(1).2, SolderPasteTop, BGA

226.507, 115.887, 90.00, I3, BGA80P14x14-179/1(1).3, SolderPasteTop, BGA
133.555, 133.1, 0.00, I11, BGA127P1x5-5/1(1).2, SolderPasteTop, LPCC
140.246, 112.684, 0.00, I12, BGA127P1x5-5/1(1).3, SolderPasteTop, LPCC
140.255, 133.1, 180.00, I13, BGA127P1x5-5/1(1).4, SolderPasteTop, LPCC

Detected format errors are listed in the Message Log window. For example:
ERROR in Export instances - Wrong (not supported) format %q.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.2.8. Example of Aperture List after Gerber File Export

When exporting in the simple Gerber file format, an additional aperture file is needed that contains the size and shapes of the used aperture, because this information is not included in the Gerber file itself, below you can see an example of such a file.



Note: For other file format such as GerberX and Excellon the size and shape of the aperture/tool is provided in the files and so no extra files is needed.

Example: aperture file "**Tutor.gbr.rep**", generated together with Gerber file "**Tutor.gbr**" In this case the export job had only one export task:

```
# APERTURE LIST
# for the GERBER file C:\Program Files (x86)\CircuitCAM7\ExampleData\Prototyping\Tutor.gbr
# Generated by CircuitCAM 7.1.2.2106
```

Export task : BottomLayer

Aperture unit : inch

```
Mode : Absolute
Unit : inch
Format : 2.4
Zeroes : Trailing Zeroes present
# Shape D-code Width Height Usage
Circle 12 0.011811 Both
Circle 11 0.039370 Both
Circle 13 0.043307 Both
Circle 15 0.051181 Both
Circle 14 0.059055 Both
Circle 16 0.062992 Both
Rectangle 19 0.098425 0.023622 Flash
```

Export task : TopLayer

Aperture unit : inch

Mode : Absolute

Unit : inch

Format : 2.4

Zeroes : Trailing Zeroes present

Shape D-code Width Height Usage

Circle 12 0.011811 Both

Circle 13 0.043307 Both

Circle 15 0.051181 Both

Circle 14 0.059055 Both

Circle 16 0.062992 Both

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.2.9. Print

CircuitCAM has printing related functions in the **File** menu, such as **Print Setup**, **Print** and **Print Preview**. The **Print Setup** and **Print** functions are standard Windows functions and their dialogue-boxes are described in Windows documentation. To display a document (either graphic or list window) as it should look when printed, select **Print Preview** function:

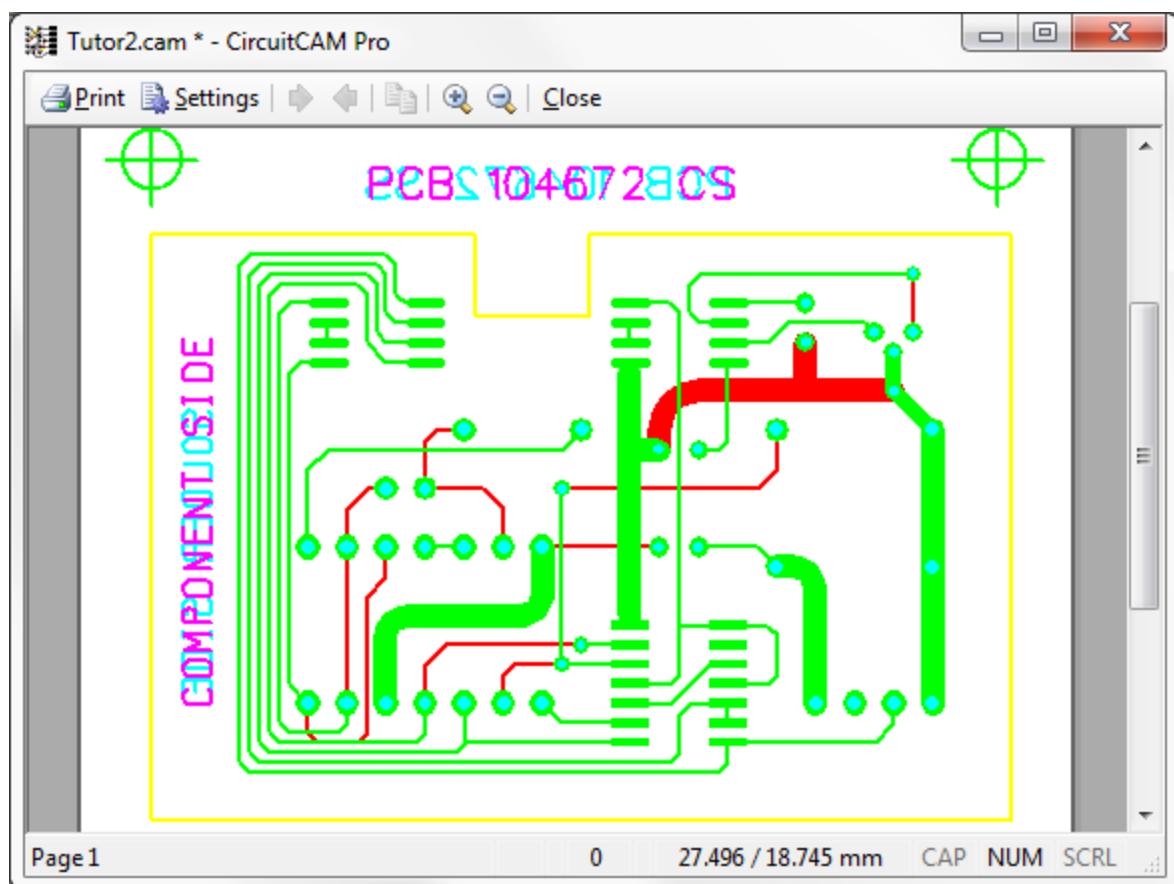


Figure 145. CircuitCAM Graphic view

Print Preview has a menu with following functions:

- **Print:** prints the document.
- **Next/Previous/Two pages:** page selection modes.
- **Zoom In/Zoom Out:** a picture is shown enlarged/reduced by a factor of 2 around its centre. To **Zoom In** around the specified point, use the special marker (magnifying glass).
- **Settings:** provides a possibility to change the size, colors and the view mode of the displayed picture:

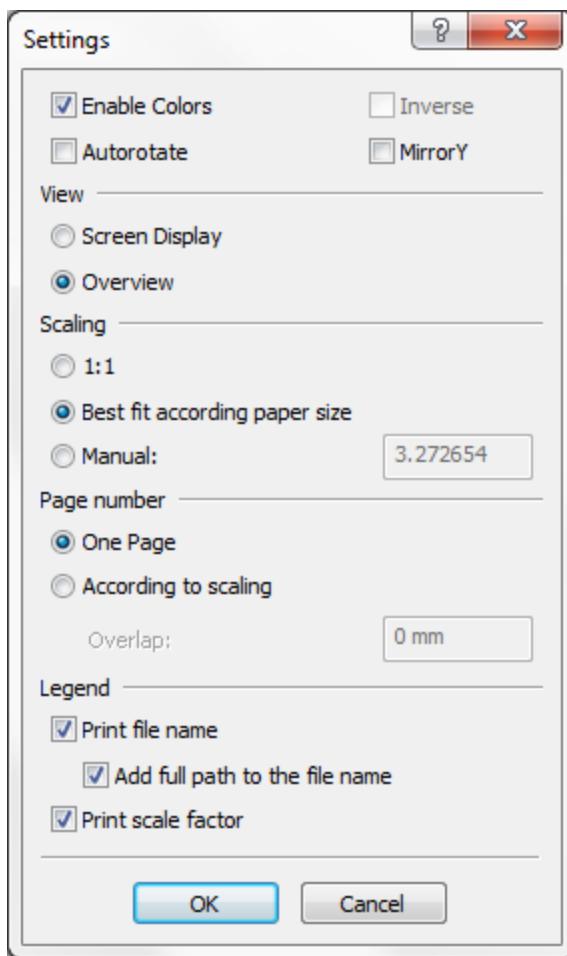


Figure 146. Settings dialogue window

- **Enable Colors:** if *On*, a picture keeps its original colors on the white background. Otherwise only the black color for the foreground is used.
- **Inverse:** if *On*, the black color is used for the background and the white color for the foreground. This option can be set *On* after the **Enable Colors** option is set *Off*.
- **Autorotate:** if *On*, a *Portrait* or a *Landscape* orientation is automatically selected to refuse a free space.
- **Mirror Y:** if *On*, transforms a graphical image into vertically reversed form.
- **Overview:** all objects are displayed fitting the screen.
- **Screen Display:** only objects on the screen are displayed.
- **1:1:** if *On*, the graphical object is displayed in its original size.
- **Best fit according paper size:** if *On*, the graphical object is resized to fit the window.
- **Manual:** user can scale the displayed picture by a new scale factor.
- **One Page:** always displays only one page (despite the *Scale* value).
- **According to scaling:** if the *Scale* value does not allow fitting a picture in one page, the picture is split into the necessary amount of pieces-pages (see picture below). To control an overlap of the picture, enter a value into the **Overlap** box. The default value is 0 mm.
- **Print file name:** if *On*, the current file name is displayed on the top left corner of the page.

- **Add full path to the file name:** if ***On***, the full path of the current file is displayed on the top left corner of the page. The file name is added only if the **Print file name** is ***On***.
- **Print scale factor:** if ***On***, the current scale factor is displayed on the bottom left corner of the page.
- Use the **Next** and **Previous** buttons to walk throughout multiple pages.

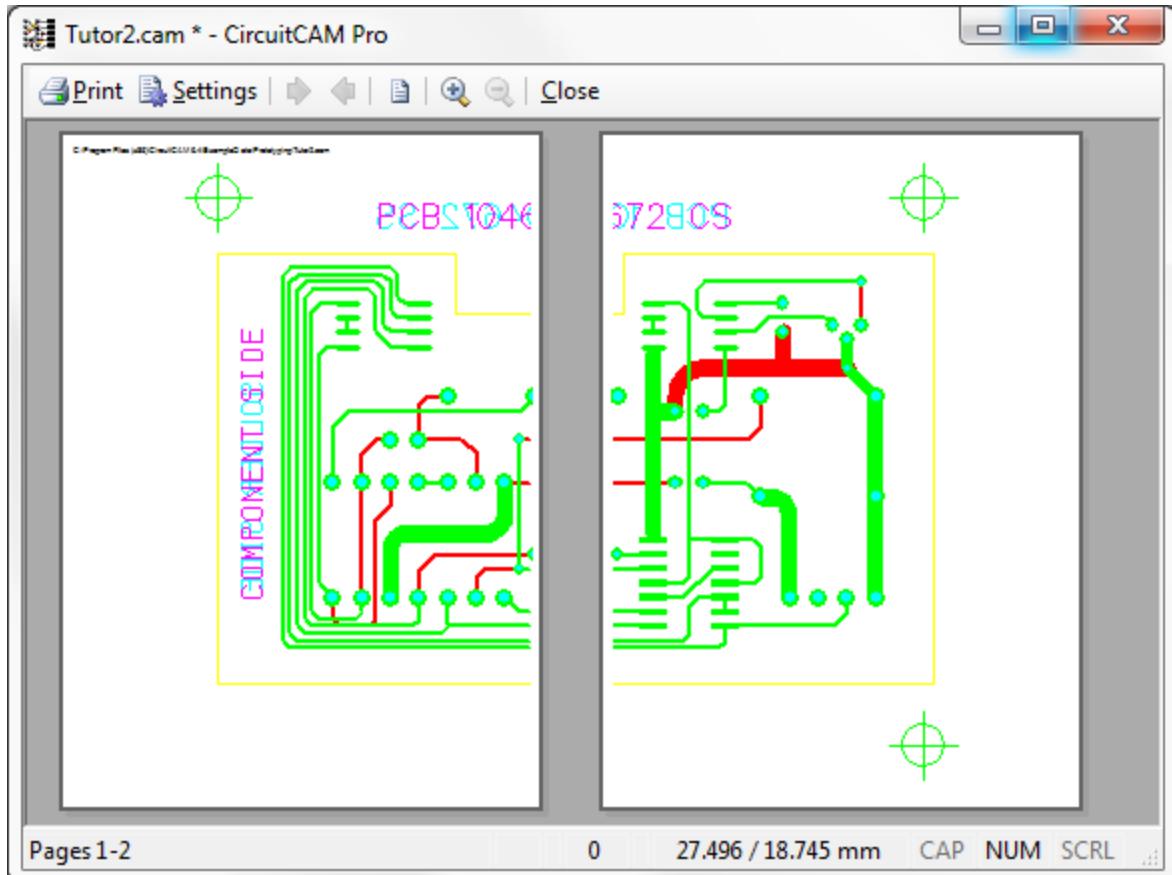


Figure 147. Print preview of graphic on multiple pages

- By using the **Next** and **Previous** buttons a user can walk throughout multiple pages.

Printer Calibration

The printer calibration allows you to correct the aspect ratio of a printer in X and Y direction to achieve higher accuracy. Due to low accuracy of common laser printers, objects printed on an A4 sheet can appear up to 2 mm bigger or smaller as defined in data, the size might be different in width and height direction. To ensure a 1:1 print, *CircuitCAM* can compensate these tolerances.

Printer Calibration is a multi-step process:

1. A big rectangle is printed with correction factors **1** in both **X** and **Y** directions.
2. The printed rectangle is measured with a ruler.
3. The measured values are entered in the **Printer Calibration** dialogue and **Correction factors** will be calculated automatically from them.
4. For verification the big rectangle is printed again and measured. It should now have the specified size.

5. Now the normal printing function can be used. It will use the specified correction factors.

The dialogue window before the calibration:

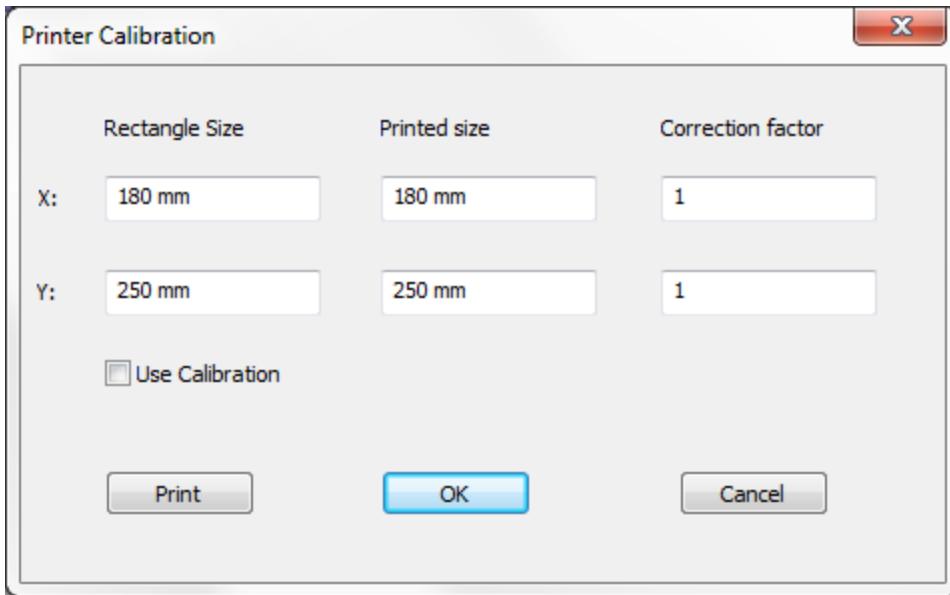


Figure 148. Printer Calibration dialogue window

Rectangle size X

Defines the X dimension of the rectangle to be printed for the calibration. This value will be used, when the option **Use calibration** is set **Off** or the **X Correction factor** is set to 1.

In case the option **Use calibration** is set **On** and the **X Correction factor** is not 1, *CircuitCAM* will use the corrected size while printing the rectangle.

Rectangle size Y

Defines the Y dimension of the rectangle to be printed for the calibration. This value will be used when the option **Use calibration** is set **Off** or the **Y Correction factor** is set to 1.

In case the option **Use calibration** is set **On** and the **Y Correction factor** is not 1, *CircuitCAM* will use the corrected size while printing the rectangle.

Printed size X

Shows the X dimension of the printed rectangle without the calibration.

Printed size Y

Shows the Y dimension of the printed rectangle without the calibration.

Correction factor X

The factor will be used to scale images to be printed in X direction.

The value is updated when a new **Rectangle size X** or the **Printed size X** is entered. On the other hand, when this value is changed, the **Printed size X** is updated.

Correction factor Y

The factor will be used to scale images to be printed in Y direction.

The value is updated when a new **Rectangle size Y** or **Printed size Y** is entered. On the other hand, when this value is changed, the **Printed size Y** is updated.

Use calibration

Defines whether the correction factors will be used during printing. Correction factors will be used to print the calibration rectangle as well.

Print

Prints the calibration rectangle. If option **Use calibration** is **On**, the rectangle will be scaled by the correction factors.

The calibration rectangle will be printed in an outline mode in a black color with a zero line width in the centre of the page. In addition, a text will be printed on the top of the page:

- If the calibration is used: “Calibrated 250 x 180 mm”.
- If the calibration is not used: “Not calibrated 250 x 180 mm”.

The numbers depend on the **Rectangle size** values, and the unit depends on the **Unit** setting in the **General Settings**.

OK

Closes the dialogue window and saves the values in the registry.

Cancel

Closes the dialogue window and ignores the changes.

How to process the calibration

1. Switch **Off** the **Use Calibration** check box.
2. Enter **Rectangle Size** values to get a rectangle which covers almost the entire page.
3. Click on **Print** button.
4. Measure the printed rectangle and enter the values into the **Printed size** fields.
5. Switch **On** the **Use Calibration** check box.
6. Click **OK** button to close the dialogue box and to save the correction values.

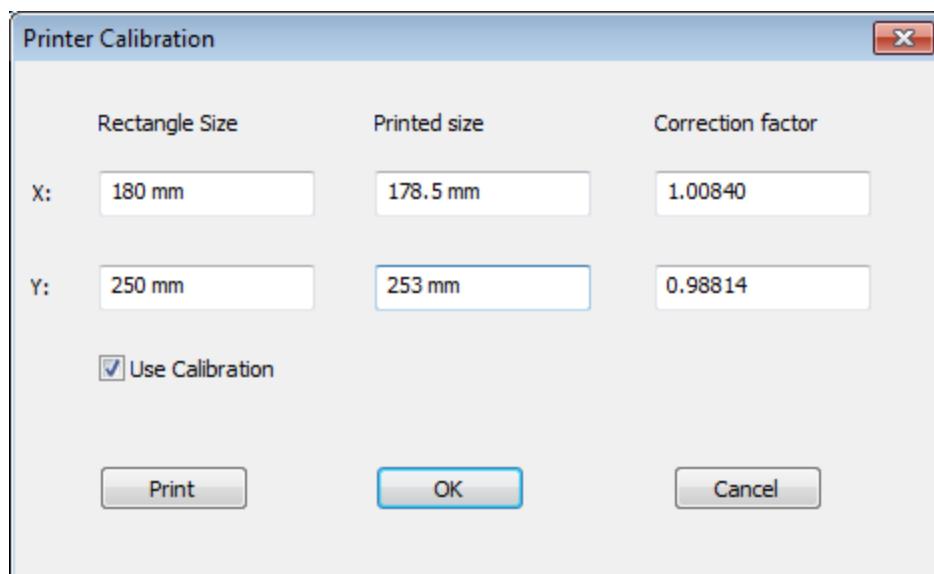


Figure 149. Printer Calibration dialogue window, Use Calibration check box

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.2.10. Read Out

In addition to ["Properties view"](#) which displays properties of a single selected object, and, ["Statistics"](#) which displays properties of all objects in a layout, the following analysis or readout

operations are available for the graphics view;

- Measure.
- Pitch of Two Pads.
- Area of Selected Objects.

Measure

The **Measure** function calculates the distance between the last selected *Anchor Point* and the current cursor position, and displays it simultaneously in Cartesian and Polar coordinates in the *status bar*, to the left of the indication of the selected object (see image).

Distance: -0.363 / 0 mm : 0.363 mm / 180° #46 Open path (4:4 Point) : width 0.3 mm 1 27.8°

Figure 150. Status bar with Cartesian and polar coordinates

In the graphic window the distance is shown with a virtual line, starting from the anchor point and ending with the current mouse position. As long as the *Measure* function is active, each left mouse click restarts the measure calculation.

For taking precise measurements, it is needed that the mouse snaps to specific points on the target object. To achieve this, while the measure function is active, holding down the **shift** button and clicking on an object will display all the marker points of the object. When clicking the mouse button while the shift button is pressed does not restart the measurement. When moving the mouse over a marker point, the cursor will snap to the exact position of this point.



Note: The **Cancel** operation (from context sensitive menu) or **ESC** button interrupts the *Measure* function.

The **Measure** function is accessible by any of the following means:

- Going to the menu; **Analysis---> Measure**.
- Right click context menu; **Measure**.
- Keyboard shortcut; **Ctrl + Shift + X**.

Pitch of Two Pads

This function calculates the distance between the centre points of two selected areas (pads). An area can consist either of one object (flash, path, rectangle, etc.) or of a few connected (overlapped) objects on the same level. Different areas can belong to different layers as well



Note: An area, which consists only of the undefined-flash or zero-radius circle, is excluded, because of its PP polygon is zero. Such selection prohibits the pitch operation (a message comes up).

If a number of selected areas is not 2, a message is shown as well.

The result of the operation is shown on the screen in a message like this:

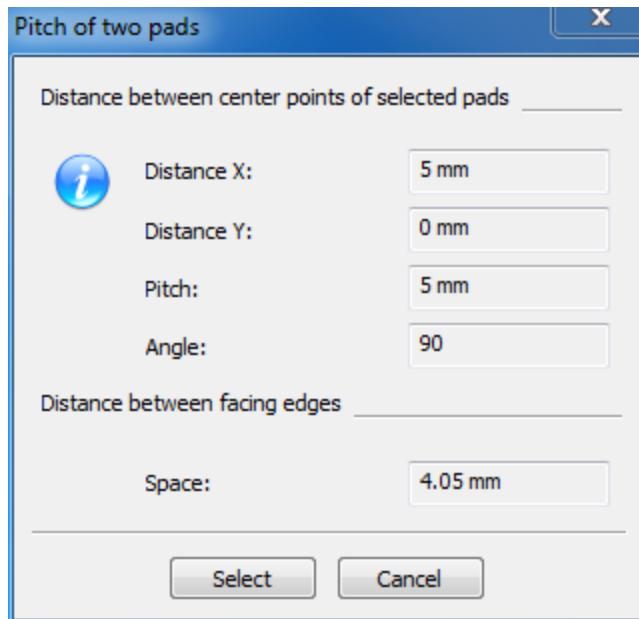


Figure 151. Pitch of two pads

Distance X: The change of the value on X-axis between the two centre points.

Distance Y: The change of the value on Y-axis between the two centre points.

Pitch: Is the distance between the two centre points of the areas.

Angle: Shows the angle between the line going through the centre points of the two selected objects, and the X-axis.

Distance between facing edges: This is the distance between the opposite sides of the two pads.

This dialogue allows the user to see the pitch and the space by just one click.

After pressing on the **Select** button, the **Selection by Pitch** dialogue window comes up (see [Pitch in Select menu](#)).

Area of Selected Objects

Procedure:

- Select an object or a group of objects.
- Select in the main menu **Analysis -> Area of Selected Objects**.

The result of operation is shown in the screen as a message:

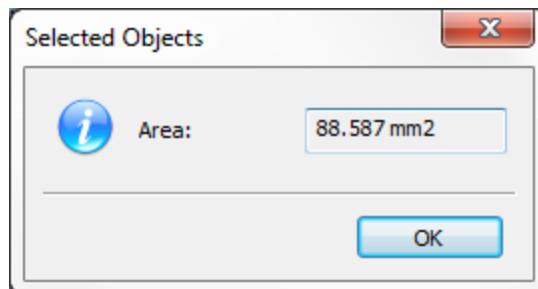


Figure 152. Selected Object message window for area

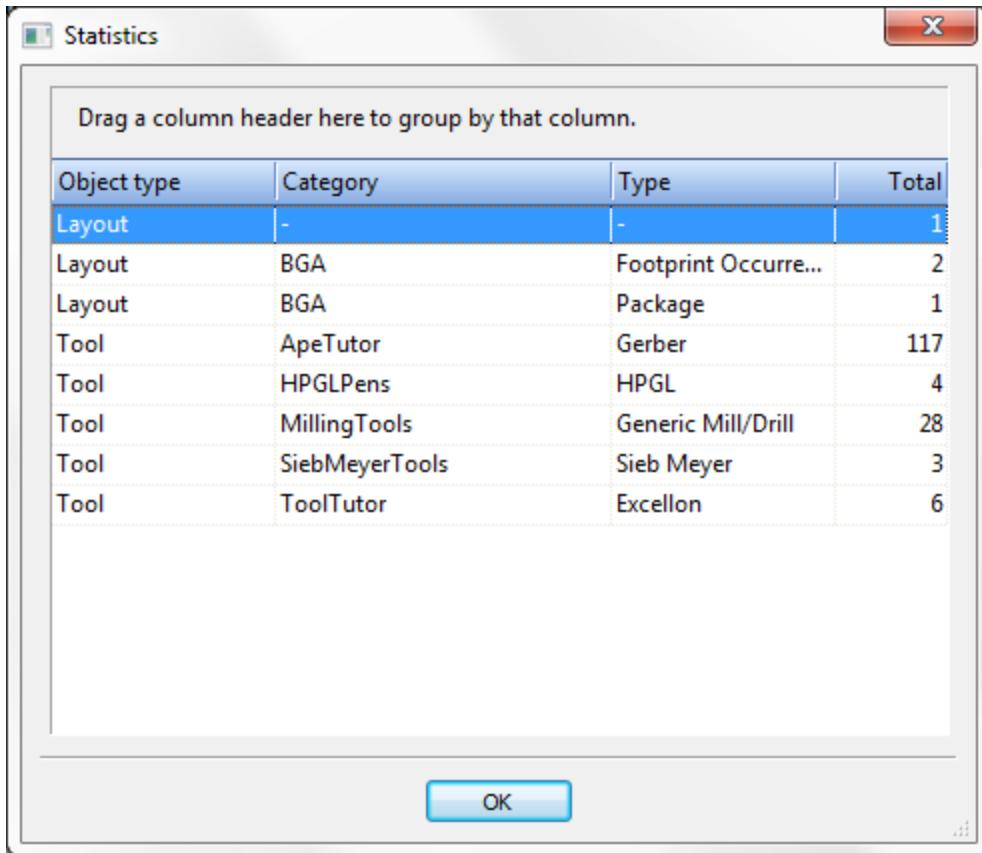
The **Area** text field in read-only mode shows an area of selected objects.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.2.11. Statistics

The **Statistics** window displays the number of tools and layouts in a document, either the current CAM file or the selected library. Entries are summed up based on type (package, land, etc.), category (BGA, PLCC, etc.). The **Statistics** dialogue for the current document is invoked via **File -> Statistics**. The **Statistics** dialogue for a library file is invoked through the context sensitive menu item **Statistics** in the **Libraries** view.

The results are displayed as dialogue box.



The screenshot shows a Windows-style dialog box titled "Statistics". Inside, there is a table with the following data:

Object type	Category	Type	Total
Layout	-	-	1
Layout	BGA	Footprint Occurre...	2
Layout	BGA	Package	1
Tool	ApeTutor	Gerber	117
Tool	HPGLPens	HPGL	4
Tool	MillingTools	Generic Mill/Drill	28
Tool	SiebMeyerTools	Sieb Meyer	3
Tool	ToolTutor	Excelon	6

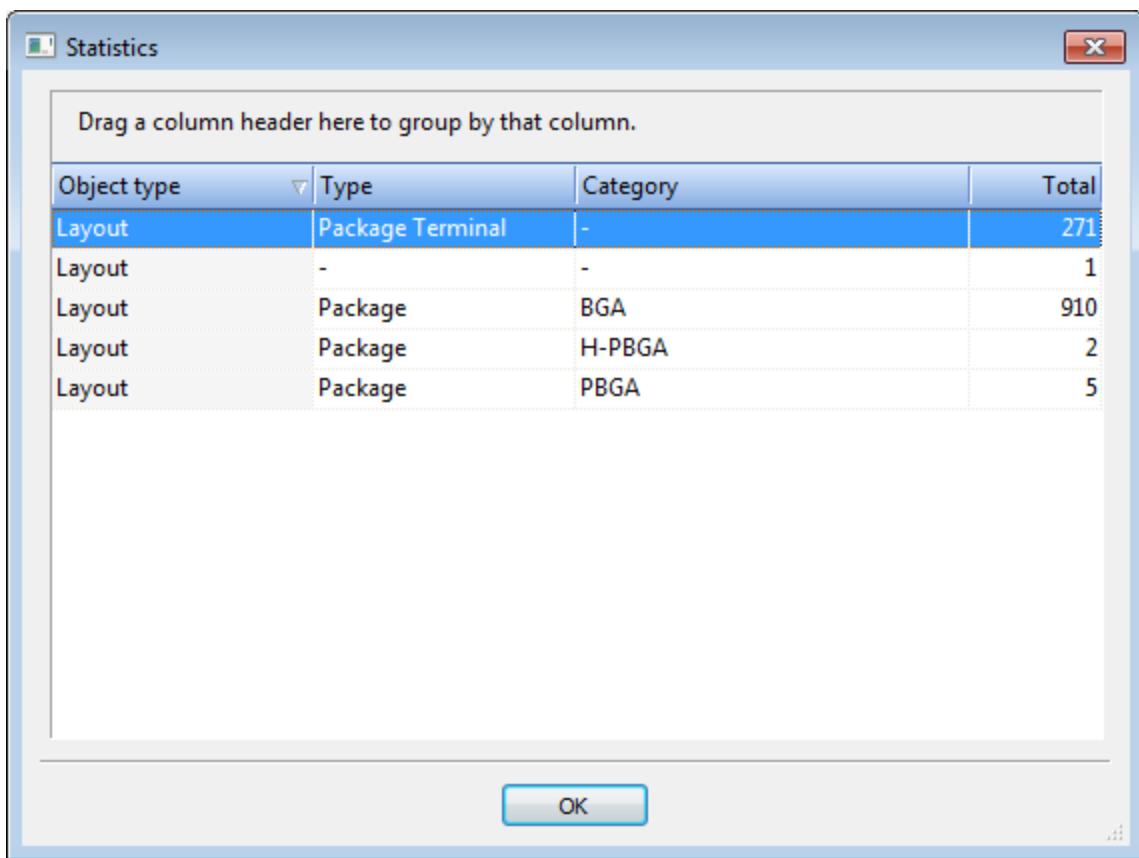
At the bottom of the dialog is an "OK" button.

Figure 153. Statistics window

There is a possibility to copy statistic results to clipboard by selecting the lines of interest and pressing **Ctrl + C**.

A user can group the entries in the **Statistics** dialogue according type, category and etc., by dragging a desired column header onto the upper part of the dialogue window.

In the below example it is shown the initial view of the **Statistics** dialogue.



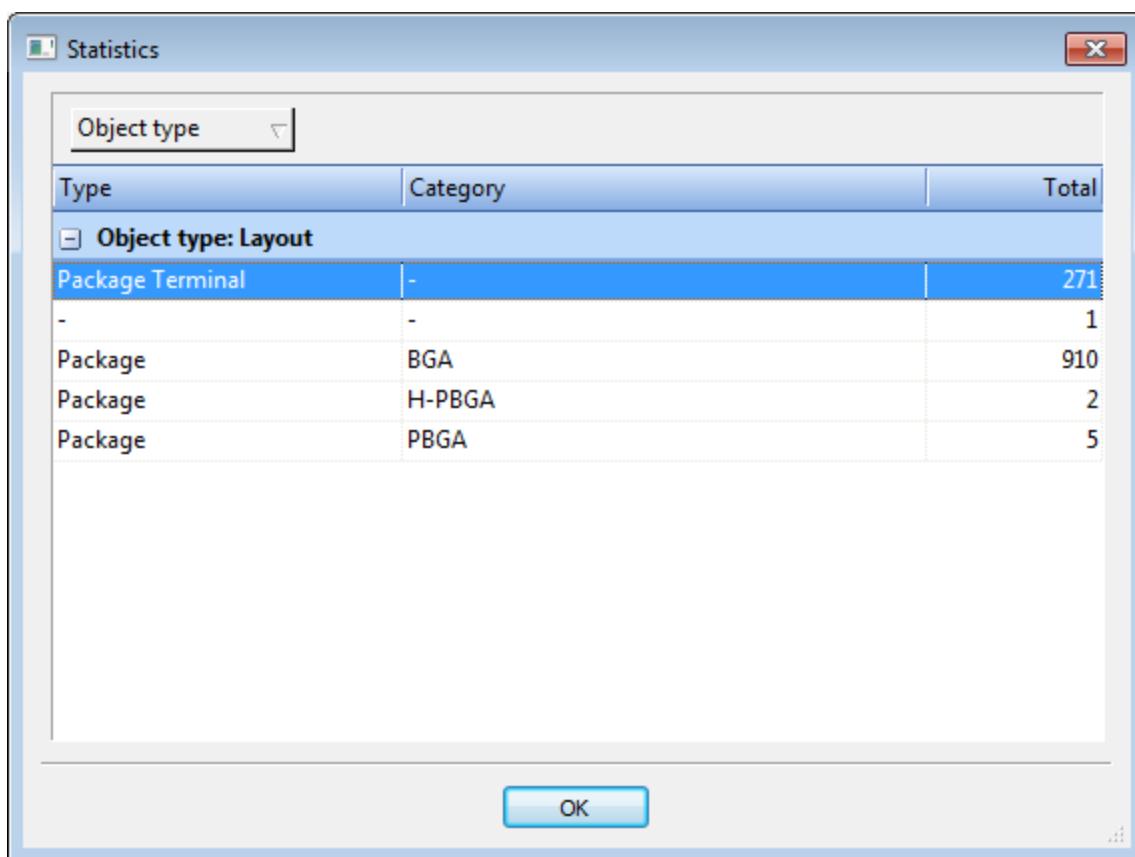
The screenshot shows the 'Statistics' dialogue box from CircuitCAM 7.3. At the top, it says 'Drag a column header here to group by that column.' Below is a table with the following data:

Object type	Type	Category	Total
Layout	Package Terminal	-	271
Layout	-	-	1
Layout	Package	BGA	910
Layout	Package	H-PBGA	2
Layout	Package	PBGA	5

At the bottom right of the dialogue is an 'OK' button.

Figure 154. Statistics dialogue for BGA-default library

- Press with a mouse on the **Object type** column header, hold and drag onto the area above the table. Two red arrows guides where the selection can be dropped.
- Release the mouse button. The column name appears on the top section of the dialogue. The entries are grouped by the object type in the **Statistics** dialogue.



The screenshot shows the 'Statistics' dialog box from CircuitCAM 7.3. At the top left is a dropdown menu labeled 'Object type'. Below it is a table with three columns: 'Type', 'Category', and 'Total'. The table is grouped under 'Object type: Layout'. The data in the table is as follows:

Type	Category	Total
Package Terminal	-	271
-	-	1
Package	BGA	910
Package	H-PBGA	2
Package	PBGA	5

At the bottom right of the dialog box is an 'OK' button.

Figure 155. Statistics entries grouped by Object type

- Press with a mouse on the **Type** column header, hold and drag onto the area above the table.
- Release the mouse button. **Types** appears under **Object type**, showing the sequence of grouping. The entries are grouped in the **Statistics** dialogue accordingly.



Note: The sequence of the grouping can be changed by moving column names. The sorting in the table can be changed by clicking on the *Column name*.

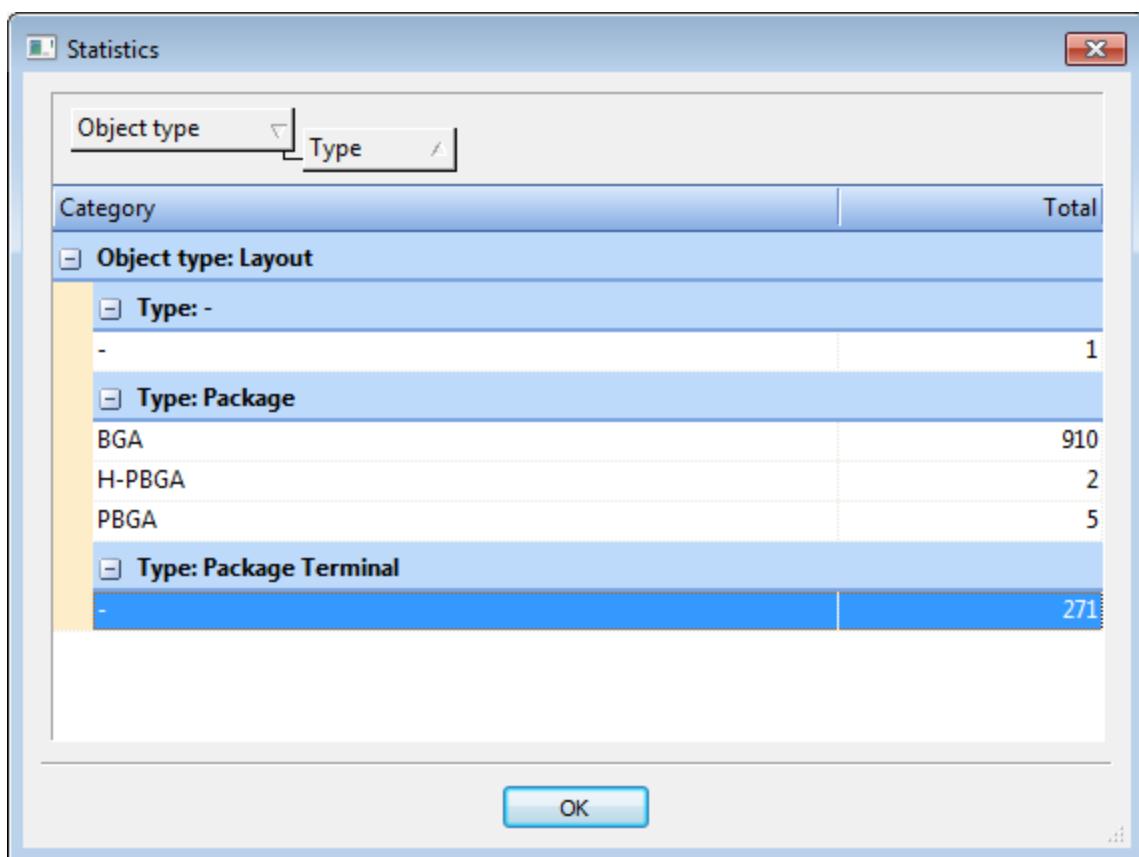


Figure 156. Statistics entries grouped by Object type and Type

To remove a grouping, drag a column name back onto the table column header.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.2.12. Batch and Background Processing

CircuitCAM can be started without or with additional program(command-line) parameters.

Parameters for the *CircuitCAM* can be specified:

- by dragging files onto *CircuitCAM* icon.
- specifying file names as command-line parameters.

Parameters can be:

- CAM file name
- CAT file
- production data file names (like *Gerber*, *GerberX*, *Excellon* data file names) and etc.

If argument is CAM file name, the file is opened and the contend is displayed in *CircuitCAM*, e.g.:

"C:\Program Files (x86)\CircuitCAM7\CircuitCam.exe" d:\Projects\tests\Tutor.cam

If parameter is CAT file name, a new CAM document based on specified CAT files is created (only in RAM, not saved).



Note: the additional flag/option "/cat" must be specified, otherwise CAT file will be opened as CAM file.

Here goes the example how to use flag "/cat":

```
"C:\Program Files (x86)\CircuitCAM7\CircuitCam.exe" /cat d:\Projects\tests\test1.cat
```

If parameters are production data file names, *CircuitCAM* will check if default CAT file has script job named "**Default**" and if jobs exists - it will be executed (more details how this can be achieved - see below), e.g.:

```
"C:\Program Files (x86)\CircuitCAM7\CircuitCam.exe" d:\Projects\tests\Tutor.boa  
d:\Projects\tests\Tutor.BOT
```

"Default" script job

"**Default**" is fixed name for a script job and this name gives special status for script job.

"**Default**" script job is automatically executed when *CircuitCAM* is started with additional parameters(not CAM or CAT file name as additional parameters, but with production data file names), e.g.:

```
"C:\Program Files (x86)\CircuitCAM7\CircuitCam.exe" d:\Projects\tests\Tutor.boa  
d:\Projects\tests\Tutor.BOT
```

In general "**Default**" is regular script job which might:

- contain various script tasks such as "Import" and etc. (see "[Script tasks](#)")
- call other jobs such as insulate, counter-routing and export.

"**Default**" script job might have the parameter "*Background processing*" ON and in this case no GUI will show up and *CircuitCAM* will terminate when execution of the job finished (see other script job settings "[Scripts \(Batch Jobs\)](#)").



Note: production data files can be specified parameters for the *CircuitCAM* and these might be automatically imported if the "**Default**" script job contains an "**Import**" script task.

The example content of "**Default**" script job can be:

- "**Import**" script task
- call the another script job "SMT Stencil"
- "**Save**" script task

Such a sequence of script tasks will perform import of production data, will call another script named "SMT Stencil" (the parameters for this stencil script job must be specified in the call job) and will save new created CAM file.

3.3. Graphic View

The graphic view of *CircuitCAM* is always visible, and typically occupies the largest portion of the program's interface. The graphic view is always displaying a **layout** from the current CAM document, by default the main layout is displayed. Another layout can be chosen from the [Layouts View](#). When in NC mode, [GMC](#) mode, or [tool edit mode](#), a special temporary layout is displayed.

The graphical data/objects in the currently displayed layout can be edited/created in the graphic view. Initially the main layout is displayed, another layout can be chosen from the [layouts view](#).

Graphic Objects

The contents of any layout consists of graphic objects, that are assigned to layers. All graphical objects define an area except for paths with a width of zero. We distinguish between basic graphic objects and composed ones.

The basic graphic objects are:

- a. [Rectangle](#): A rectangle is defined by two opposing points, and is oriented parallel to the X and Y axis.
- b. [Circle](#): A circle is defined by a centre point on the coordinates, and a diameter.
- c. Path: A path is a sequence of lines and arc segments. A path maybe a [Closed Path](#) -start and end point connected together- or it maybe and [Open Path](#) path -with distinct starting and ending points.
 - Line Segment: Straight lines which connects two points together.
 - Arc segments: Circle arc segments which connect two points on a circle. Because of numerical stability, a circular arc is typically limited to 270 degrees, arcs bigger than that are automatically cut into two smaller arcs.
 - [Circle in Closed Path](#): A path which consists of a single segment that is one full circle.
- d. [Polygons](#): Any closed figure defined by a border which is the outline; this border consists of a sequence of lines and arc segments, similar to those of a closed path. Normally these lines and arc segments are not allowed to touch or cross each other.
- e. [Polygon Cutouts](#): It is a polygon with an outer border and one or more inner borders cut into it.



Note: When a rectangle is rotated, it is converted to a polygon.

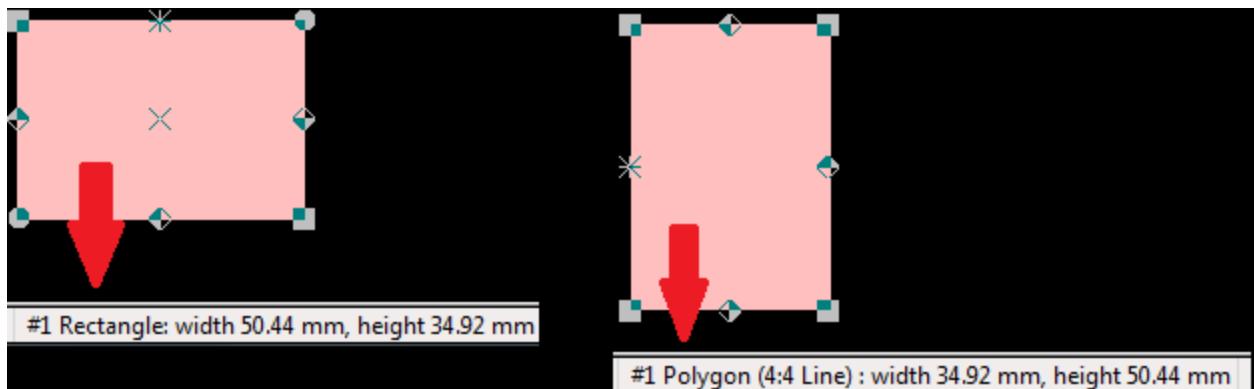


Figure 157. When a rectangle is rotated, it is converted to a polygon.

The composed graphic objects are:

- a. **Flashes:** A flash is defined by a configItem (toollist), a coordinate, and rotation.
- b. Sub layout instance: The sub layout stores within it various information, like; the layer information, the rotation and orientation of objects, sizes of objects.



Note: When creating text or data matrix, they are decomposed into basic graphical objects, *e.g.: a data matrix is decomposed into several small scaled rectangles.*

For more details about graphical objects and inserting them, please refer to: [Insert](#)

Display Grid

The grid is a set of visual points that are laid symmetrically, these grid points will help you align your graphical objects with ease.

To show and hide the grid, simply click on the **Show Grid** button in the **Unit/Grid toolbar**.



Figure 158. Unit/Grid toolbar

The Grid Spacing menu, enables you to choose the distance between grid points.

The **Measurement Units** drop menu, allows you to choose the unit which will be used for all measurements in CircuitCAM, you can also set the measurement unit from the [General Settings menu](#).

Snap to Grid

The snap to grid feature explains as follows: When you draw, resize, or move a shape or an object, it will align or "snap to" the nearest point in the grid, even if the grid is not visible. If your shape or object does not move where you want, turn off the snap to grid option or choose a smaller scale grid.

You can change the value by which the mouse moves to the grid from the **Cursor Grid Spacing** in the toolbar.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.3.1. Zooming and Panning

Navigating through a document or a project in *CircuitCAM* is made easy, explained in this page is the various means of navigation used in *CircuitCAM*.

Zooming

The process of zooming, is simply changing the scaling of the viewed workspace. The viewable area of a document/workspace can increased or decreased.

Zooming options are also accessible through [View](#) menu and [Zooming Toolbar](#).

Zooming In and Out

The graphical representation is shown enlarged or minimized by the factor of two (*half and double*).

Either using keyboard hotkeys **Page Up** and **Page Down**, or using the mouse scroll wheel, the current cursor position is used instead of the screen centre point. If the mouse cursor is outside the graphic view area, the last recorded mouse position will be used as zooming point.

Zoom Window

Allows drawing a rectangle around a specific area to bring it into focus. This function is accessible by holding down the Alt button and drawing a rectangle around the desired place.



Figure 159. The process of Zoom Window

Zoom Selected

This function re-scales the view to fit the selected object(s).

This function is accessible from the view menu, or from the zoom toolbar.



Figure 160. The process of Zoom Selected

Overview

When using this function, all graphical objects are shown fitting the screen.

This function is accessible from the view menu, zoom toolbar, or keyboard hotkey **Home**.

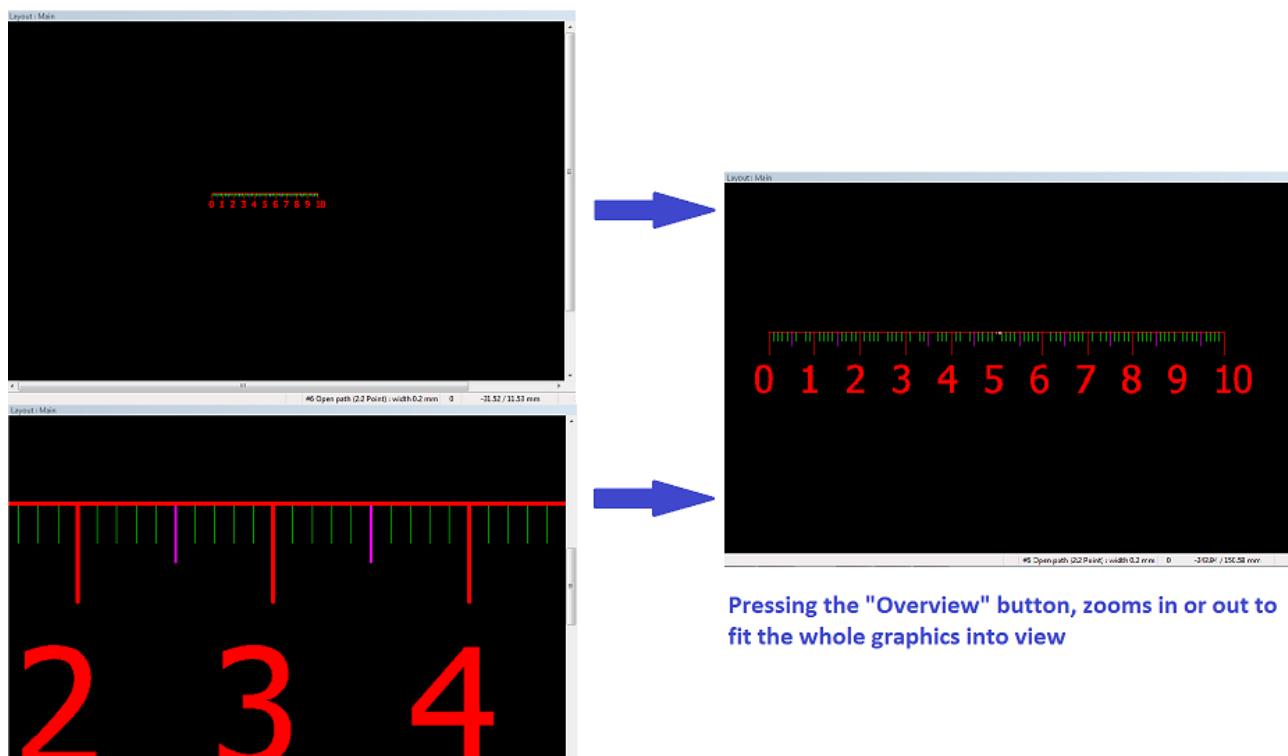


Figure 161. Zoom Overview

Previous Views

CircuitCAM stores a history of the previous views, CircuitCAM saves the last twenty views (current view + nineteen old views).

The previous view can be restored through the view menu, zoom toolbar, or keyboard hotkey **Insert**.

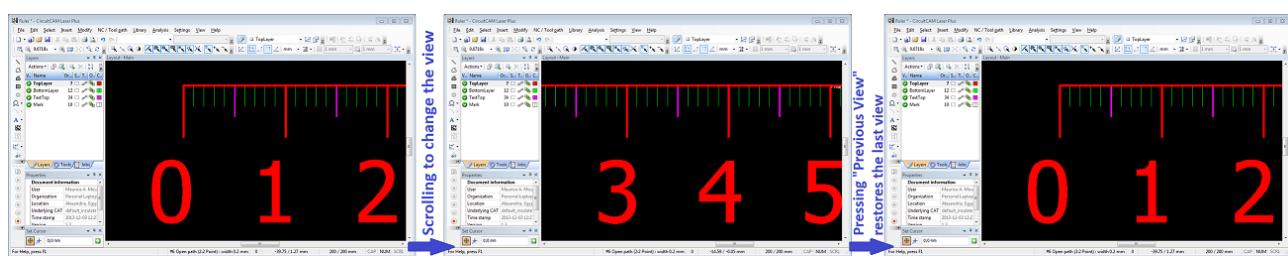


Figure 162. Previous View; after clicking **Insert** the last view is restored

Panning

Panning (changing visible area of the layout) is performed by dragging the layout at any place in the Graphic View while the **Right Mouse Button** is held, or, by simply clicking the place to view in the [Preview Window](#). Alternatively you can also use the horizontal and ver-

tical scrollbars. The size and location of the scrollbars is relative, the length of the slider indicates the current visible region of the layout.

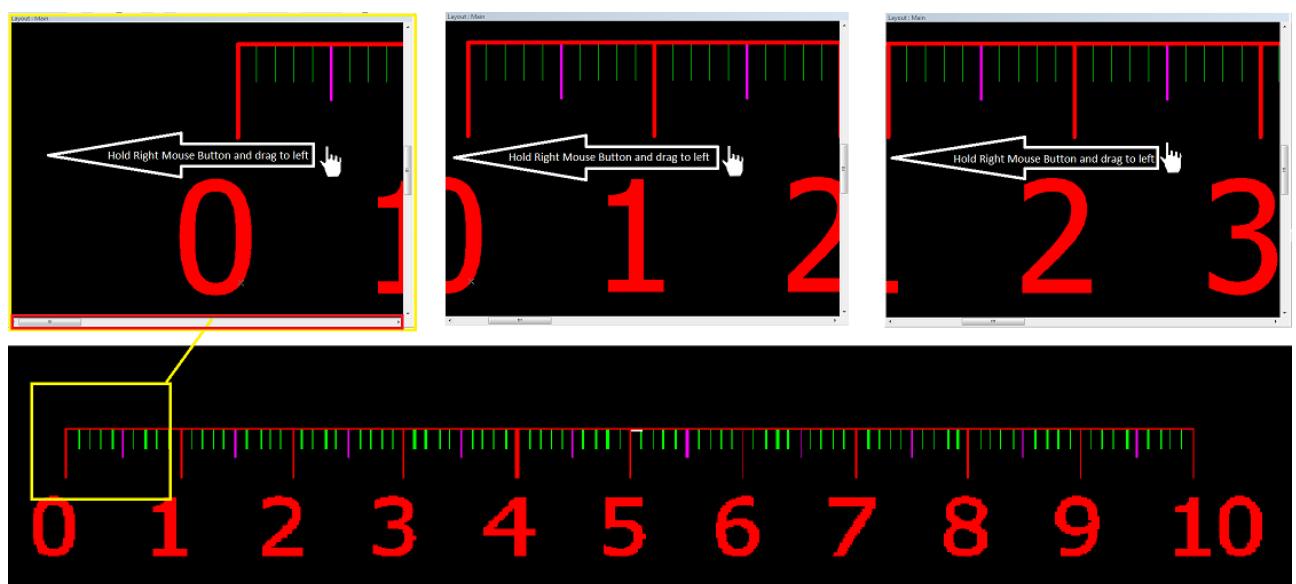


Figure 163. Right click panning in Graphic View.



Figure 164. Navigating by selecting a place from the Preview window

3.3.2. Manipulation of Graphic Objects

Manipulation operations for graphic objects affect the single points or segments, which are identified by the selected marker, or work with a total group of selected objects.

Single Object

Manipulation operations on paths and polygons belong to the Single object group. Any *manipulation operation* can be called by using one of the following methods (for more information on the methods see [Manipulation operations](#)):

- **Curve** submenu of the **Modify** menu or keyboard **Ctrl + character**.
- **Unit/Grid** toolbar.

Group of Objects

Some manipulation options are available for a group of selected objects. Almost all of these operations are available for a single selected object. The most of the functions are called from the **Modify** menu. When the mouse is in the graphic window, the right mouse click will access a *context menu* for editing of selected objects (see [Right Mouse Button Menu](#)).

Markers

The single selected object has one or more markers, which can be accessed by individual manipulation operations. The number and shape of these markers depend on the type and complexity of the selected object.

The following markers can appear on the selected object:

- - anchor point marker;
- - centre marker;
- - point marker;
- - supplemental point marker (appears only on rectangles);
- - arc segment marker (the dark side of the marker image indicates the direction of a path/polygon);
- - line segment marker (the dark side of the marker image indicates the direction of a path/polygon).

While using various functions, it is needed to make precise mouse movement; it is needed that the mouse snaps to specific points on the previously created (*already existing*) objects. To achieve this, while the selected function is active (*whether; drawing, editing, moving, or measuring*), holding down the shift button and clicking on an object will display all the marker points of the object. When moving the mouse over a marker point, the cursor will snap to the exact position of this point.

Clicking the mouse button while the shift button is pressed does not alter or change the selection.

Circle:

- Two markers opposite to each other on the perimeter - for the size change.
- One marker in the Centre - for moving.

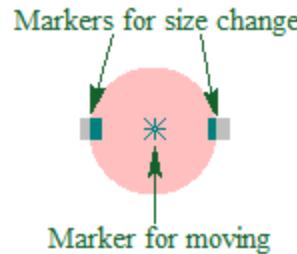


Figure 165. Circle markers

Rectangle:

- Four markers at the corners - for the size change
- Four markers on the edges - for the size change.
- One marker in the Centre - for moving.

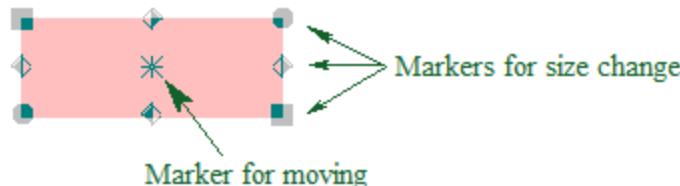


Figure 166. Rectangle markers

Path and Polygon:

- One marker at each structural point (start point, end point and line support point).
- One marker on each arc or line segment.
- One marker on the Middle point of every arc segment.

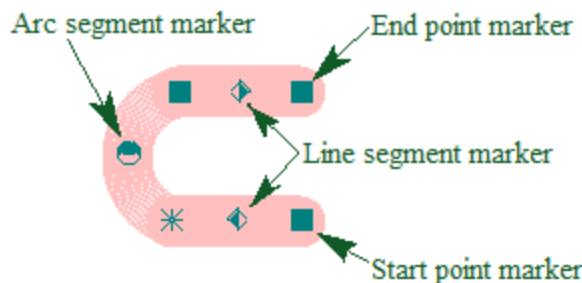


Figure 167. Path with arc markers

Flash:

- Only one marker at the point of origin.



Figure 168. Flash marker

Depending on the size of the object and the zoom factor, only some of the markers may be visible. *CircuitCAM* displays marks on the screen, which do not overlap with their neighbours.

One of all displayed markers may exist as the selected marker or **Anchor Point**. The marker is indicated by * and is used as:

- reference point for the move/duplicate group,
- reference point for the rotate/mirror group,
- starting point for path/polygon manipulation operations.

Coordinates of the **Anchor Point** are displayed in the [Status bar](#), together with actual coordinates of the cursor. Please notice, after the single object is selected, the *Anchor Point* is set to the coordinates of the selected marker. The *Set Anchor* function (see [Set Anchor Point](#)) allows a user to change an anchor point coordinates.

To move from one marker to a neighbouring marker, press the **Plus** (*Next* marker) or **Minus** (*Previous* marker) key. Press and hold down one of the keys to move the marker repeatedly. Particularly in case of a path and polygon, it is helpful to know that the *Next* marker moves the selection attribute in a direction of the structural sequence and the *Previous* marker moves respectively in the opposite direction. In case of a closed path/polygon the marker moves in a loop.

The single Object Manipulation Operations

Add Point/Segment

Inserts a new support point between two existing ones, or adds a new support point at the beginning or the end. It depends on what support point is chosen – start/end point or one of segment middle points.

1. **Line segment:** A new support point is inserted between existing support points.
2. **Start or end point of a path:** Appends new support points/segments to the existing path.

Cut Point/Segment

Divides a path or a polygon into two parts, with polygons/closed Paths always first converted into paths.

1. **Central support point:** The path is divided into two paths at this point.
2. **Segment:** The selected segment is divided into two parts at the segment middle point.

Delete Point/Segment

The single support point or a complete segment with adjacent support points is removed from a path or a polygon. If during this operation less than two supports points remained, the path or polygon is completely removed. The following cases are possible:

1. **Start or end support point:** The support point is removed together with the connected line or arc segments.
2. **Central support point:** The central support point is removed. The line segment links the adjacent support points.
3. **Line segment:** The line segment with adjacent support points is removed.
4. **Arc segment:** The arc segment is converted into a line segment.

Move Point/Segment

Moves a single support point or a segment.

1. **Support point:** If a support point is moved, the neighbouring segments are changed respectively. With arc segments, the bulge is kept. The direction and length of the segment is changed.
2. **Line or arc segment:** A shape, a length and a direction of the selected segment are retained. However, a length and a direction of neighbouring segments are changed.

Create or Move Arc

Changes a line segment into an arc segment or the bulge of an existing arc. It works only on segments and not on line support points. The support points adjacent to the segment are not changed.

1. **Line to arc:** A line segment is converted into an arc segment.
2. **Change arc:** The bulge of an existing arc segment is changed.

Combine

Combines paths, where the ends are closer than a minimum specified distance (**Gap distance** value in the **Curve combine** dialogue), to a single path. The statistics of the *Combine* operation is presented in the **Message Log** view:

x path connections were done, x resulted in a closed path(s).

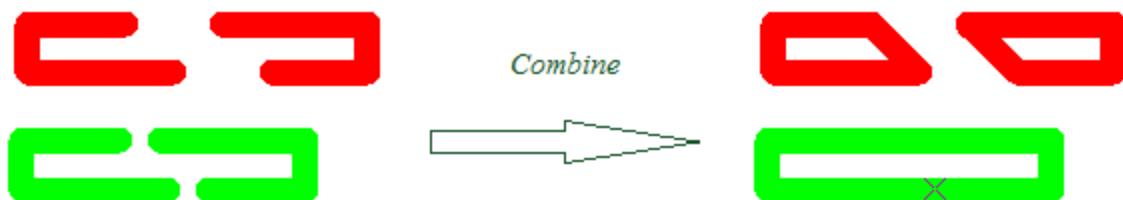


Figure 169. Combine operation initial data and result

Close

The selected open path is converted to the closed path. An additional line segment is added to connect start and end points.

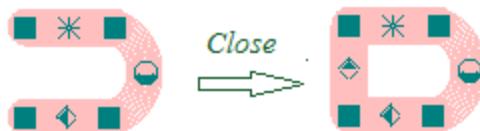


Figure 170. Close operation initial (open path) data and result (closed path)

Path to Polygon

All selected paths are closed to polygons, therefore losing their width.



Note. New polygons take up a different area in comparison to the paths area before the operation. Furthermore, all paths inside selected flashes are closed to polygons too (including nested flashes).

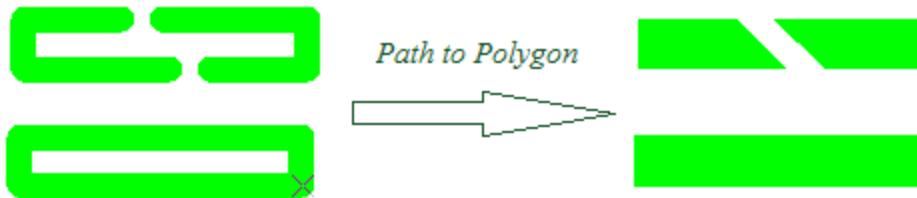


Figure 171. Path to Polygon initial data and result



Restrictions:

- A path with just one line segment cannot be converted into a polygon.
- The paths must not intersect. Please check this.

Combine to Polygon

The purpose of this function is to combine paths, where the ends are closer than a minimum specified distance, to close them and convert them to polygons.

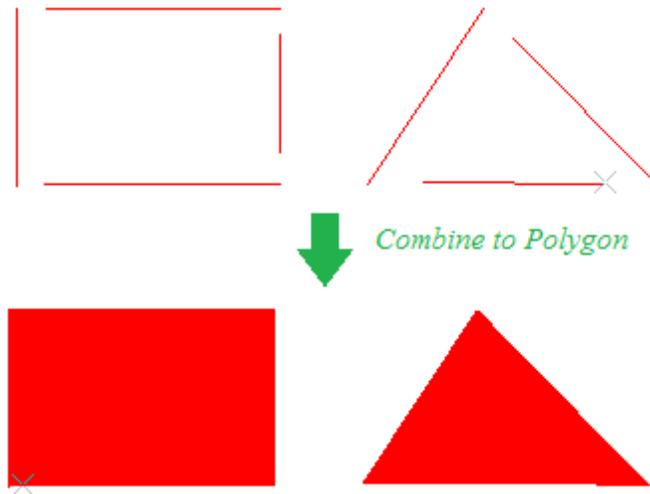


Figure 172. Combine to Polygon initial data and result

Convert to Polygon

Any selected simple graphic object, i.e. circle, rectangle and path is converted into polygons of the same shape. The original objects are deleted. This is useful where targeted manipulations must be carried out on these objects.

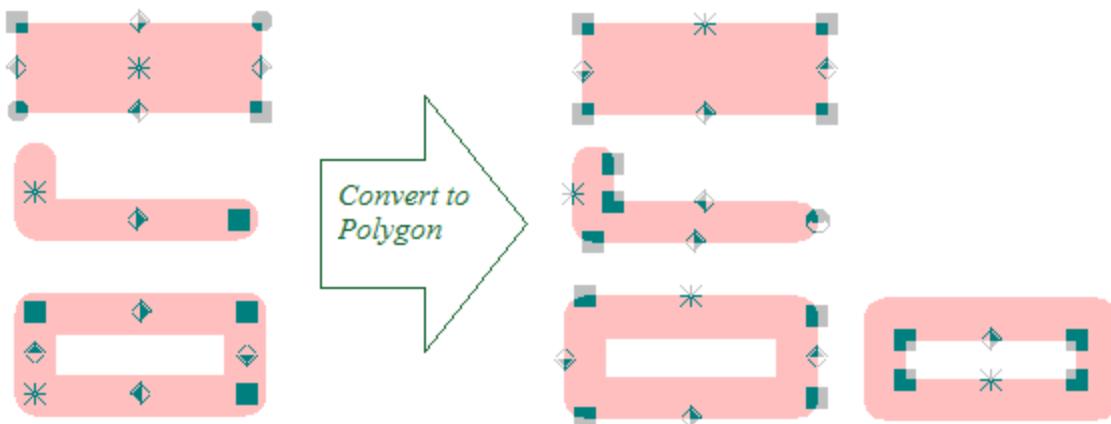


Figure 173. Convert to Polygon initial data (rectangle, open path,closed path) and result (polygons)



Note. Since objects remain the same shape and belong to the same layers, no changes are visible on the screen, except for the markers - they are generated anew.

Convert to Closed Outline Path

All selected graphics objects (circle, rectangle, polygon, flash, etc.) are converted into an outline path with a zero path width. The original objects are deleted.

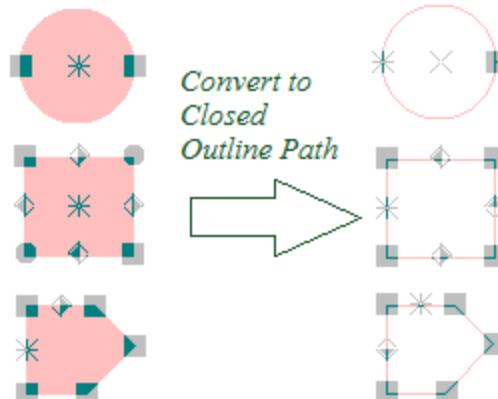


Figure 174. Convert to Closed Outline Path initial data (circle, rectangle, polygon) and result (outlines)

Denest Polygon Cutout

The selected *polygonCutOut* (boundary) and all it's *cutOut's* (inner figures) are replaced in a graphic window with their origin shapes (circle, rectangle, path and so on).



Figure 175. Denest Polygon CutOut initial data (selected one hole) and result (denested all holes)



Note. This function is only available in the *CircuitCAM* variants STANDARD and PRO.

Denest polygon Cutout Hole

Only selected *cutOut's* (inner figures) are replaced in a graphic window with their origin shapes (circle, rectangle, path and so on). In case the *polygonCutOut* has no more *cutOut's*, it is denested too.

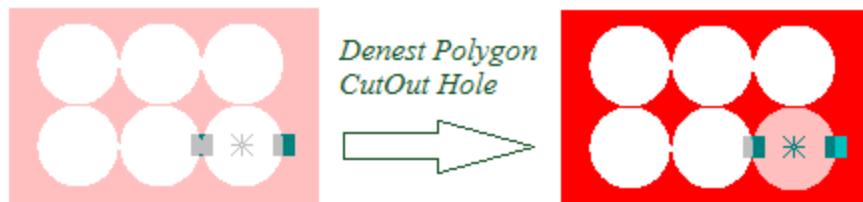


Figure 176. Denest Polygon CutOut Hole initial data (selected one hole) and result (denested only selected hole)



Note. This function is only available in the *CircuitCAM* variants STANDARD and PRO.

Convert to Circle

Converts the selected simple graphic object (i.e. flash, rectangle and path or a group of selected objects) into a circle of the same area. The original object is deleted. In case of a group of objects, all these objects must belong to the same layer. The destination layer coincides with the source layer.

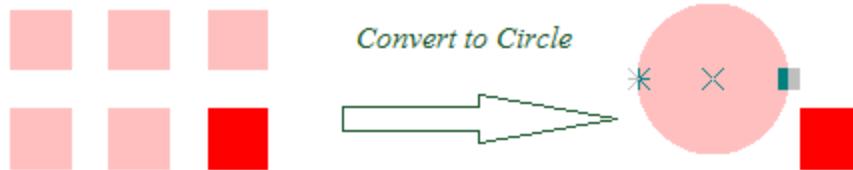


Figure 177. Convert to Circle initial data (selected rectangles) and result (circle)



Note. To view an area of selected objects use *Area of Selected Objects* command. For a single object, the area is defined in the **Properties** view.

Draw to Flash

Converts selected objects to flashes.

Selected objects, which make a connected area and belong to the same layer, are converted to a flash. For absolutely similar areas only one tool is created ("similarity" depends on **Tolerance 'Draw to Flash' [%]** value).

Draw to Flash always keeps the original structure of selected objects to the additional '*DrawToFlash_Backup*' layer; which is accumulated but never deleted.



Note. After *Activity-Cancel* a backup for the last selected group is not created (i.e.: '*DrawToFlash_Backup*' remains the same which was before last *Draw to Flash* operation).

You can split a layout (select not all objects at once) and run *Draw to Flash* at several times – the final result will be the same.

The configuration must be given as a destination place (any object with these attributes should be selected before). All flashes after converting remain on its origin (drawing) layers.



Note. The *Tolerance 'Draw to Flash' [%]* is set in the **General Settings -> Calculation Parameters** sheet (to see/modify it, open the **Settings** menu, select the **General settings...** line and click on the **Calculation Parameters** node on the left side of the window). This function is available in the *CircuitCAM PRO* variant only.

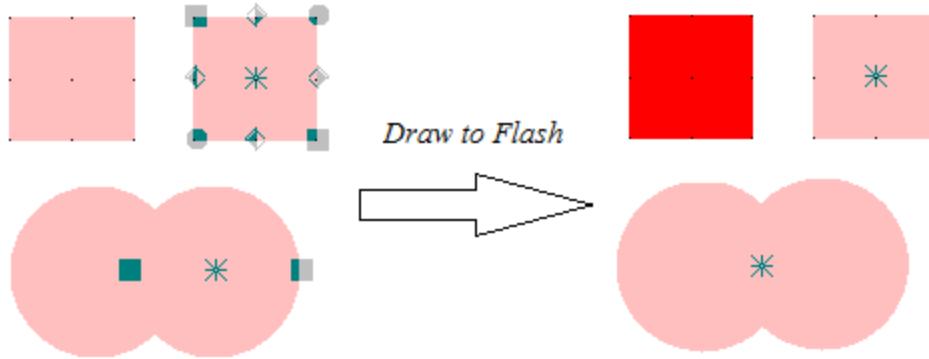


Figure 178. Draw to Flash initial data (selected rectangles and overlapping circles) and result (generated flashes)

Convert to One Flash

Converts selected objects into a single flash.

All selected simple graphic objects (i.e.: circle, rectangle, flash, polygon and path) are converted into a new flash. The **Configuration** field of dialogue-box allows a user to select/change a tool list for the new flash.

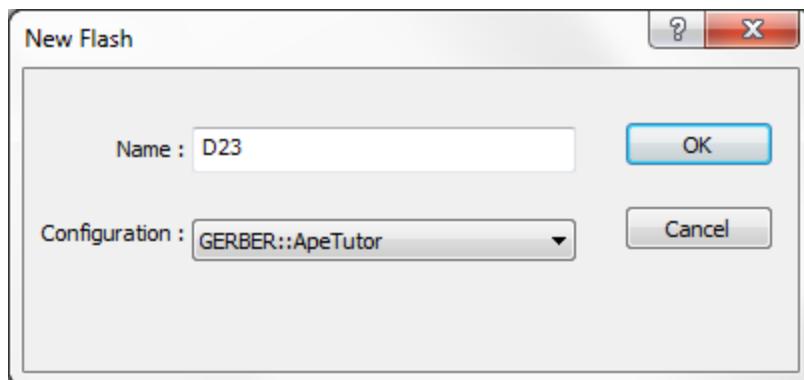


Figure 179. New Flash dialogue window

After the **Configuration** is set, the next suitable tool name is suggested in the **Name** field automatically, but user can enter an applicable tool name, too.

In addition, a mass point or centre point is calculated on selected objects. The type of a point depends on *Point method* setting – **Mass** or **Centre** in the **General Settings** dialogue.

The original objects are deleted and a new object is made as selected.

In case the original objects belonged to different layers, a new object is placed on one of them randomly.



Figure 180. Convert to One Flash initial data (two object selected) and result (only one object selected)



Note: This function is only available in the *CircuitCAM* variants STANDARD and PRO.

Convert to Instance

Converts selected objects into a single instance/layout.

All selected simple graphic objects (i.e. circle, rectangle, flash, polygon and path) are copied into a new layout. The **Name** field of dialogue-box is for defining of a layout name, fields **Type** and **Category** allow a user to select additional layout properties - type and category.

Flag **Insert instance** - if *On*, creates a new instance of the selected objects in the current layout, otherwise selected shapes remain in the current layout without any changes.

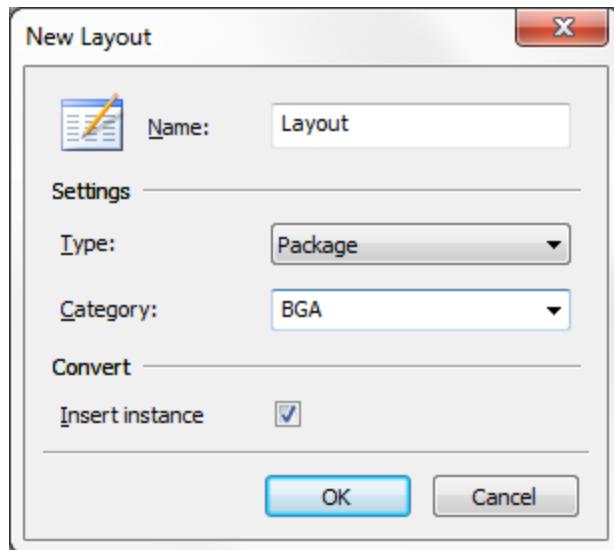


Figure 181. New Layout dialogue window

Exchange with Flash

Compare Shape

A piece of a topology, which is not alone flash and is identical to the selected flash, is replaced with a flash.

The original objects are deleted and all new objects are made as selected.

The **identical** means not only the same amount and size of objects, but the same relative locations too - any difference can't exceed the **Tolerance 'Draw to Flash' [abs]** value.

The **Tolerance 'Compare shape/BBox/Outline' [abs]** is set in the **General Settings -> Accuracy** page (accessible from **Settings -> General settings...** menu).



Note. This function is available in *CircuitCAM* PRO variant only.

Compare Bounding Box

A connected piece of a topology, which size of the bounding box (on both X and Y directions separately) differs from a bounding box of the selected flash less than the **Tolerance 'Draw to**

'Flash' [abs] value, is replaced with a flash. Any bounding box is assumed as the "black box".

The original objects are deleted and all new objects are made as selected.

The Tolerance "Draw to Flash" [abs] is set in the **General Settings -> Accuracy** page (accessible from **Settings -> General settings...** menu).

The function is based on resolution of *Polygon Package*.



Note. This function is available in *CircuitCAM PRO* variant only.

Compare Outline

If a difference between an area of the connected piece of the topology and an area of the selected flash when they are matched according to their mass points is less than the **Tolerance 'Draw to Flash' [abs]** and **Tolerance 'Draw to Flash' [%]** values, the piece is replaced with a flash.

The original objects are deleted and all new objects are made as selected.

The **Tolerance 'Draw to Flash'** is set in the **General Settings -> Accuracy** page (accessible from **Settings -> General settings...** menu).

The function is based on resolution of *Polygon Package*.



Note. This function is available in the *CircuitCAM PRO* variant only.

Step and Repeat

Selected objects are repeated in a matrix $((x*y) - 1)$ times in a right-up direction, according to the settings in the dialogue window.

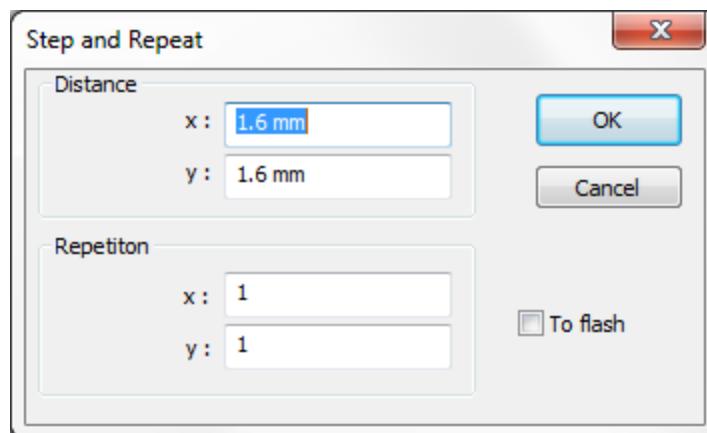


Figure 182. Step and repeat dialogue window

- **Distance:** defines a space between repetitions. The default values are calculated as the bounding-box of selected objects.
- **Repetition:** specifies a size of repetition matrix. The default value is [1, 1] (no repetitions).
- **To flash** flag - if **On**, converts selected objects to one flash (like *Convert to Flash* operation) and later duplicates it in accordance with settings in **Distance** and **Repetition** areas.



Note. This function is available in the *CircuitCAM PRO* variant only.

Create Bounding Shape

Creates a circular bounding shape one per each group of overlapped graphic objects. This function works on one layer, because the fiducials are located on a separate layer. The **Source layer** combo box indicates a name of the source layer. If the source layer is empty, *CircuitCAM* will display a message.

There is a special (so called *Butterfly-Fiducials*) case, when two square flashes do not overlap and *CircuitCAM* creates two circles. To avoid this the **Merge distance** value is introduced. If a distance between objects are less than **Merge distance** value, these objects are merged and just one circle on the centre point of the objects is created.

The original objects are not deleted. The destination layer is selected from the **Destination layer** combo box.

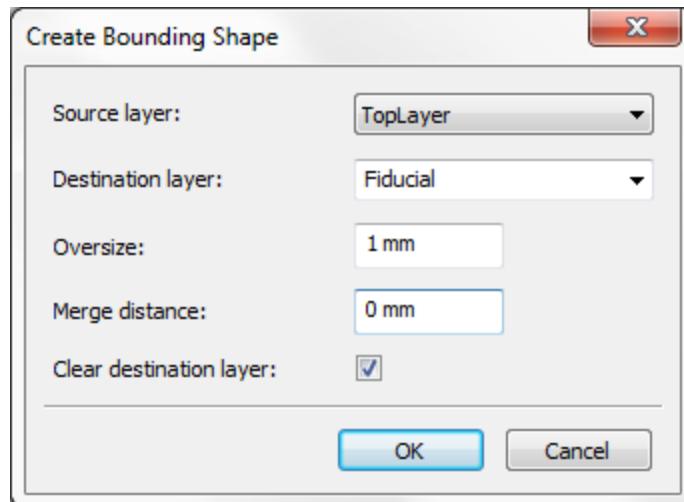
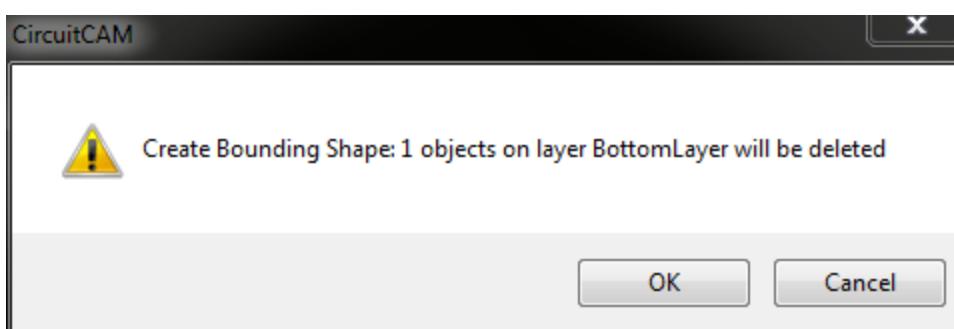


Figure 183. Create Bounding Shape dialogue window

The **Oversize** field allows creation of a bigger or smaller bounding shape.

If the **Clear destination layer** check box is set *On*, the destination layer will be cleared out before creating the new bounding shape, a confirmation message will appear to approve the removal of objects.



Note: This function is only available in the *CircuitCAM STANDARD* and *PRO* variants.

Create Outline

Generates an outline on the edge of each object located on the **Source layer**. This function works on one layer, selected in the **Source layer** combo box. If the source layer is empty,

CircuitCAM will display a message.

The original objects are not deleted. Resulting outlines will be placed on the **Destination layer**. A destination layer is selected from the **Destination layer** combo box (a destination layer will be created if a new name was entered).

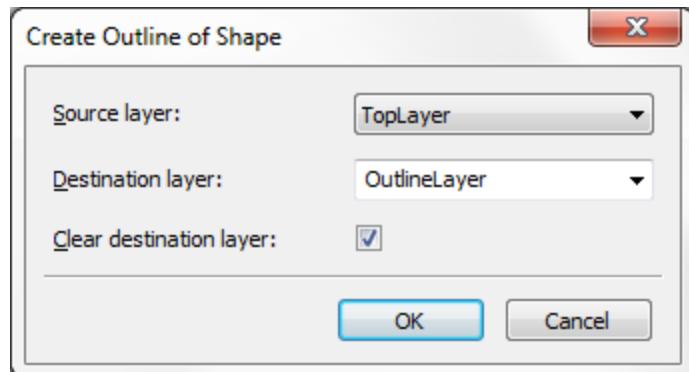
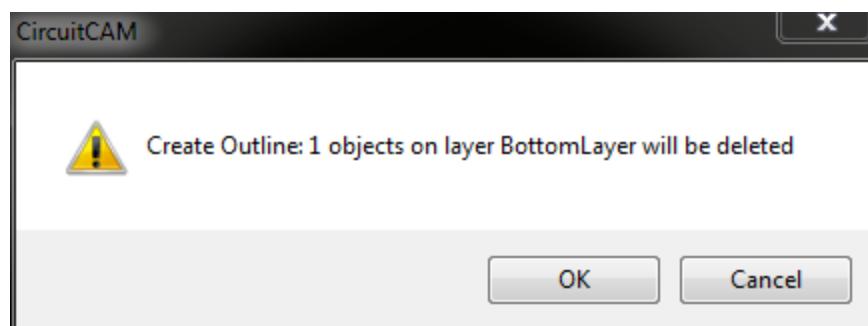


Figure 184. Create Outline of Shape dialogue window

If the **Clear destination layer** check box is set *On*, the destination layer will be cleared out before creating the new outlines, a confirmation message will appear to approve the removal of objects.



Note: This function is only available in the *CircuitCAM STANDARD* and *PRO* variants.

Perfect Shape

Checks selected objects on a perfect rectangle, a rounded rectangle, a bevelled rectangle, a circle and an oval according to the given tolerance. If yes, converts the perfect shapes into flashes of mentioned types.

New created apertures are saved in the selected configuration (**Aperture/Tool List**).

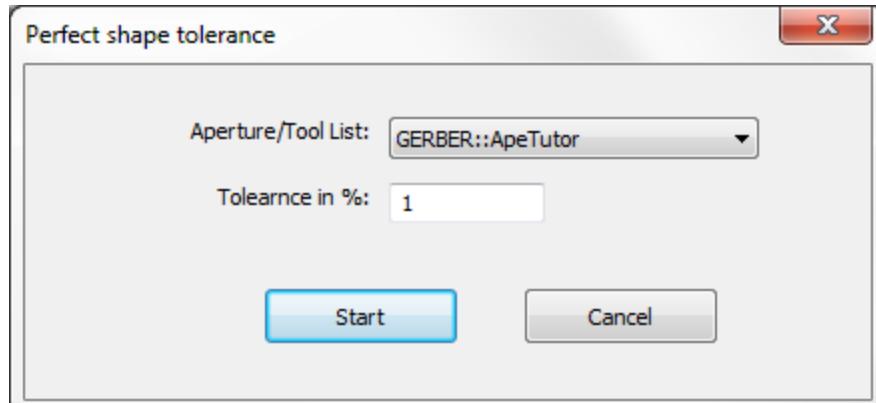


Figure 185. Perfect shape tolerance dialogue window



Note. This function is available in the *CircuitCAM* PRO variant only.

Polygon to Path

Converts polygons to paths.

Remove Holes

Removes holes inside a closed area of pp-polygons.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.3.3. Boolean Operations

CircuitCAM provides basic set operations on graphical objects, the applicable set operations include; union, difference, intersection, XOR, paths on area, and paths off area.

To perform Boolean operations properly, some notes must be taken into consideration;

- The primary selected object (the object with markers) is the main operator for Boolean operations, the rest of the selection group will be considered as one polygon and will be treated as the secondary operator.
- Boolean operations only affect graphical objects that are present on the same layer.
- Boolean operations only affect graphical objects that are overlapped.

Explained next are the various Boolean operations included in *CircuitCAM*, a unified example will be used to demonstrate these operations.

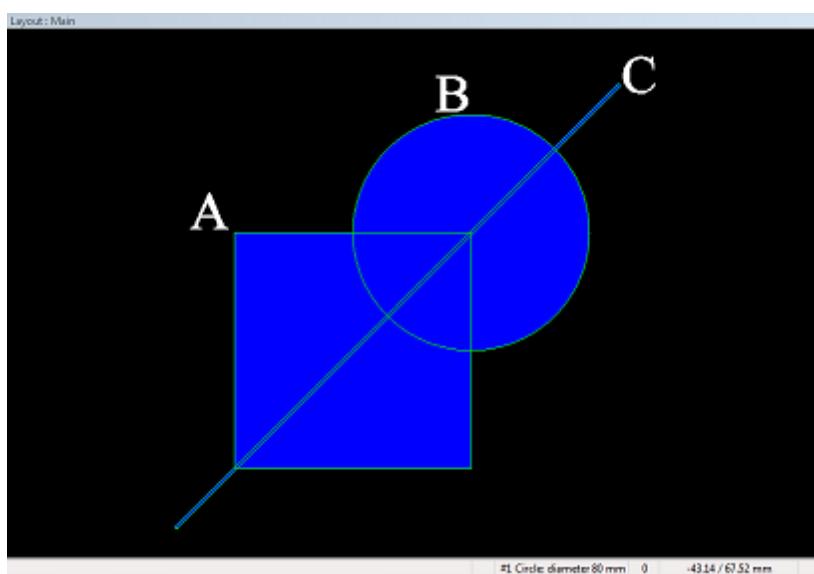


Figure 186. The plain example used to demonstrate Boolean operations; object A is a rectangle, object B is a circle, object C is an open path

Union

All graphical objects whether polygons or paths, are eligible to the *union* operation, known from the theory of sets.

The union operation is the process of merging the selected overlapped graphical objects to create one new polygon represented on the same layer.

The union operation is simplified as the word "*or*", e.g.: in the figure below; the operation is: **Graphical objects available in; A or B or C.**

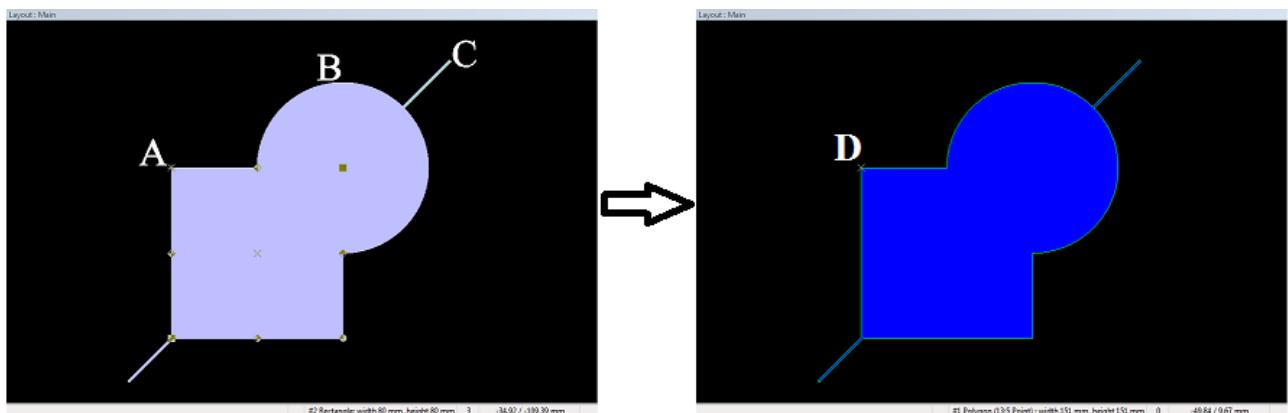


Figure 187. After the three objects are selected, the union operation is executed, the result is a new polygon D



Note: The union operation is still effective even if there is not a primary selected object.

Difference

All graphical objects whether polygons or paths, are eligible to the *difference* operation, known from the theory of sets.

The difference operation is the process of calculating the graphical differences between the main operator and the secondary operator(s), and representing those differences as a new polygon(s) on the same layer.

The difference operation is simplified as the word "*not*", e.g.: in the figure below; the operation is: **Graphical objects available in; A not B.**

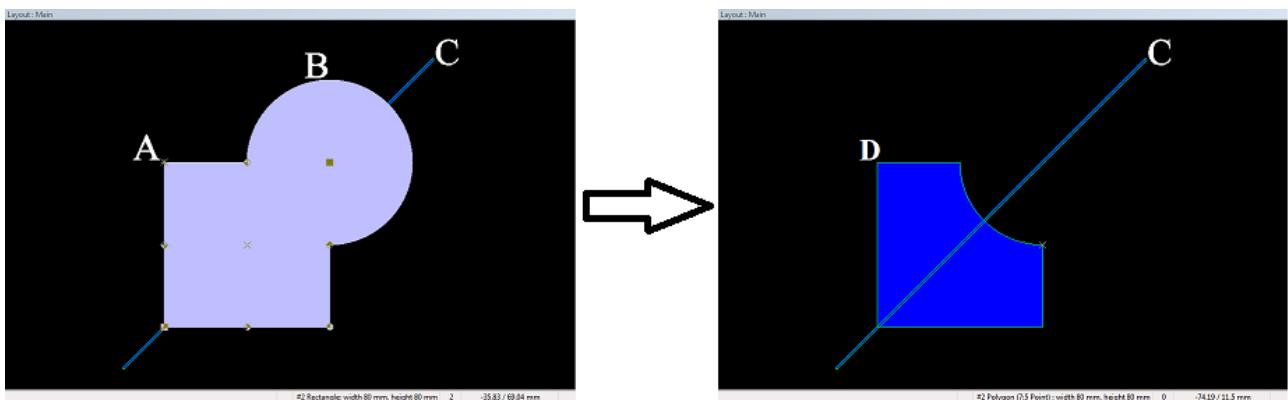


Figure 188. After the two objects A and B are selected, A is the primary selected object (operator);
the difference operation is executed, the result is a new polygon D

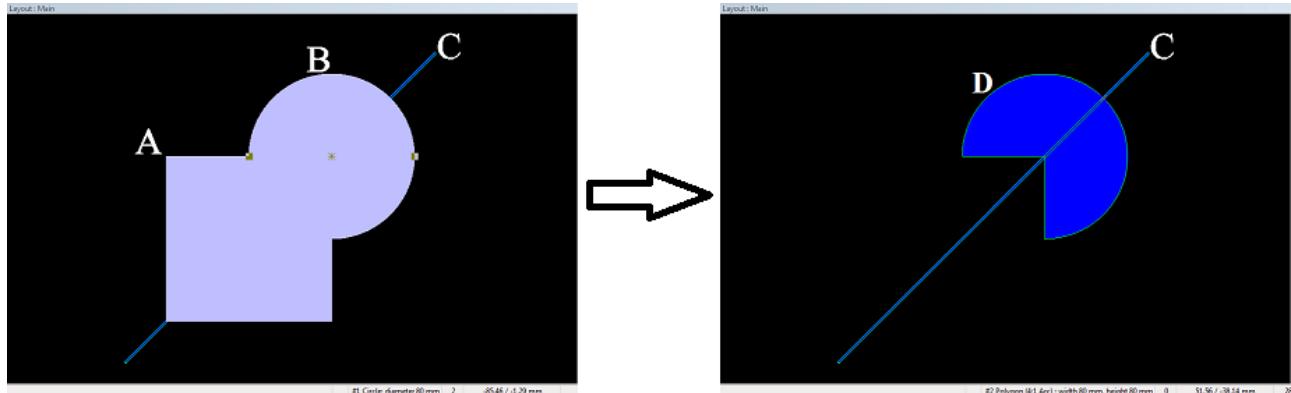


Figure 189. After the two objects A and B are selected, B is the primary selected object (operator);
the difference operation is executed, the result is a new polygon D

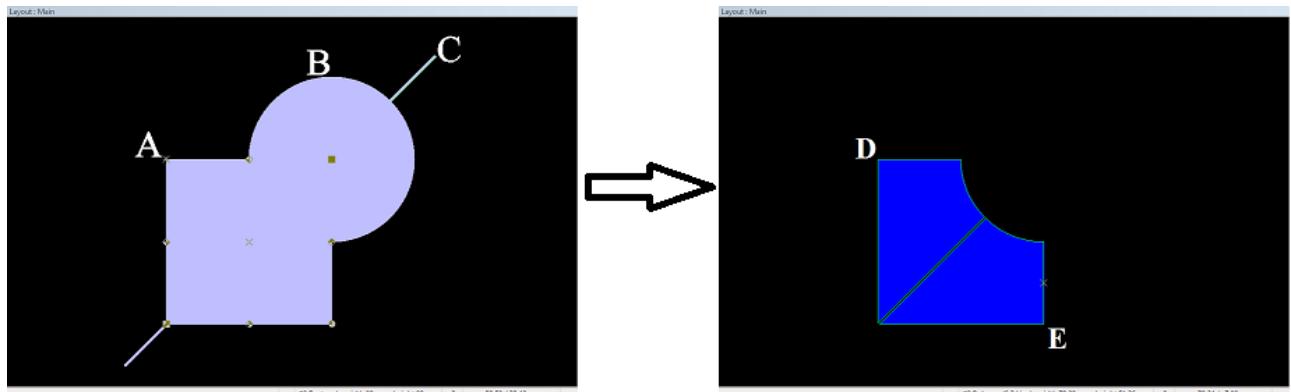


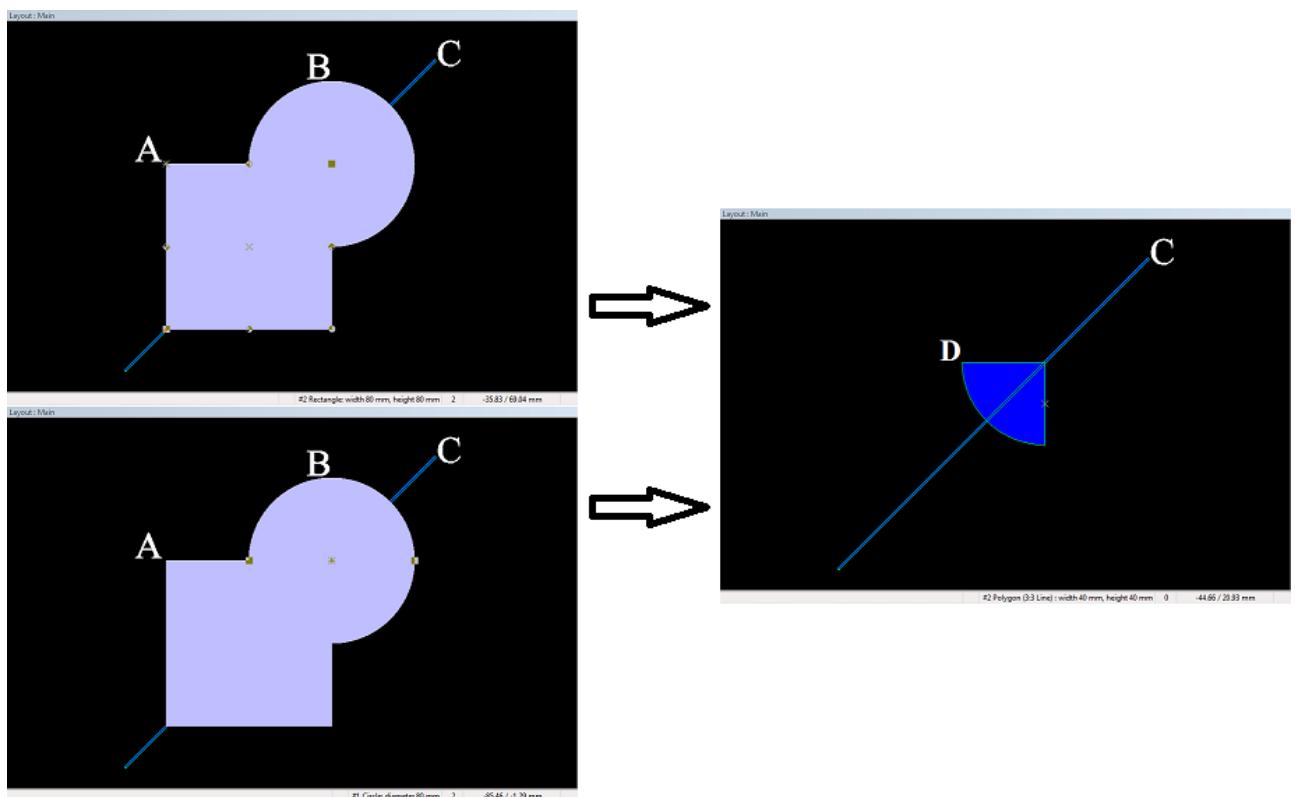
Figure 190. After the three objects A, B, and C are selected, A is the primary selected object (operator);
the difference operation is executed, the result is TWO new polygons D and E

Intersection

All graphical objects whether polygons or paths, are eligible to the *intersection* operation, known from the theory of sets.

The intersection operation is the process of calculating the graphical intersections (overlaps) between the main operator and the secondary operator(s), and representing those intersections as a new polygon(s) on the same layer.

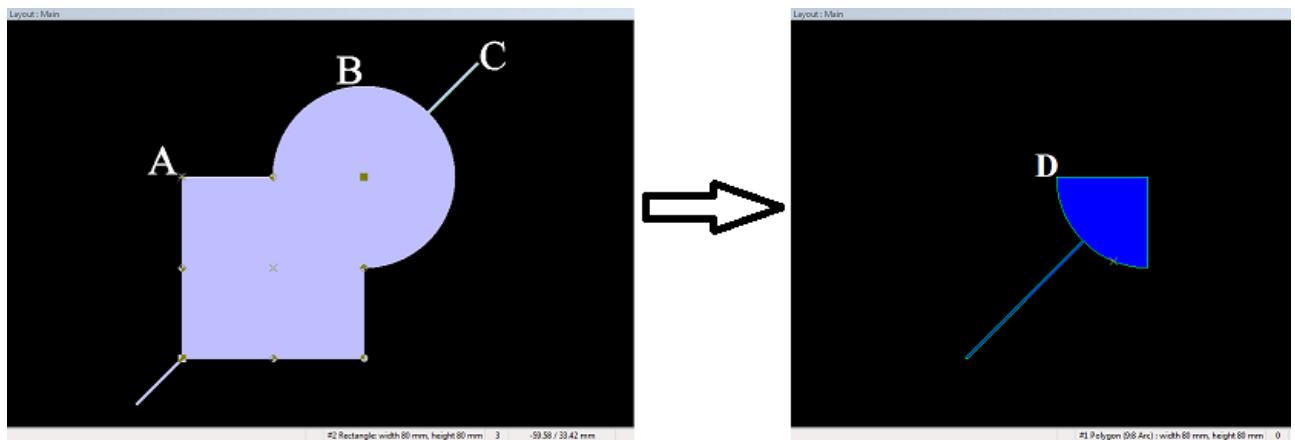
The difference operation is simplified as the word "*and*", e.g.: in the figure below; the operation is: **Graphical objects available in both; A and B.**



**Figure 191. After the two objects A and B are selected, the intersection operation is executed;
the result is a new polygon D**



Note: While using the intersection operation, when working with only two graphical objects; the primary selected object does not alter the result, as seen in the figure above.



**Figure 192. After the three objects A, B, and C are selected, A is the primary selected object (operator);
the intersection operation is executed, the result is a new polygon D**

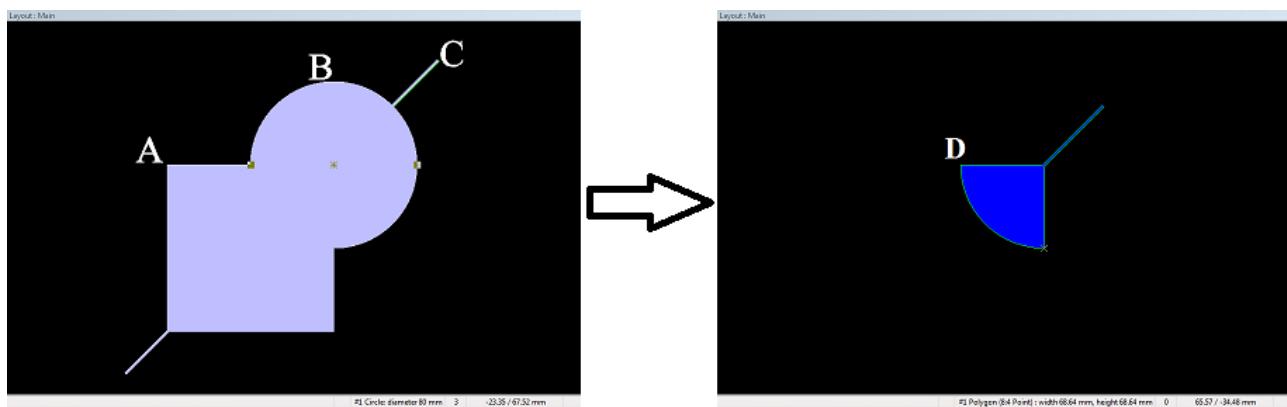


Figure 193. After the three objects A, B, and C are selected, B is the primary selected object (operator); the intersection operation is executed, the result is a new polygon D

XOR

All graphical objects whether polygons or paths, are eligible to the **XOR (Exclusive Disjunction)** operation, known from the theory of sets.

The XOR operation is the process of calculating the graphical intersections (overlaps) between the main operator and the secondary operator(s), and removing those intersections. The result is the exclusive areas of the selected objects.

The XOR operation can be simply explained as the inverse (negative) of intersection operation. See and compare example figures of [intersection](#) and [XOR](#).

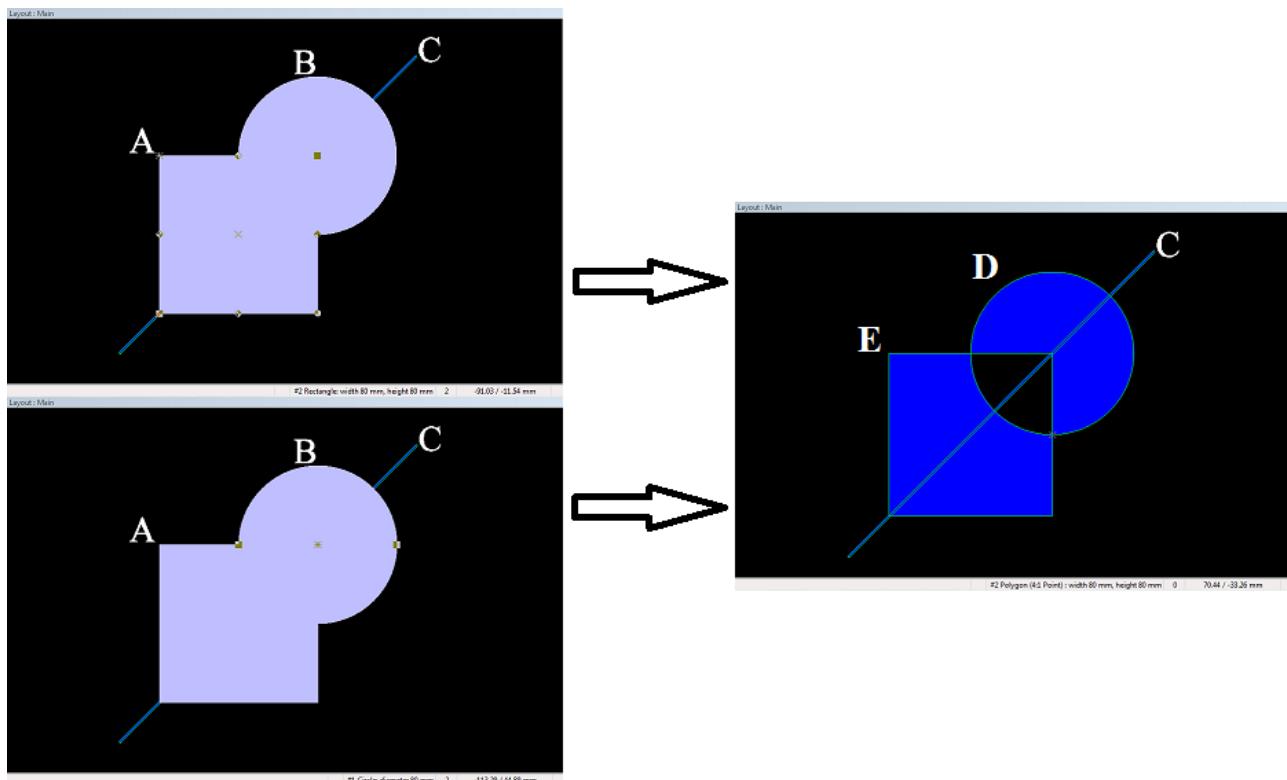


Figure 194. After the two objects A and B are selected, the XOR operation is executed; the result is TWO new polygons D and E



Note: While using the XOR operation, when working with only two graphical objects; the primary selected object does not alter the result, as seen in the figure above.

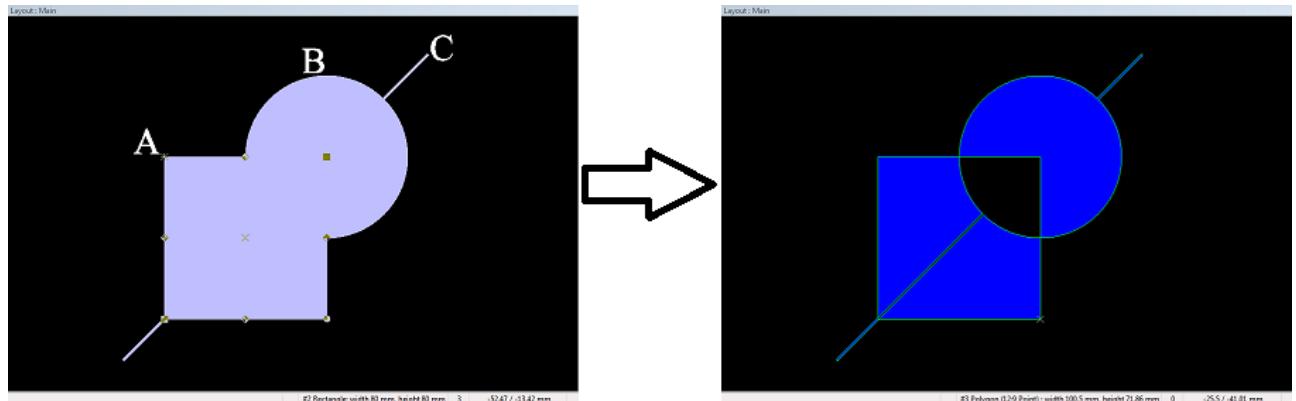


Figure 195. After the three objects A, B, and C are selected, A is the primary selected object (operator);
the XOR operation is executed, the result is a set of new polygons

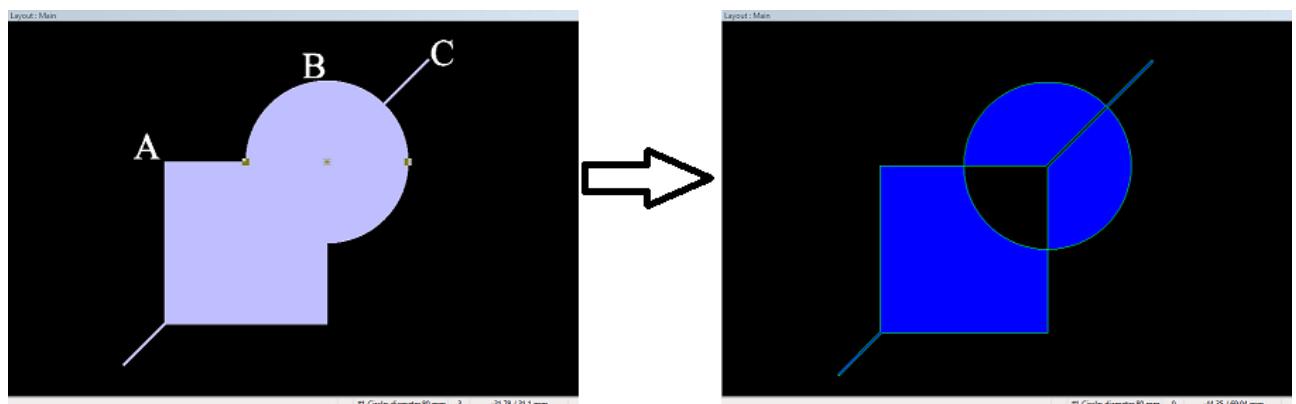


Figure 196. After the three objects A, B, and C are selected, B is the primary selected object (operator);
the XOR operation is executed, the result is a set of new polygons

Paths on Area

This operation trims the paths, leaving only the portions that are present inside an area (polygon).

For this operation to work, at least one polygon and one path must be selected.

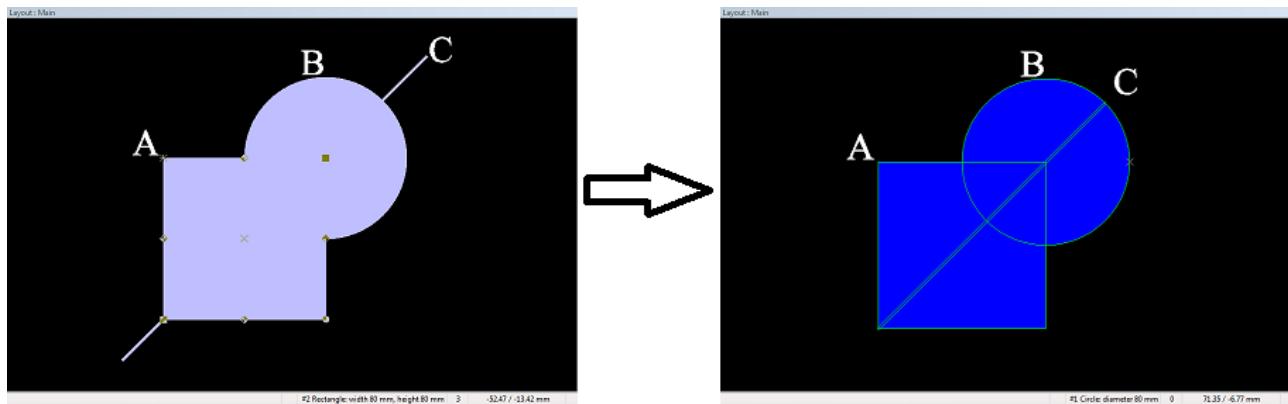


Figure 197. After the three objects are selected, the Paths on Area operation is executed,
the result is trimming of the path C as shown

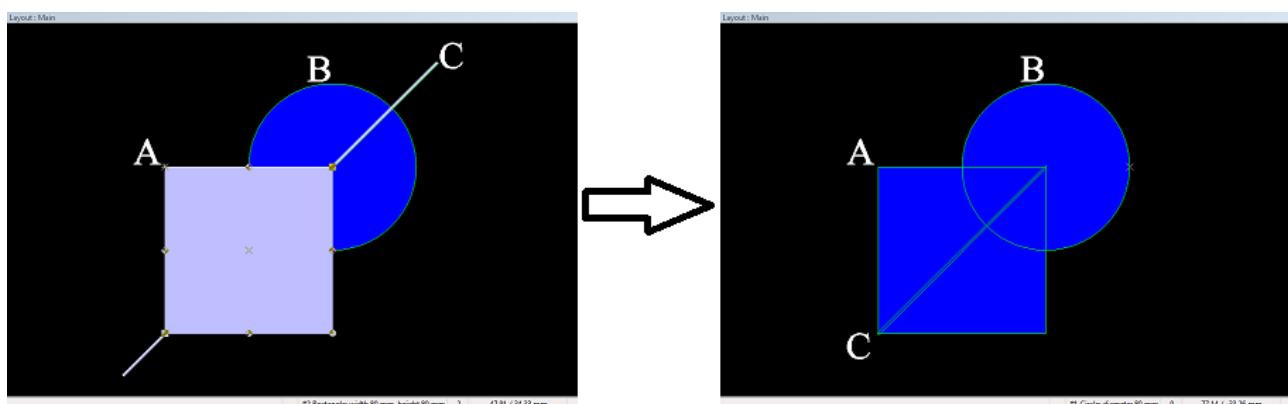


Figure 198. After the two objects A and C are selected, the Paths on Area operation is
executed,
the result is trimming of the path C as shown



Note: The Paths on Area operation is still effective even if there is not a primary selected object.

Paths off Area

This operation trims the paths, leaving only the portions that are present outside of an area (polygon).

For this operation to work, at least one polygon and one path must be selected.



Figure 199. After the three objects are selected, the Paths off Area operation is executed, the result is trimming of the path C as shown

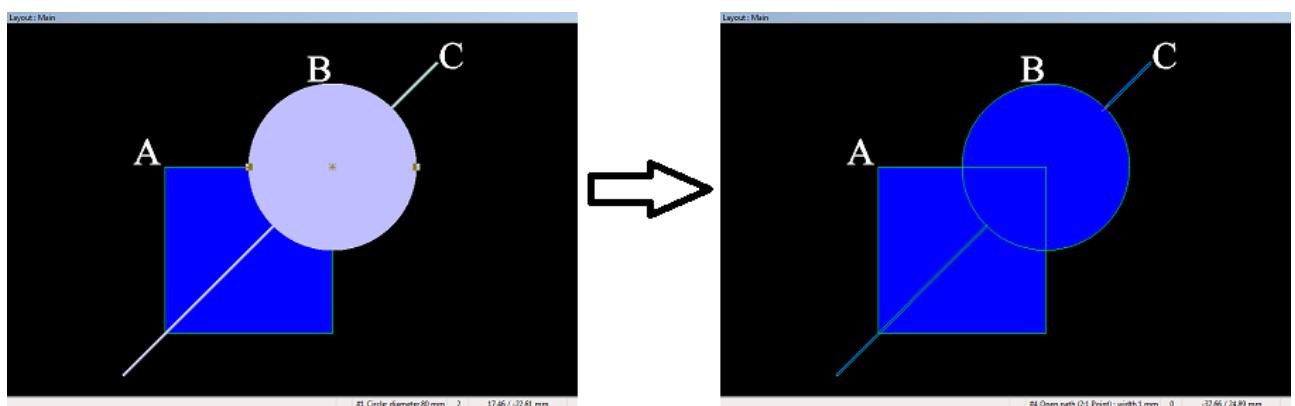


Figure 200. After the two objects B and C are selected, the Paths off Area operation is executed, the result is trimming of the path C as shown



Note: The Paths off Area operation is still effective even if there is not a primary selected object.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.3.4. Size Modification

The *Size modification* allows a user to scale the size of the selected graphical object. In case a few objects are selected, these *modification* are available when objects belong to the same level only.

Size Modification Dialogue

To open the dialogue window, select the **Size Modification** from the **Modify** menu. Most of these operations use *Polygon Package* with **Resolution of Polygon Package** from the **Accuracy** tab of [General settings](#) window.



Note: This function is only available in the *CircuitCAM PRO* variant.

Scaling

These shape modification operations are based on recalculation of coordinates (x,y) for each shape point. A new point coordinates (x,y) are get by scaling offsets between the point coordinates and the shape *Mass/Centre* point coordinates in both X and Y directions. The scaling can be done with the same scale factor in all directions or with the different scale factors for the different directions, according to which type of the scaling is selected:

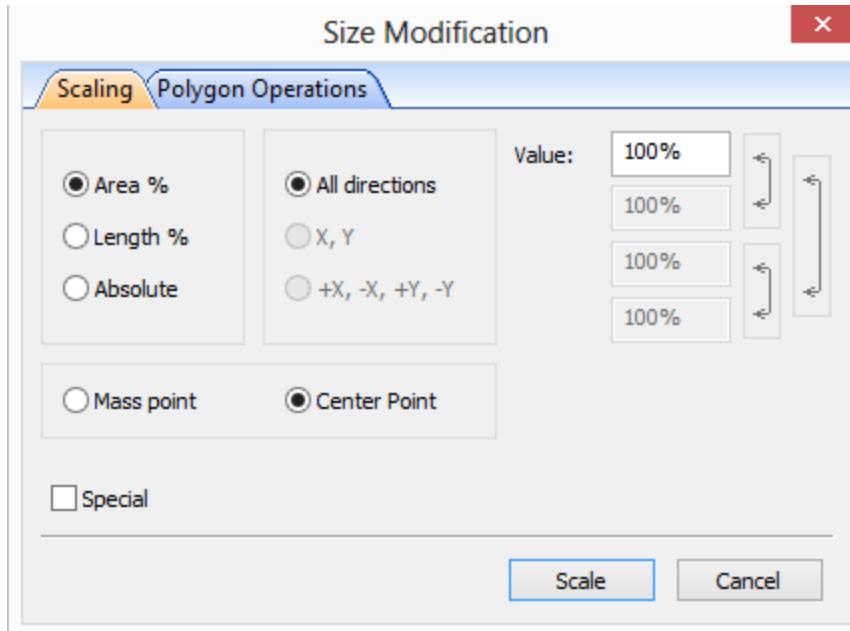


Figure 201. Shape modification dialogue window, Scaling panel

- **Area:** The scaling factor is a quotient of a square root of a shape area by a square root of the shape area increased by a value from the **Value(%)** field (this field appears only after the *area* option is *On*). The same scale factor is in force in all directions (the radio button **All directions** is allowed only).
- **Length:** The scaling factor is calculated directly from a value, defined in the **Value(%)** field. In different directions the same or different scale factors can be used. That depends on which type of scaling is selected (**All directions/x,y/+x,-x,+y,-y**). The picture below shows original pad (green color) and modified pad (red color) (to better emphasize differences the scaled shape is shown in outline mode). Oval pad size - 4 mm by 2 mm; scaling values: **+X(%) = 20, -X(%) = -10, +Y(%) = 10, -Y(%) = -5**:

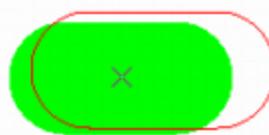


Figure 2. Original pad (green color) and modified pad (red color)

- **Absolute:** The scaling factor is a quotient of a length of a shape bounding box by a length of the shape bounding box increased by an absolute value from the **Value(%)** field. The same scale factor is in force in all directions (the selection button **All directions** is allowed only).
- **All directions:** The scaling is calculated in all directions with the same scale factor.
- **X,Y:** The scaling is calculated in x and y directions with the different scale factors. It can be used in the *Length* scaling mode only.

- **+x,-x,+y,-y :** The scaling is calculated in **-x, +x, -y, +y** directions with the different scale factors. It can be used in the *Length* scaling mode only.



- The arrows allow the user to interchange **x** and **y** values (also **+x** with **-x**, **+y** with **-y** and **+x, -x** with **+y, -y**).
- **Mass point:** The offset values are calculated between shape points and *Mass point*.
- **Centre point:** The offset values are calculated between shape points and *Centre point*.
- **Special:** while flag is *On* a rounded rectangle keeps ratio of its original shape during the scaling.

Polygon Operations

Polygon operations are used to extend, round or truncate a shape. These operations must be performed strictly in the following order:

1. Oversize or Rounding;
2. Stretch;
3. Convert to the rectangle/circle.

Any of these operations can be switched on/off. The **Polygon operations** panel has the following switches:

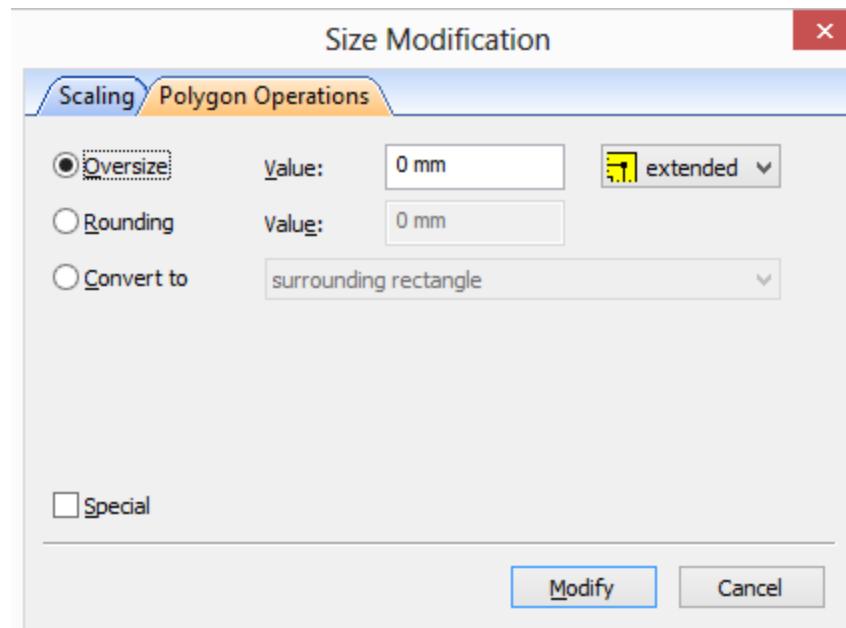


Figure 202. Shape modification dialogue window, Polygon Operations panel

- **Oversize:** oversizes a shape in all directions by a value from the **Value** field. The selection of a type of the over-sizing (*Extend*, *Truncate* or *Round*) is allowed.
- **Rounding:** performs rounding operation in all directions. The operation is executed two times - the shape size is decreased and increased with the same oversize value and the result is the same size with all rounded corners only by a value from the field **Value**. The picture below shows original (green) shapes and shapes (red) after rounding (in outline mode):

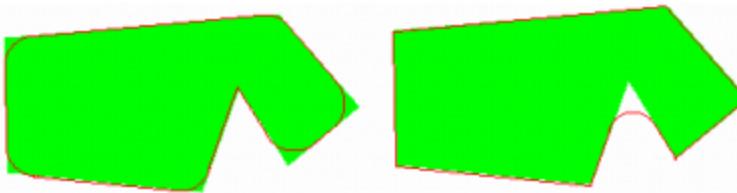


Figure 203. Rounding results: positive value (left) and negative value (right)



Note: A positive rounding value works on convex corners only (left shapes in the picture was done with **Value** = 1.3 mm), negative value – on concave corners only (right shapes in the picture was done with **Value** = -1 mm).

Convert to:

- **surrounding rectangle:** A *Bounding Box* of a shape is calculated and the shape is converted into rectangle of the same size as the *Bounding Box*.
- **surrounding rectangle with same area:** An area of the shape is calculated and the shape is converted into rectangle of the same size as the area. Mass point is assumed as a centre point of the rectangle.
- **circle with same area:** An area of a shape is calculated and the shape is converted into circle with the same area. Mass point is assumed as a centre point of the circle.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.3.5. Properties

Sometimes it is useful to specify main characteristics of the selected graphical object very quickly. For example, a line width and corner types of the path, a circle diameter, a number of lines and arc segments in a polygon. For this purpose the function *Properties* with a respective dialogue box can be used. These dialogue boxes allow modifying some properties too.

The *Properties* function can be called from the **Edit** menu or from the *context menu* (to open the *context menu* click the right mouse button when the mouse is in the graphic window) or by clicking on a particular object in the graphic view. Three different actions are possible:

1. **Document information window appears:** no object was selected before.
2. **Properties dialogue window appears:** a path, polygon, pad or circle was selected.
3. **Area of selected objects is calculated and displayed:** more than one graphical object was selected.

Document Info Window

The **Properties** window of the current document, which is opened in a graphic window, is displayed:

Properties	
Document information	
User	Andrius
Organization	LKSoftWare GmbH
Location	Kaunas
Underlying CAT file	C:\Program Files\CircuitCAM7\New_Templates\default_insulate.cat
Time stamp	2013-04-23 16:54:09
Version	1.2
Comment	CircuitCAM 7.1.2 build 2107

Figure 204. Properties panel, Document information

This information is kept in a *CAM* document. With each *Save/Save As* operation this information is updated with the **User Information** on page [General Settings](#).

In addition, the **Underlying CAT file** field shows the template *CAT* file that was used to create the document.



Note: Some parameters, like [General settings](#), are automatically stored into **Underlying CAT file**. So, if the **Underlying CAT file** does not exist on the current PC, the warning message “*Invalid name of underlying CAT file. General Settings Assignments are not saved*” comes up.

The **Time stamp** field informs user when the underlying template file was created/modified. The **Comment** field shows the *CircuitCAM* version, which was used by saving the *cam* file.

Path Properties

This dialogue window appears in the **Properties** panel if only one open/closed path was selected (in case of closed path the **Object type** field value is switched to *Closed Path*):

Properties	
Selection	
Object type	Open path
Layer	BottomLayer
Configuration	Gerber::ApeTutor
Tool	D10
Line segments	2
Arc segments	0
Path width	0.1 mm
Path length	10.72 mm
Circle diameter	
Local draw attributes	
End type	rounded
Corner type	rounded

Figure 205. Properties panel, Closed path

- **Layer:** Shows the layer of the selected path.
- **Configuration:** A name of the aperture list for the selected path. The list is empty if the **Local Draw Attributes** check box is switched *On*.
- **Tool:** An aperture name from the aperture list, which is shown in **Configuration** field, for the selected path. There is possibility to change the tool - select it from the **Standard** toolbar or the **Tools** panel (*Activate tool* function in context sensitive menu). The list is empty if the **Local Draw Attributes** check box is switched *On*.

- **Line segments/Arc segments:** The information text fields in read-only mode to inform user how many line and arc segments are used in the selected path.
- **Path width:** A width of the line in any unit the selected path is drawn (only positive values are allowed). The list is editable if the **Local Draw Attributes** check box is switched *On*.
- **Path length:** The total length of the path.
- **Circle diameter:** The text field shows a diameter of the circle in the selected closed path. This field is used and is enabled when the selected path is closed path that consists only of a single circle.
- **Corner Type / End Type:** A shape of the line end (End Type) and corners (Corner Type) for paths. If we assume that a round tool draws a path, then the attributes for both are ROUND. The other options EXTEND and TRUNCATE are however also specified by the EDIF standard, but the graphical representation of these shapes is restricted. The lists are enabled, if the **Local Draw Attributes** check box is switched *On*.

Circle Properties

This dialogue window appears in the **Properties** panel if only one circle was selected:

Properties	
Selection	
Object type	Circle
Layer	DrillPlated
Diameter	4 mm
Area	12.566 mm ²
Perimeter	12.566 mm

Figure 206. Properties panel, Circle

- **Layer:** The layer of the selected circle.
- **Diameter:** The circle diameter is displayed in the same unit used for drawing e.g.: **mm or inch**. The list is always enabled.

The next fields show additional circle properties (in read-only mode):

- **Circle area:** Shows the circle area.
- **Circle perimeter:** Shows the circle perimeter.

Rectangle Properties

This dialogue window appears in the **Properties** panel if only one rectangle was selected:

Properties	
Selection	
Object type	Rectangle
Layer	BottomLayer
a	1.042 mm
b	0.5 mm
Area	0.521 mm ²
Perimeter	3.084 mm

Figure 207. Properties panel, Rectangle

- **Layer:** The layer of the selected rectangle.
- **a and b:** The text fields show the sizes of the rectangle.

The next fields show additional properties (in read-only mode):

- **Rectangle area:** Shows the rectangle area.
- **Rectangle perimeter:** Shows the rectangle perimeter.

Polygon Properties

This dialogue window appears in the **Properties** panel if only one polygon was selected:

Properties	
Selection	
Object type	Polygon
Layer	BottomLayer
Line segments	3
Arc segments	1
Perimeter	200,706 mm

Figure 208. Properties panel, Polygon

- **Layer:** The layer of the selected polygon. This list is always enabled.
- **Line segments/Arc segments:** Indicates how many line and arc segments are used in the selected polygon.
- **Polygon area:** Shows the polygon area.
- **Polygon perimeter:** Shows the polygon perimeter.

Flash Properties

This dialogue window appears in the **Properties** panel if only one pad or hole (flash) was selected:

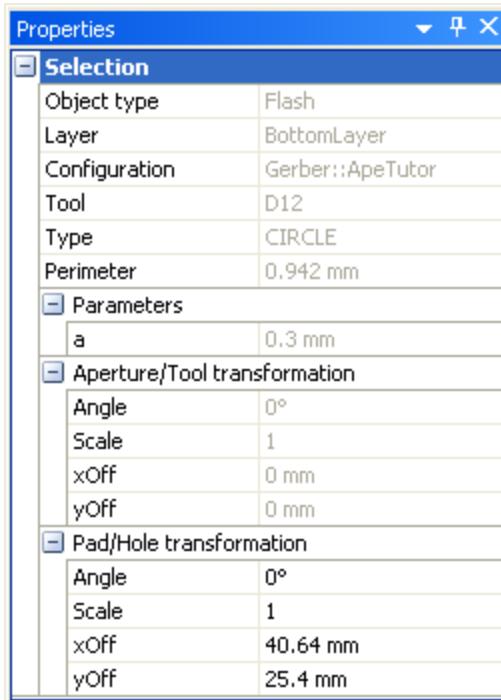


Figure 209. Properties panel, Flash

- **Layer:** The layer of the selected pad/hole.
- **Configuration:** The name of the aperture list for the selected pad/hole.
- **Tool:** An aperture name from the aperture list, which is shown in the **Configuration** field for the selected pad/hole. There is a possibility to change the tool - select it from the **Standard** toolbar or the **Tools** panel (*Activate tool* function in context sensitive menu). The list is always enabled.
- **Type:** The text field in read-only mode shows a type of the tool (for more information on tool types, see the **Flash Attributes** sheet of [Tool properties](#)).
- **Parameters:** Shows dimensions of the tool in any unit (for more information on tool dimensions, see the **Flash Attributes** sheet of [Tool properties](#)).
- **Aperture/Tool transform:** Four text fields in read-only mode show the location of the point (xOff, yOff), scale factor and rotation angle. The dimensions of the point are relevant to the tool dimensions, presented in the **Parameters** field.
- **Pad/Hole transform:** The text fields show where and what rotation the selected tool is placed in the graphical window. Values of these fields can be modified.

Selection Properties

This dialogue window appears in the **Properties** panel if more than one graphical object is selected:



Figure 210. Properties panel, selected 43 items

- **Number of items:** The number of selected items.
- **Size:** The size of the rectangle for the selection area.
- **Bounding area:** The bounding area of the selection.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.3.6. Selection of Graphic Objects

To change graphical information inside the graphic window (move, delete etc.), the elements to be amended must be selected by clicking them with the mouse. One or more objects can be selected at the same time; this is called a selection group, which contains the selected objects.

The selected objects are visually highlighted by a different color, for the highlight color see [Layers View](#).

In most cases one of the selected objects is the primary object, thus indicated by markers on its construction points. see [Manipulation of Graphical Objects](#).

The primary selected object is the main operator in all editing and moving operations; such as [Boolean Operations](#).

Adding or removing a selection from the group will changes the primary selected object.

CircuitCAM has a number of functions and switches to determine what kinds of graphical objects may or may not be selected. These functions are available by the [Select menu](#) or on the [Select toolbar](#):



Note: Every selected element can be deselected with the **ESC** key or **Cancel** from the context sensitive menu.

The summary information of the selection appears in the [Properties](#) window: the number of items, the size and the bounding area.

Single Object

A single graphic object is selected by a single mouse click. Objects, which are closest to the cursor position, are selected first. If a number of objects are superimposed at the same place, one object after another can be selected by repeated clicks.

There are three selection modes on a single object:

- *select single object (new selection mode):* selects a new object by left-click on an applicable object. All objects selected before will be deselected. To set the **new** mode use one of the following:
 - a. clicking on the
 - b. *or* set the *New* function from the *Select* menu;
 - c. *or* *Cancel* operation.
- *add single object (add selection mode):* adds a new object to the selected group by left-click on the object to be add. To set the **add** mode use one of the following:
 - a. clicking on the
 - b. *or* set the *Plus* mode from the *Select* menu;

- c. **or** holding down **Ctrl** key while clicking with the mouse.
- *remove single object* (**remove** selection mode): removes an object from the selected group by left-click on the object to be remove. To set the **remove** mode use one of the following:
 - a. clicking on the  item in the **Select** toolbar;
 - b. **or** set the *Minus* function from the **Select** menu;
 - c. **or** holding down **Shift** key while clicking with the mouse.

Area

All objects inside a rectangular area are selected. The rectangular area is defined by two opposite corner points with the mouse as in the following procedure;

Press Set the first corner point by pressing and holding the mouse button.
Move Move to the next corner point with the button held down
Release Set the second corner point by releasing the mouse button.

During these operations, the rectangular area is displayed for visual checking. There are three selection modes on an area:

- *select single area* (**new** selection mode): only the objects from the rectangle area are selected (all previous selections will be cancelled). To set the **new** mode use one of the following:
 - a. clicking on the  item in the **Select** toolbar;
 - b. **or** set the *New* function from the *Select* menu;
 - c. **or** *Cancel* operation.
- *add new area* (**add** selection mode): add all objects of a new rectangle area to the selected group. To set the **add** mode use one of the following:
 - a. clicking on  item in **Select** toolbar;
 - b. **or** set the *Plus* mode from the **Select** menu;
 - c. **or** holding down **Ctrl** key while clicking with the mouse.
- *remove area* (**remove** selection mode): remove all objects of a new rectangle area from the selected group. To set the **remove** mode use one of the following:
 - a. clicking on the  item in the **Select** toolbar;
 - b. **or** set the *Minus* function from the **Select** menu;
 - c. **or** holding down **Shift** key while clicking with the mouse.



Note: All selection functions are altered by changes in the "[Selection Type](#)" in the [Select Menu](#).

Sequence

This is a special selection mode, which allows the user to select objects in the order of their appearance (in the order these objects were drawn) or otherwise. For more details, see [Sequence](#).

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.3.7. Context Menu "Right Mouse Button"

While using the right mouse button in the client area of a **Graphic** view, the following popup menu (mouse *context sensitive menu*) will appear:

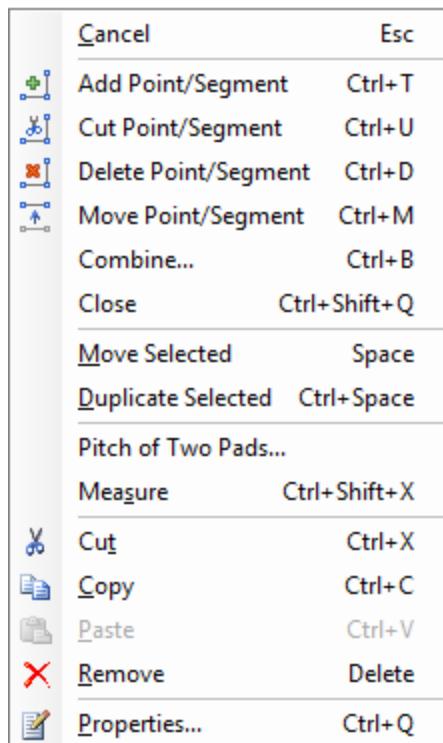


Figure 211. Right mouse button menu (context sensitive menu)

	Cancel	Finishes/Interrupts the current action. E.g.: deselects selected objects; finishes/interrupts any <i>Insert</i> operation; finishes <i>Measure</i> operation; interrupts <i>Copy/Duplicate</i> operations and so on.
	Add Point/Segment	The menu item appears only if selection is made in the Graphic view. See Add Point/Segment .
	Cut Point/Segment	The menu item appears only if selection is made in the Graphic view. See Cut Point/Segment .
	Delete Point/Segment	The menu item appears only if selection is made in the Graphic view. See Delete Point/Segment .
	Move Point/Segment	The menu item appears only if selection is made in the Graphic view. See Move Point/Segment .
	Create or Move Arc	The menu item appears only if selection is made in the Graphic view. See Create or Move Arc .
	Combine...	The menu item appears only if selection is made in the Graphic view. See Combine....
	Close	The menu item appears only if selection is made in the Graphic view. See Close .

	Move Selected	See Move selected .
	Duplicate Selected	See Duplicate Selected .
	Pitch of Two Pads	Calculate a distance between centre points of two selected areas (pads) only.
	Measure	See Measure .
	Cut	Move the selected objects to the standard clipboard.
	Copy	Copy the selected objects to the standard clipboard.
	Paste	Extract the objects from the clipboard.
	Remove	Delete all selected objects.
	Properties	Activate Properties panel, where the characteristics of the selected object are specified.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.3.8. Editing operations

Create Gap

This command adds a gap at the active marker point. A width of the gap can be specified from the **Breakout Tab** field in the **Settings -> General settings -> Diverse**, page and can be set to a new value.

This feature only works on paths (*open or closed*), in case of a polygon, first it must be converted to a closed outline path. [See more](#).

This function can be applied at:

1. **Start or end support point:** The selected point is assumed as the gap's centre point (*the path is shortened by the specified Gap Value/2*).
2. **Central support point:** The path is divided into two paths at this point, and this point is assumed as the gap's centre point.
3. **Segment:** The selected segment is divided into two parts at the selected point, and this point is assumed as the gap's centre point.

The **Create Gap** function is accessible by any of the following means:

- Going to the menu; **NC/Tool path ---> Create Gap**.
- Keyboard shortcut; **Ctrl + G**

Reverse

This command reverses the direction of the selected object.

The **Reverse** function is accessible by any of the following means:

- Going to the menu; **NC/Tool path ---> Reverse**.
- Keyboard shortcut; **Ctrl + R**.

Rotation and Mirroring

There are **two** different modes to rotate selected objects:

- The ***relative*** mode is available for any type of objects, where the resulting rotation/mirroring of any object depends on the position of its current anchor point. ***Relative*** mode rotates and mirrors multiple selected objects around the anchor point of the whole selection group.
- The ***absolute*** mode (*also called flash rotation*) is possible only for flashes/pads. ***Absolute*** mode rotates and mirrors selected objects around its centre point (0,0), thus, all selected flashes will have the same rotation/mirroring (location of flashes is not changed).

The rotation/mirroring functions are accessible by any of the following means:

- Going to the menu; **Edit ---> Rotate Relative**.
- Going to the menu; **Edit ---> Set Flash Rotation Value**.

- Accessing the functions from the **Relative Rotations** toolbar
or
- 
- the **Flash Rotations** Toolbar
- 

In the **Rotate** dialogue window, the function **Value** allows the user to define the direction of the rotation and enter the desired angle.

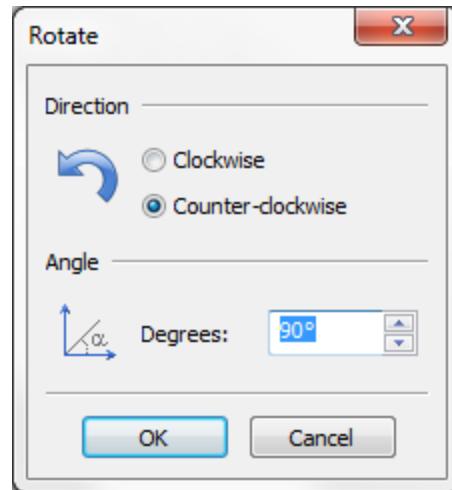


Figure 212. Rotate dialogue window

Other functions execute fixed actions.

This example object illustrates the effects of the various functions:

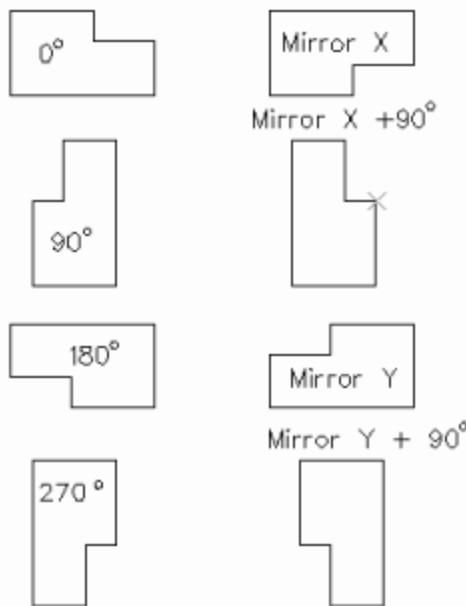


Figure 213. Rotate function effects

Value ...	Mirror X
90°	Mirror X+90°
180°	Mirror Y
270°	Mirror Y+90°

The given point of the original (x, y) is changed in the following way:

- **90**(-y, +x)
- **180**(-x, -y)
- **270**(+y, -x)
- **MX**(+x, -y)
- **MXR90**(+y, +x)
- **MYz**(-x, +y)
- **MYR90z**(-y, -x)

User can choose any case for a fixed rotation from the following list, the first line **Value....** was already described above:

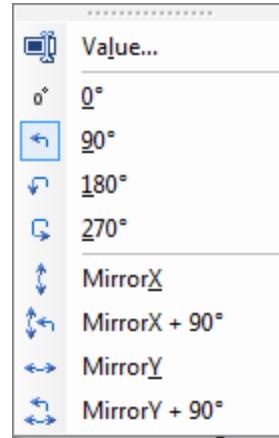


Figure 214. Edit -> Set Flash Rotation Value menu content

The framed menu icon indicates a current rotation for the selected object, and is used only in the *absolute* mode.

Set Anchor Point

Initially the anchor point is set to the centre of the coordinate system (0/0). The *Set Anchor Point* allows to set an anchor point to any other coordinates.

After the selection of a graphic object, *CircuitCAM* displays markers on the screen, which do not overlap with their neighbours. One of these markers is selected as **Anchor Point**. The anchor point is displayed with X-Y values, which are shown in the right area of the [status bar](#).

The **Set Anchor point** function allows changing the anchor point to the coordinates (X;Y), which are typed in the text field of the **Set Anchor** dialogue window.

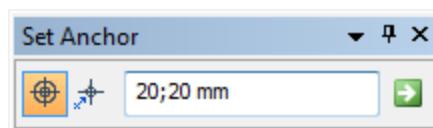


Figure 215. Set Anchor Point panel

To set the **Anchor Point** at the current mouse position press **Ctrl + Shift** and click **left mouse button** any place on the graphic view. The particular X and Y coordinates appear in the **Set Anchor** window.

The anchor point is used as a reference point for move/duplicate group; used as a starting point for path/polygon manipulating operations for setting a new zero point; used as a start point for measuring the component sizes and others operations.

An anchor point can be set either in *absolute* or in *relative* mode. A user can switch the current mode to another temporarily by entering '!' in front of the "X;Y" in the text field.

The **Set Anchor Point** function is accessible by any of the following means:

- Going to the menu; **Edit ---> Set Anchor Point**.
- Keyboard shortcut; **Ctrl + Shift + A**.

Set Cursor

This function sets a new cursor position, and simulates a *mouse left click* on that position.

The mouse cursor coordinates are taken from corresponding dialogue window. The **Set Cursor** function can be used in drawing/manipulating graphic objects.

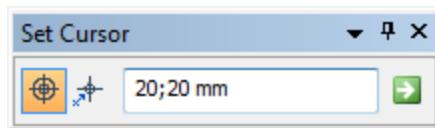


Figure 216. Set Cursor dialogue window

The **Set** button has the same meaning as **OK**, but keeps dialogue box open for the next manipulations. E.g.: it is useful while adding a new path. Furthermore, a new cursor position can be set either in \oplus *absolute* or in \triangleleft *relative* mode. The user can switch the current mode to another temporarily by entering '!' in front of the 'x;y' in the text field.

The **Set Cursor** function is accessible by any of the following means:

- Going to the menu; **Edit ---> Set Cursor**.
- Keyboard shortcut; **Ctrl + Shift + C**.

Swap

The *Swap* function changes the order of objects. It works on the selected object(s) inside the same layer and between objects. The current position of the selected object is shown on the left field of the *Status Bar*. For example, the current fragment shows the selected flash-circle has an order number 24 among all existing flashes in the layer that the selected flash-circle belongs to.



Figure 217. Status bar with flash number

Four different swap operations are available:

- Move to the next position in the group (**Shift + J**);
- Move to the previous position in the group (**Shift + I**);
- Move to the first position in the group (**Shift + F**);
- Move to the last position in the group (**Shift + L**).

The **Swap** function is accessible by any of the following means:

- Going to the menu; **NC/Tool path---> Swap**.
- Using the previously explained keyboard shortcuts.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.3.9. Inserting Operations

CircuitCAM allows various objects (polygons/paths/pads...etc) to be inserted to the workspace, all inserting operations are accessible via the menu; **Insert**.

General Hints:

- When inserting a new object, first you must select the desired layer, otherwise the currently active layer is used.

- If one or several objects (from the same layer) are already selected, when inserting a new object, the original object(s) will be deselected, and the new object will be added to the current layer.
- If several objects from different layers are selected, inserting operations are not possible (objects names and icons are greyed out).

Arc

Paths and outlines of polygons, consist of line and arc segments. A line segment is always drawn first. The start and end points of the segment are fixed. Then the line segment can be converted into an arc segment by using one of the two following methods:

1. **Centre Point:** The circle sector is determined by defining the *Centre point* of the circle.
2. **Point on Arc:** An *additional point on the arc* determines the circle sector.

Drawing procedure:

- Start by drawing a path or a polygon.
- Draw a line segment, which is changed into an arc segment. Any line segment can be used.
- Activate the **Centre Point Arc** or **Point on Arc** function in the **Insert** menu or click on the **Centre Point Arc** or **Point on Arc** in the **Insert** toolbar.
- Depending on the selected arc method, one of the following actions is carried out:
 1. **Centre Point** method: The Centre point of the arc is specified;

or

 2. **Point on Arc** method: A point is specified on the arc;
 - The conversion of the line segment into an arc segment is interrupted by **Esc** or right mouse button click.
 - The next segment then can be drawn.

Circle

CircuitCAM supports three different methods to draw a circle.

Centre Method

The *Centre point* and a *point on the edge* of the circle are entered. The circle is filled.

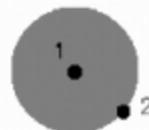


Figure 218. Filled Circle (1- centre point, 2- point on edge)

Drawing procedure:

- Activate the **Dynamic Radius Circle** function in the **Insert** menu or click on the **Dynamic Radius Circle** in the **Insert** toolbar.
- Set the *Centre point* of the circle with a mouse.
- The circle stretches between the Centre point and the current mouse position.
- Set a point on the perimeter of the circle with the mouse; the circle is complete.
- Set the *Centre point* of the next circle or end the circle function with **Esc** or the right mouse button click.

Centre + Diameter Method

The diameter must be entered in the **Diameter** field of the dialogue window; only the Centre point is specified. The circle is filled.

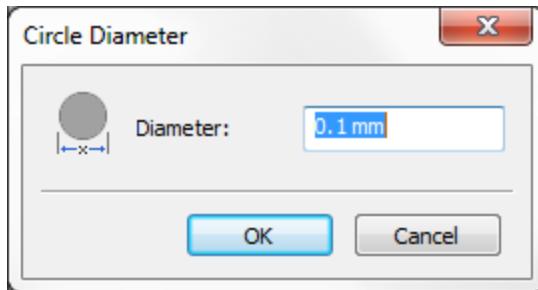


Figure 219. Circle Diameter dialogue window

Drawing procedure:

- Activate the **Fixed Diameter Circle** function in the **Insert** menu or click on the **Fixed Diameter Circle** in the **Insert** toolbar.
- Type the diameter in any unit in the **Diameter** text field.
- Set the *Centre point* of the circle with a mouse. The circle is complete.
- More circles can be defined by setting more *Centre points*. Quit the function with **Esc** or the right mouse button click.

Circle in Closed Path method

The method works like **Centre + Diameter** method **but** the circular ring (donut) is assumed as a closed path. The circle is not filled.

Line Drawing Modes

While drawing lines and an arc segments, there is often a need to align the segments exactly vertically. This can be done using the line drawing methods. The following methods can be used:

1. **90 angle:** The effective mouse position changes only vertically/horizontally.
2. **45 angle:** The effective mouse position changes only at 45 degrees angles and increments.

3. **Any angle:** The effective mouse position agrees with the displayed mouse position.
Any angle can be drawn.

Open Path

The function inserts a new path that consists of one or more lines or arc segments. The start and end points of these segments are called the line *support points*. The sequence of the line support point is specified when drawing a new path. It determines the subsequent output direction when generating production data.

The following layer attributes determine the shape of a path:

- PathWidth: Defines the line width.
- EndType: Defines the shape of the first and last line support points.
- CornerType Determines the shape of the inner line support points.

The types ROUND, EXTEND and TRUNCATE are available for the corner type and the end type.

Drawing procedure:

- Select an aperture/tool, otherwise the path with zero line width and local draw attributes will be drawn.
- Activate the **Open Path** function in the **Insert** menu or click on the **Insert Open Path** in the **Insert** toolbar.
- Set the first line support point with the mouse.
- The line stretches between the first line support point and the current mouse position.
- Set the second line support point with the mouse.
- End the path with **Esc** or the right mouse button click. The next path now can be drawn.
- Another **Esc** or the right mouse button click ends the line function.

Closed Path

The *Closed Path* function is very similar to *Open Path* function except the last path point will be joined to the start point. In the drawing procedure the *Open Path* function must be changed to the *Closed Path* function.

Pad

Follow this routine to insert a new pad (flash):

- Select an aperture/tool, otherwise the drawing is disabled.
- Activate the **Pad** function in the **Insert** menu or click on the **Pad** in the **Insert** toolbar.
- Set the *Centre point* of the flash with the mouse.

- The next flash now can be drawn.
- **Esc** or the right mouse button click ends the drawing function.

Polygon with Cutouts

A *polygonCutOut* is a filled area with holes, defined by outer outlines and one or more inner outlines. The outlines consist of two, three or more lines or an arc segments. An outline is not allowed to intersect itself (like an “8” shape) or with other outlines. The start and end points of these segments are called *support points*. The support point sequence is specified when drawing a new polygon.

The function inserts a new *polygonCutOut*. The *polygonCutOut* consists of at least two figures. The first figure is always an outer figure (boundary), all others figures are inner figures (cut-outs).

A boundary shape may be a polygon, a circle and a rectangle. For the inner shape additionally a flash and a path can be used. The figure below is a *polygonCutOut* with a rectangular outer and a circular inner shape.



Figure 220. *polygonCutOut* with rectangular outer and circular inner shape

The type of the object, and the type of the cutout are both represented in the properties window and the status bar;

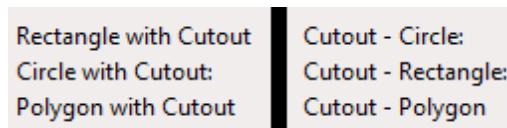


Figure 221. Types of cutouts identified in the status bar

To denest the selected *CutOut* from *polygonCutOut* and to destroy the selected *polygonCutOut* at all (shapes are not removed but they become simple shapes as a polygon, a circle, a rectangle and so on) use corresponding **Denest cutOut** and **Denest polygonCutOut** functions from the **Modify** menu.

- Select the items (polygon, circle, rectangle, flash), which should be a cut out, in the **Graphic** view.

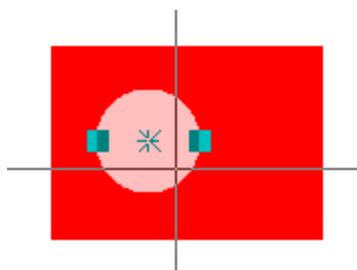
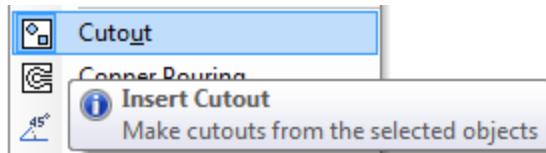
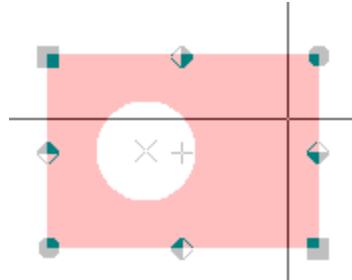


Figure 222. Selected circle

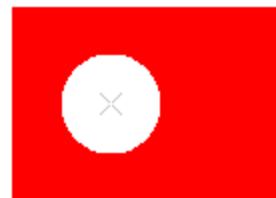
- Select in the main menu **Insert -> Cutout** menu item. The *Select outline for cutouts* mode is turned *On* (see message line in the status bar). The special mouse cursor  appears on the screen.

**Figure 223. Cutout menu item**

- Select a figure, a candidate to boundary for a new polygonCutOut.

**Figure 224. Selected a boundary polygon for polygonCutOut**

- Press **Esc** button to switch *Off* selection mode to see the result of the *Cutout* operation.

**Figure 225. Result of Cutout operation**

Note: This function is only available in the *CircuitCAM* variants **STANDARD** and **PRO**.

Polygon

A polygon is a filled area defined by an outline. The outline consists of two, three or more lines and an arc segments. The outline is not allowed to intersect itself (like an “8” shape), because the software cannot decide what is inside and what is outside. The start and end points of these segments are called support points. The support point sequence is specified when drawing a new polygon.

Drawing procedure:

- Activate the **Polygon** function in the **Insert** menu or click on the **Insert Polygon** in the **Insert** toolbar.
- Set the first support point with a mouse.
- The line stretches between the first support point and the current mouse position.
- Set the second line support point with the mouse.
- Set the third and all other support points.
- End the polygon with **Esc** or the right mouse button click. The next polygon now can be drawn.
- Another **Esc** or the right mouse button click ends the polygon function.

Rectangle

Two opposite points describe a rectangle. The sides of the rectangle are parallel to the x and y axis.

Drawing procedure:

- Activate the **Rectangle** function in the **Insert** menu or click on the **Insert Rectangle** in the **Insert** toolbar.
- Set the first corner point in the graphics area by clicking with the mouse.
- The rectangle stretches between the first corner point and the current mouse position.
- Set the opposite corner point with the mouse. The rectangle is now drawn.
- Now the next rectangle can be started, or the function can be ended with **Esc** or the right mouse button click.

Rubout Area

The function inserts a new *rubout area*. This *area* is a zone where the isolation channels around conductor tracks and pads are milled, and also all residual copper between the conductor tracks and pads is removed from the circuit board.

A shape for the **rubout area** (polygon, rectangle, circle and etc.) can be selected by using the corresponding functions from the **Insert** menu or **Insert** toolbar. The rectangular shape is set by default.

Top layer: Creates a *rubout area* on the top side of the circuit board.

Bottom layer: Creates a *rubout area* on the bottom side of the circuit board.

All layers: Creates a *rubout area* on both sides.

You can see more inserting operations in the [Insert](#) menu.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.3.10. Manipulation Operations

CircuitCAM supports some manipulation operations on paths and polygons (see [Manipulation of graphic objects](#)). They affect individual support points or segments, the support point or segment to be modified is identified by the selection marker.

Any manipulation operation can be carried out by one of the following methods:

1. Standard method:

- Select the standard operation from the **Path/Polygon** operations list.
- Select a path or a polygon.
- Move the cursor to one of the markers so that the cursor's shape changes.
- Click the left mouse button to execute the standard operation.
With some operations, the mouse button simply needs to be clicked (cut and delete).
While with others the cursor is moved with the button held down (add/append point, move point, create or move arc).

2. **Temporary method - Temp x** (method is very similar to standard method):
 - Move the cursor to one of the markers so that the cursor's shape changes.
 - Select temporary operation by pressing and holding down the corresponding keyboard shortcut.
 - Click the left mouse button to execute the temporary operation.



Note: Releasing the keyboard shortcut key disables the temporary operation.

3. **Direct method:**

- Activate the support point or segment marker to be manipulated.
- Activate the direct manipulation operation from the **Modify --> Curve** menu.
- With some operations these steps are sufficient, while with others a new position must be specified by the mouse.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.3.11. Text and Barcodes

3.3.12. Text

CircuitCAM supports creation of text using pre-installed fonts on the windows system (Please note that *CircuitCAM* only supports TTF and TTC fonts), or special fonts that are part of *CircuitCAM*. In contrast to TTF and TTC fonts, the special *CircuitCAM* fonts have the advantage of a clear centre path, so that they can be manufactured efficiently by a milling/laser tool of the chosen width:

- *CircuitCAM* default. LKSoftWare
- ISO3098. LKSoftWare
- *CircuitCAM* stencil. LKSoftWare, the stencil font is used when creating SMT stencils, the advantage of this font is that it creates gaps between cuts, thus maintaining the stencil free of large cut holes, as seen in the preview image with the letter "o".

The stencil font is designed in such a way that when the material is cut through by laser or milling tool, no material will fall out. This is ensured by creating texts as little gaps. See in the example the letter "o" is created using four small gaps. When using the other fonts, the material inside the letter "o" will fall out.

When creating text in *CircuitCAM*, that text is converted into basic graphic objects on the specified layer. Because of this the text can not be edited afterwards.

Follow this routine to insert a new text:

- Activate the **Text** function in the **Insert** menu or click on the **Text** in the **Insert** toolbar and fill fields of text dialogue window.

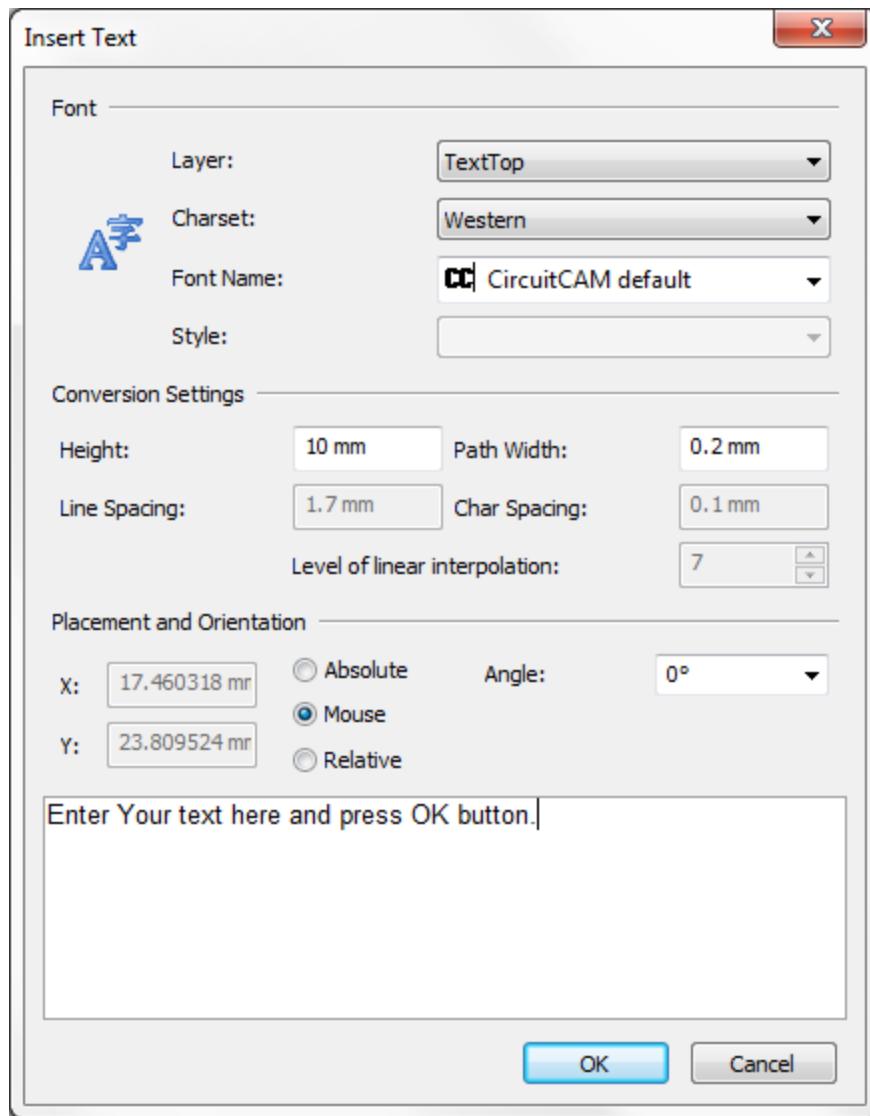


Figure 226. Insert text dialogue window

- Set a destination layer in the **Layer** field.
- Set a font. Select a font family from the **Font Name** combo box. The font is a complete set of characters, letters, and symbols of a particular typeface design. In addition, from the **Style** combo box you can choose a type style (for example, *Regular*, *Bold*, *Italic*, *Bold Italic*). The range of available type styles depends on the selected font. Select a character set from the **Charset** combo box (for example, *Arial* font has *Western*, *Baltic*, *Turkish*, *Russian* and other character sets).



Note: Some fonts in *Windows 2000* cannot display all *Unicode* characters. If you see that any characters are missing in your text, you can change the font to one that includes these characters. Generally, the "Microsoft Sans Serif" font is a good choice for *Unicode* characters.

- Set a text height and the path width in the **Height** and **Path width** fields.
- Set a line distance in the **Line Spacing** field. The line spacing specifies the spacing between baselines.

- Set a character distance in the **Char Spacing** field. The character distance controls the spacing between two characters. A positive character distance value moves characters apart, any negative value is invalid.
- Set an interpolation level in the **Level of linear interpolation**. The level of linear interpolation controls the quality of drawing glyphs. To reach best quality, the level of linear interpolation must be set to 10.
- Set a text insert mode in the **Mode** group. Possible modes are *Mouse*, *Absolute* and *Relative*.
- Set a rotation in the **Angle** field.
- Press **OK**. In case of the *Mouse* mode, you need to set the start point of the text in the graphic window with the mouse and do left-click to fix the text.

3.3.13. Barcode

The create text function can also be used to create barcodes. In this case the created graphics does not represent the text itself, but instead consists of a sequence of rectangles which make up the barcode.

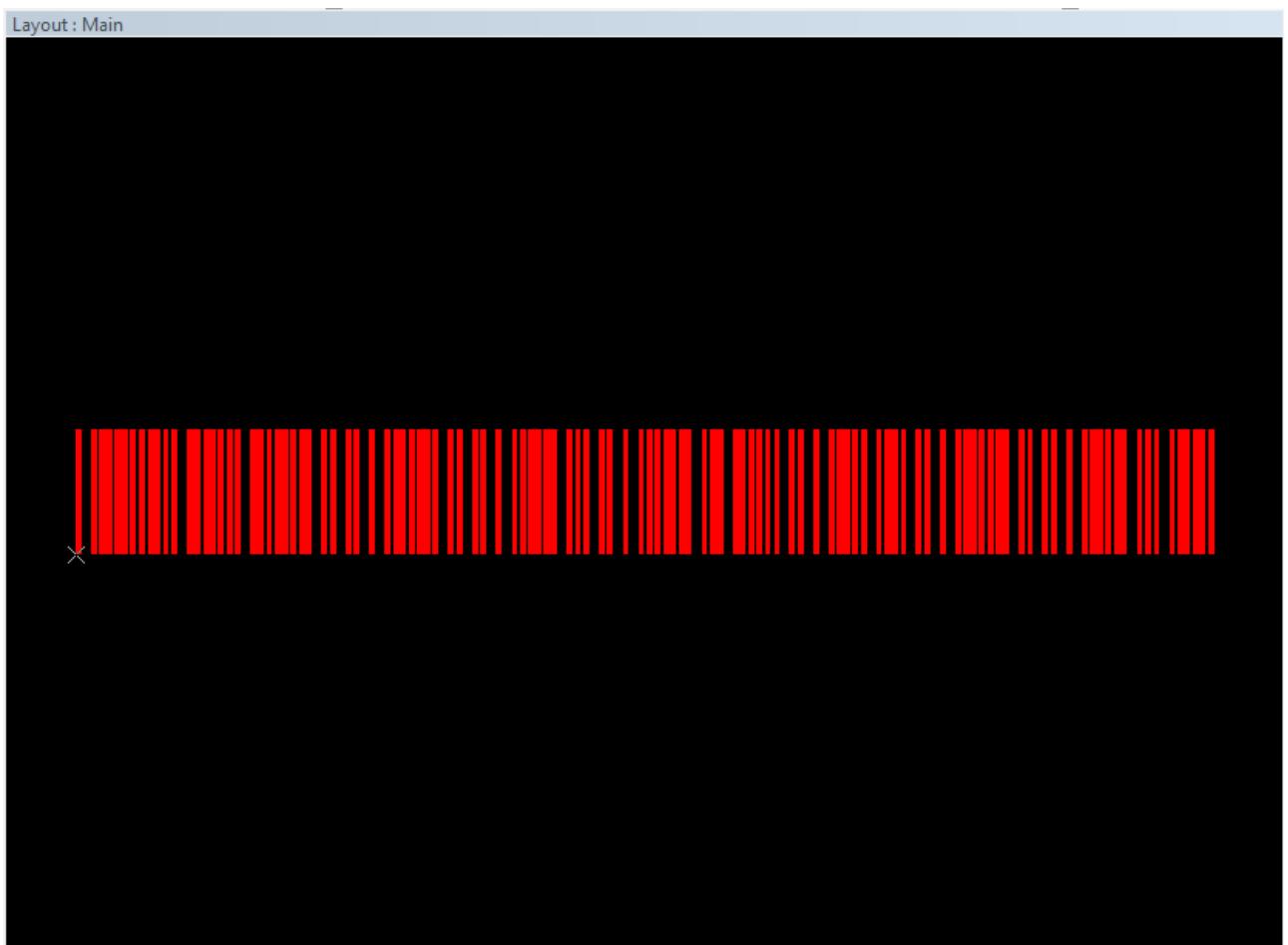


Figure 227. Barcode Sample

To insert a barcode, special additional fonts to support barcode must be installed. The procedure is the same as inserting text but the font is changed to a barcode supported font.

Barcode fonts can be obtained from several places over the internet, either paid or for free. Barcode fonts are usually available in the TTF ([True Type Font](#)) format, or TTC ([True Type](#)

Collection) format. To install new TTF or TTC font into a Windows system, a right mouse click is applied on the file, then install is selected from the context menu.

To read more about TTF and TTC fonts, please refer to: "http://en.wikipedia.org/wiki/TrueType" and "http://en.wikipedia.org/wiki/TrueType_Collection#TrueType_Collection"



Figure 228. Installing a new font

Listed below are some of the fonts we tested at DCT;

- <http://www.barcodesinc.com/free-barcode-font/>
- <http://www.free-barcode-font.com/>
- <http://www.ttfont.com/free/barcode-fonts/>
- <http://www.windows8downloads.com/win8-bar-code-3-of-9-wbccwyxh/>
- <http://www.windows8downloads.com/win8-bar-code-2-of-5-interleaved-cragiwnl/>
- <http://www.windows8downloads.com/win8-ean-bar-codes-coxrgxyk/>
- <http://www.windows8downloads.com/win8-connectcode-datamatrix-barcode-fonts-gcy-mauwa/>



Note: Some fonts have some additional requirements/settings to operate; e.g.: to use the first two example fonts, the data embedded in the barcode must be contained between asterisks "*", i.e.: to write "**ABC123**" in barcode it should be entered as "***ABC123***", those asterisks will be ignored by the barcode scanner/reader. **For other fonts requirements please refer to each font individual documentation.**



Note: All mentioned **brands, websites, and companies** are only listed as examples and external references, material/information included in the external links are subject to change without notice.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.4. List Views

The list window (list view) displays layers, configurations, apertures, tools and other sections of a CAM document in a tabular form. The following functions are supported in the tabbed panels:

- User can edit, delete or add new entries. It is supported by *Copy/Paste*, *Delete* operations and in **Properties** dialogue boxes as well

- Multiple selection is supported by using the standard selection keys (**Ctrl** + mouse, **Shift** + mouse and **Shift** + arrows).
- All list view windows supports standard *Print Preview* and *Print* commands.

Selected and Active List Entry

entries in the list views can be selected by either the mouse or the up / down arrow keys. When a list view is active, the properties view shows information about the selected item. An item can be made active by either double clicking it or by choosing **Make Active** from the context menu. An **active object** is indicated by **bold** text. In the case a graphical object is selected, the name of the corresponding items in the list views (layer, config-item, layout) is indicated in **bold red** color.

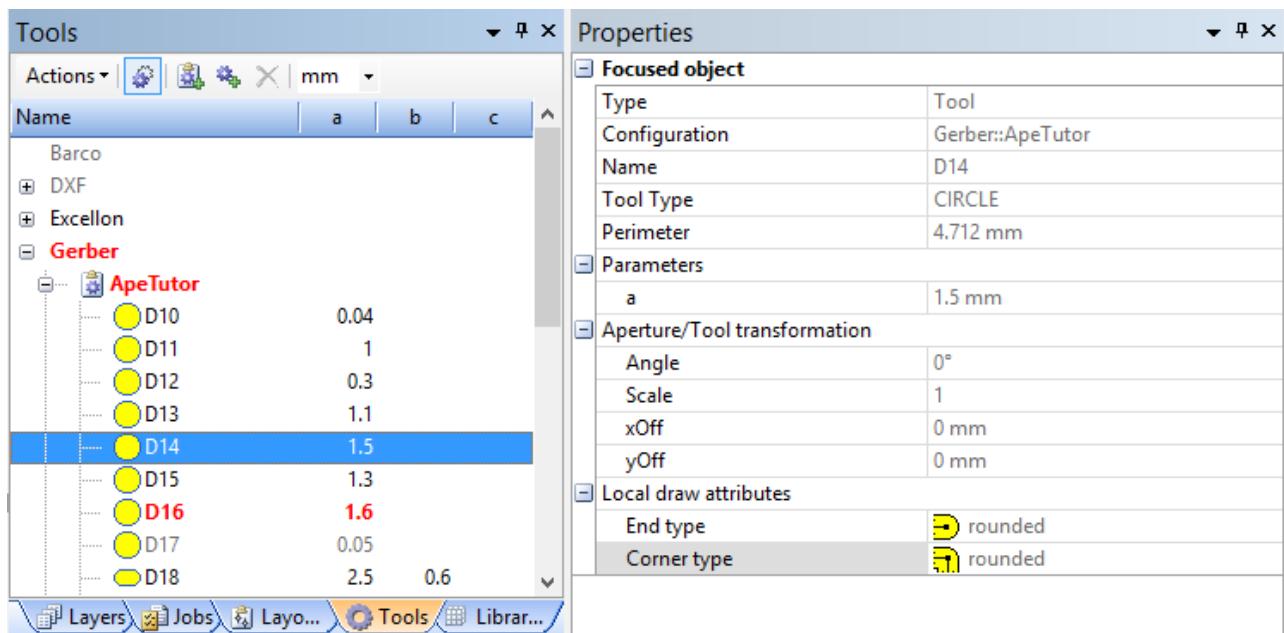


Figure 229. D14 is the selected item and the Properties View shows its characteristics.
D16 is the active item of the selected graphic object, shown in red bold color.

Paste Conflicts

Conflicts may appear when pasting data from one **CAM** document into another **CAM** document because objects with the same name already exist. For the following types conflict solution is supported through a special dialogue:

- Layers
- Tools
- Layouts
- Jobs

The paste dialogue will only show up only if conflict exists. User can choose to either overwrite, not overwrite, overwrite all, not overwrite all or change the name of the object.

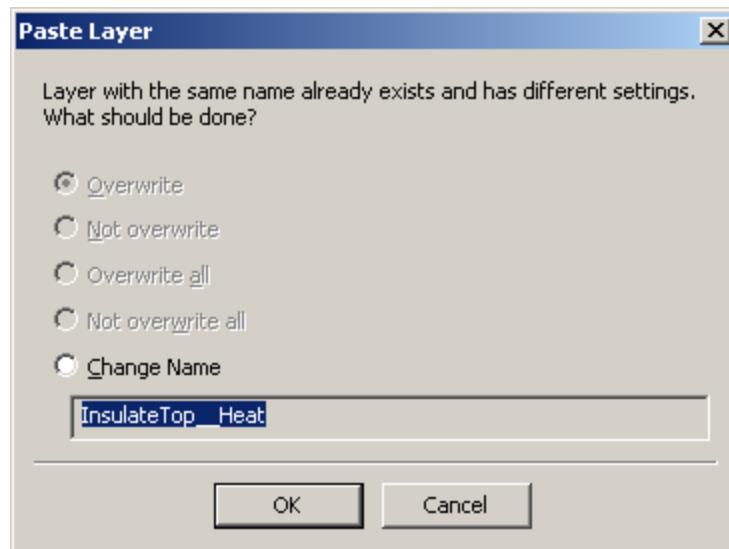


Figure 230. Paste Layer Dialogue

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

3.4.1. Layers

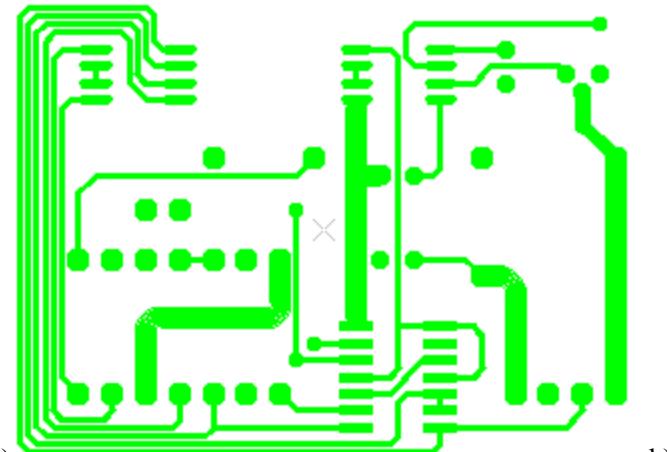
Every graphical object is assigned to a layer. Layers with their current settings are listed in the **Layers** view. The window can be activated in the following ways:

- Selecting **Layers** in the **View** menu;
- or clicking the layer button in the **Layers** toolbar;
- or by pressing and holding **Ctrl** + tapping on **Tab**, then select **Layers** in the dialogue window.

Layers								
Actions		Visible	Name	Order	Selectable	True Width	Outline	Colors
<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>	Fiducial	1	<input type="checkbox"/>			
<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>	TopLayer	5	<input type="checkbox"/>			
<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>	BottomLayer	10	<input type="checkbox"/>			
<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>	DrillPlated	2	<input type="checkbox"/>			
<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>	DrillUnplated	3	<input type="checkbox"/>			
<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>	BoardOutline	4	<input type="checkbox"/>			
<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>	TextTop	15	<input type="checkbox"/>			
<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>	TextBottom	16	<input type="checkbox"/>			
<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>	CuttingOutside	17	<input type="checkbox"/>			
<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>	CuttingInside	18	<input type="checkbox"/>			
<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>	SolderMaskTop	19	<input type="checkbox"/>			
<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>	SolderMaskBottom	20	<input type="checkbox"/>			
<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>	SolderPasteTop	26	<input type="checkbox"/>			
<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>	SolderPasteBottom	27	<input type="checkbox"/>			

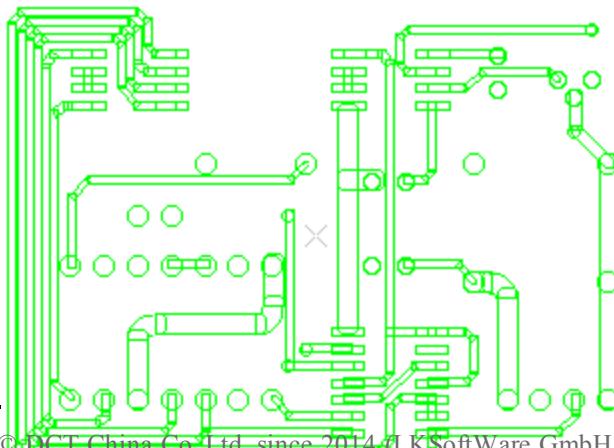
Figure 231. Layers view

- **Visible:** Click on this check box to make the marked layer visible or invisible, i.e.: it is no longer shown in a graphics window. At the same time, when a layer is invisible, it cannot be selected. If the layer is subsequently made visible again, it still cannot be selected and must be made selectable again.
 - **Name:** Specifies the name of the layer. A layer can be activated by pressing **Alt + double click** on its name, or selecting it from the **Layer** toolbar drop down list, or by selecting *Make Active* from the context sensitive menu (right mouse button on an applicable layer).
- The status of the layer is indicated by the text style and color:
- **Grey** - The layer is not assigned (has no available object data);
 - **Black** - The layer is assigned (has associated object data);
 - **Red** - The layer is assigned and some of the available objects are selected.
 - ****Bold** Text (**black** or **red**)** - the layer is active. New data are saved on this layer.
- **Order:** Click on the number to make the marked layer the highest (top) level. The number 1 is assigned to this layer and the number of all other layers is incremented by one.
 - **Selectable:** Click on this check-box to switch the marked layer to "not selectable" , i.e.: the concerned layer cannot be selected by any of the above facilities and thus not modified either. Clicking the check box again switches back to "selectable" .
 - **True Width:** Click on this check box to show the elements on the marked layer with their real line width (**True Width**). Otherwise the width=0 mode is used (**Line Only**).
 - **Outline:** Graphic elements (a line, a rectangle, a circle and a polygon) are shown filled as standard (**Filled**). Click on this check box to show the elements on the marked layer only in outline (**Outline**).



a)

b)



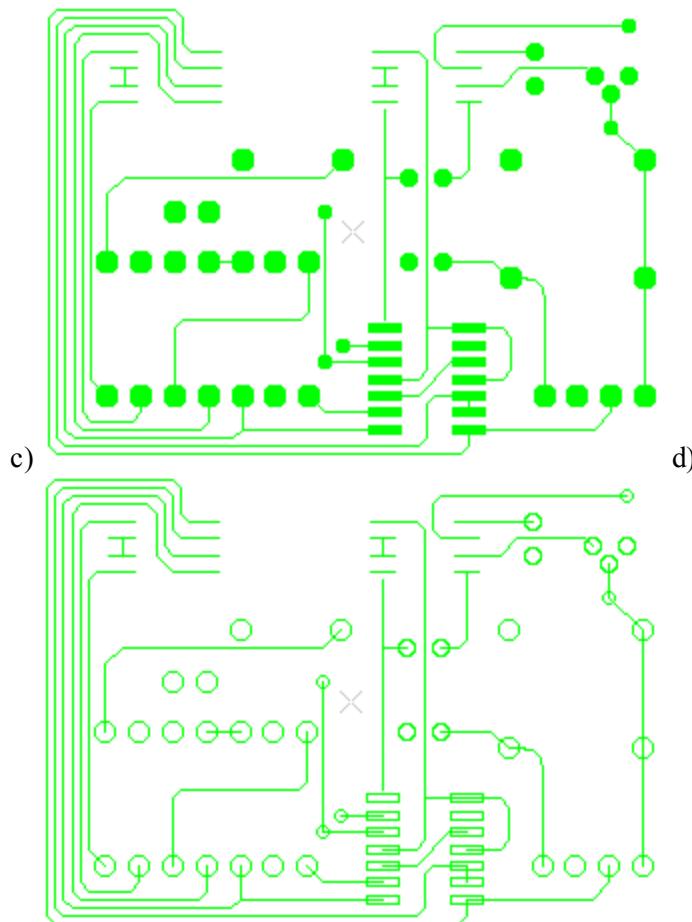


Figure 232. Graphic view: a) **True Width and Filled**, b) **True Width and Outline**, c) **Line Only and Filled**, d) **Line Only and Outline**

- **Color:** Specifies a color of the graphical object on a particular layer. The colored square is divided in two parts. The right part specifies a color of the flashes on the layer, the left - for other graphical objects on the same layer. There are two ways to select a color for the layer: In the [Layer properties](#) dialogue window (double click on an applicable layer to open the dialogue window);
- In the **Colors** dialogue window, which is invoked by using the **Select color** sub menu of the [context sensitive menu](#)):

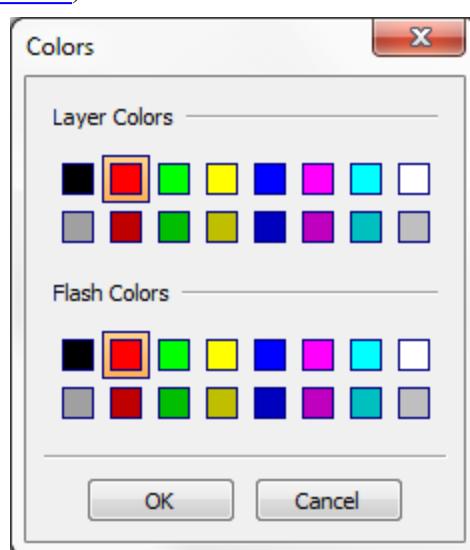


Figure 233. Colors dialogue window

Context Menu

A context sensitive menu is available (mouse right button) to activate, move, copy, add new, delete a layer, change properties of the layer, add graphical objects to the selected layer and etc.

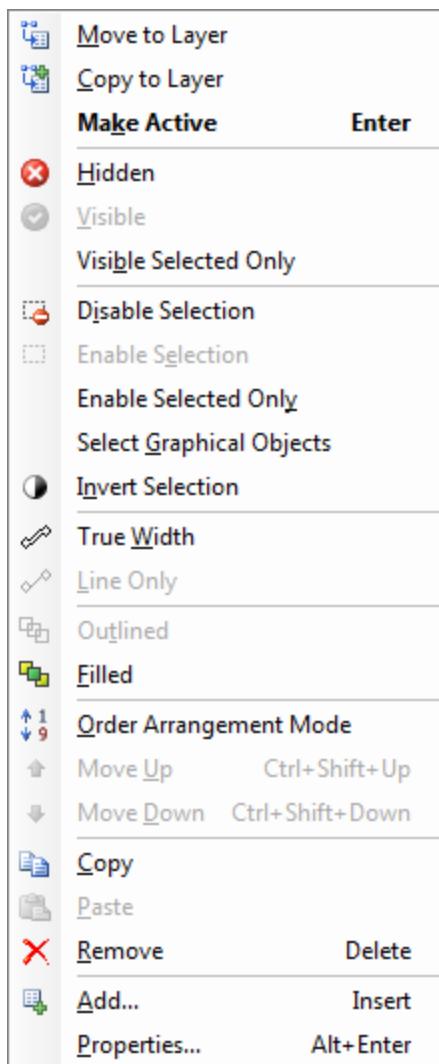


Figure 234. Layers view context sensitive menu

	Move to Layer	Move selected objects to the focused layer.
	Copy to Layer	Copy selected objects to the focused layer.
	Make Active	Make the current layer active.
	Hidden	Make the current layer invisible on the graphics window.
	Visible	Make the current layer visible on the graphics window.
	Visible Selected Only	Make the current layer visible on the graphics window, other layers made invisible.
	Disable Selection	Make the current layer non-selectable.
	Enable Selection	Make the current layer selectable.
	Enable Selected Only	Make only the current layer selectable.
	Select Graphic	Select graphic objects on the layout according to selected layer(s).

Objects	
 Invert Selection	Select all non-selected objects and deselect previously selected objects.
 True Width	Show elements on the marked layer with their real line width.
 Line Only	Show elements on the marked layer in line mode=0.
 Outlined	Show elements on the marked layer in outline only.
 Filled	Show all graphical elements filled.
 Order Arrangement Mode	Set the order arrangement mode <i>On</i> .
 Move Up	Change the order of the layer by moving it up.
 Move Down	Change the order of the layer by moving it down.
 Copy	Copy the selected layer to the standard clipboard.
 Paste	Extract the objects from the clipboard.
 Remove	Remove the selected layer.
 Add	Create a new layer.
 Properties	Open the Layer Properties window.

Toolbar

The **Layers** view has its own toolbar.



Figure 235. Layers view toolbar

 Actions ▾	The list of available actions in the Layers view. This list contains items available in the context sensitive menu.
 Show all layers	Show all layers. If the option is set <i>Off</i> , only layers, which contain any object are listed.
 All Visible and Selectable	Make all layers visible and selectable.
 Add Layer	Add a new layer.
 Remove Layer (Delete)	Remove a layer.
 Order Arrangement Mode	
 Move Up	Change the order of the layer by moving it up.
 Move Down	Change the order of the layer by moving it down.
 Move to Layer	Move selected objects to the focused layer.
 Copy to Layer	

Copy selected objects to the focused layer.

Layer Properties

The **Layer Properties** dialogue window helps to create a new layer, or change properties for an already created layer.

It comes up with:

- context menu (right mouse button);
- double-click on a cell in the **Name** column.

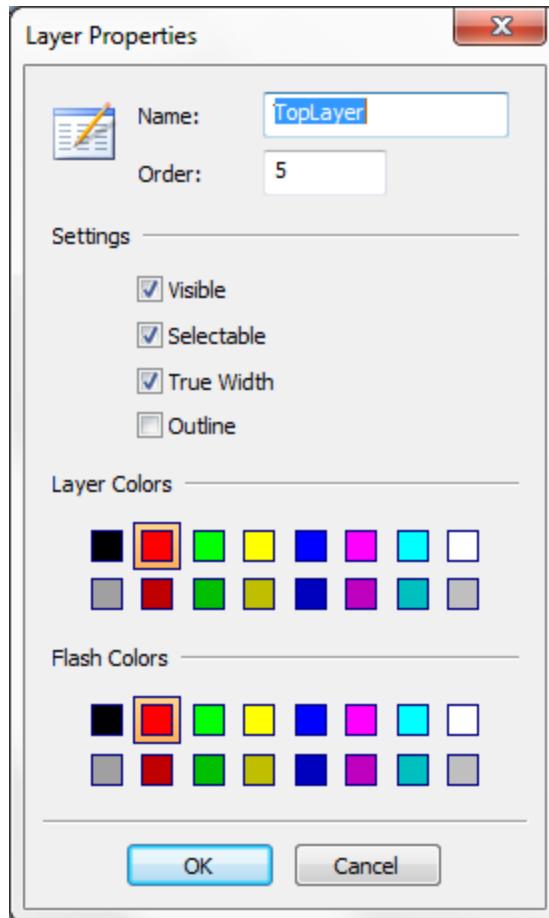


Figure 236. Layer Properties dialogue window

- **Name:** Specifies the name of the layer.
- **Order:** Indicates order number for the layer.

Settings group

- **Visible:** If *On*, the current layer is visible in the graphics window.
- **Selectable:** If *Off*, the current layer is not selectable and cannot be modified.
- **True Width:** If *On*, the elements on the marked layer are shown with their real line width.
- **Outline:** If *On*, the elements on the marked layer are shown only in outline.

Layer Colors group

Select a color to display graphic objects in the layer. The **Flash Colors** group works in a similar way but for flashes only.

The double click on a cell in the **Order** column makes this layer first in the list.



Note: New options will be applied to all selected layers simultaneously.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4. Tools View

A **tool**, also called **aperture** or **configItem** is an object with [geometric properties](#), that can be used to create flashes or to draw paths. In the case the object is used for milling or drilling machines, that are driven by certain NC formats, we call the object a **tool**. In the case the object is used for photo plotters or layouts formats, we call it an **aperture**. The generic term in *CircuitCAM* for all these cases is **configItem**.

ConfigItems are grouped into lists, that are called configurations, tool lists or aperture lists. Each list belongs to a format, and depending on the type of the format, special [properties](#) apply to these lists. For the **PostScript** format any kind of configItems can be used. Tools and tool lists can also be created independent of a particular format under the **Diverse** category.

The tools view is organized in a tree structure, the highest level is the data format, the second level is individual lists, and the third and final level is the tools and apertures.

Supported NC formats:

- [Excelon](#)
- [LMD](#)
- [G-Code](#)
- [HP-GL](#)

Supported Layouts/photo plotters formats:

- [Gerber](#) -including extended Gerber "[Gerber X](#)"
- [Barco](#)
- IEC61182
- [ODB++](#)

Other formats:

- PostScript
- Diverse

The **Tools** view can be activated in the following ways:

- Select **Tools** in the **View** menu, or you can select a tool from the combo box in the standard toolbar;
- or by pressing and holding **Ctrl** + tapping on **Tab**, then select **Tools** in the dialogue window.

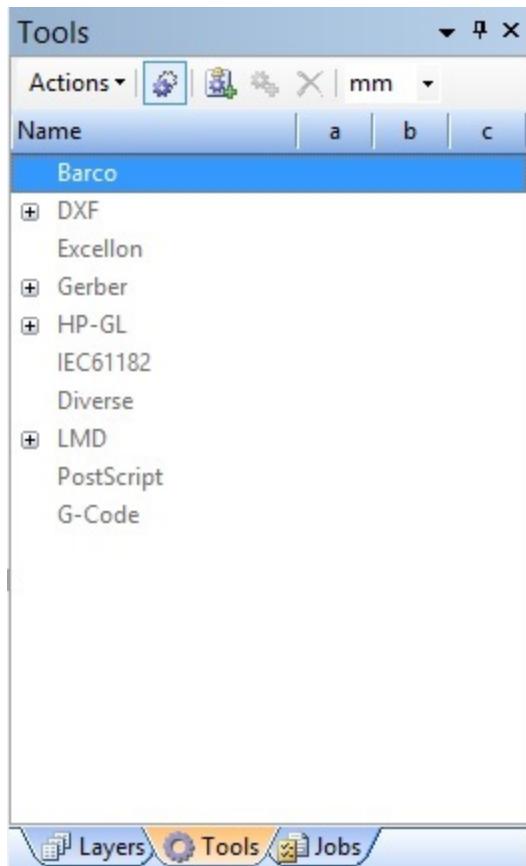


Figure 237. Tool view

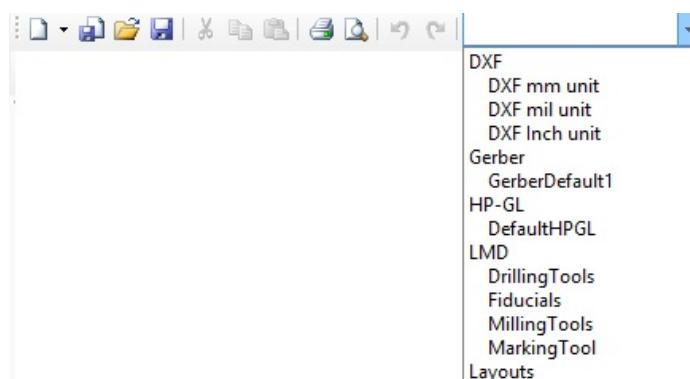


Figure 238. Combo box, standard toolbar view

Expand the nodes to view the list of apertures/tools:

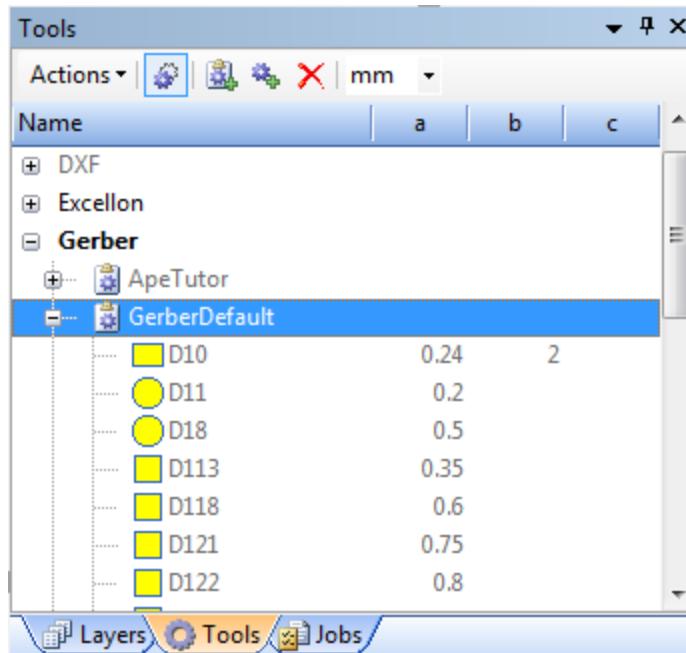


Figure 239. Apertures/Tools list in Tools view

The tools view is divided into several columns:

- **Name:** This field shows the type, the list name, and the aperture/tool name with the corresponding shape icon; identifying the shape of the tool: - circle; - square; - oval; - rectangle; - rounded rectangle; - bevel rectangle; - octagon; - marker; - special; - undefined.
- **a, b, c, ... , j :** These fields describe the shape dimensions. E.g.: **a** - value means a diameter of a circle or a width of a rectangle/oval; **b** - height of the rectangle; **c** - value sets a bevel or radius for a bevel/rounded rectangle. In case IEC special shape library is used, these values are set according to the shape description that is standard of IEC library. An empty field means either the value is zero or this field is not used. If a tool is used only in a drawing mode, **a** and **b** fields show the end type and the corner type of the path.

To see more detailed tool properties or to modify them (e.g.: to change mode, type, dimensions and so on) a special tool dialogue window can be used (see [Tool Properties](#)).

Context Menu

While using the right mouse button in the client area of the **Tools** view, the following popup menu (mouse *context sensitive menu*) will appear:

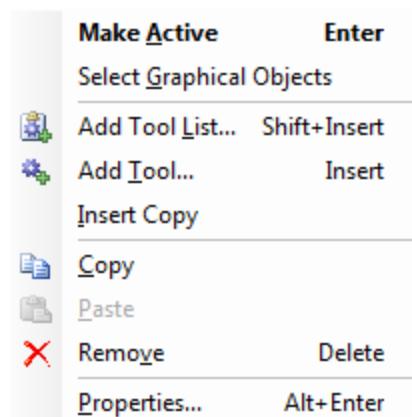


Figure 240. Tools view context sensitive menu

Make Active	Set the tool for a graphical object (path, flash).
Select Graphical Objects	Select graphical objects on the layout according to selected tool list(s)/tool(s).
 Add Tool List...	Create a new tool list.
 Add Tool...	Create a new tool.
Insert Copy	Create a new tool list/tool copy.
 Copy	Copy the selected tool list/tool to the standard clipboard.
 Paste	Extract the objects from the clipboard.
 Remove	Delete selected tool/list.
Properties...	Activate the Aperture/Tool properties window.

Toolbar

The **Tools** view has its own toolbar.

**Figure 241. Tools view toolbar**

Actions ▾	The list of available actions in the Tools view. This list contains same items available in the context sensitive menu.
 Show All Tools	Show or hide unused apertures/tools in the tools view.
 Add Tool List	Create a new tool list. A new tool list will appear below the selected tool list.
 Add Tool	Create a new tool. A new tool will appear below the selected tool in the list.
 Remove Tool/List	Delete selected tool/list. Only unused tools can be removed.
 Aperture/Tool Unit	Change unit for aperture/tool dimensions.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.0.0.1 Tools List Properties

Gerber / Excellon / IEC61182 / Sieb & Meyer Format Properties Dialogue

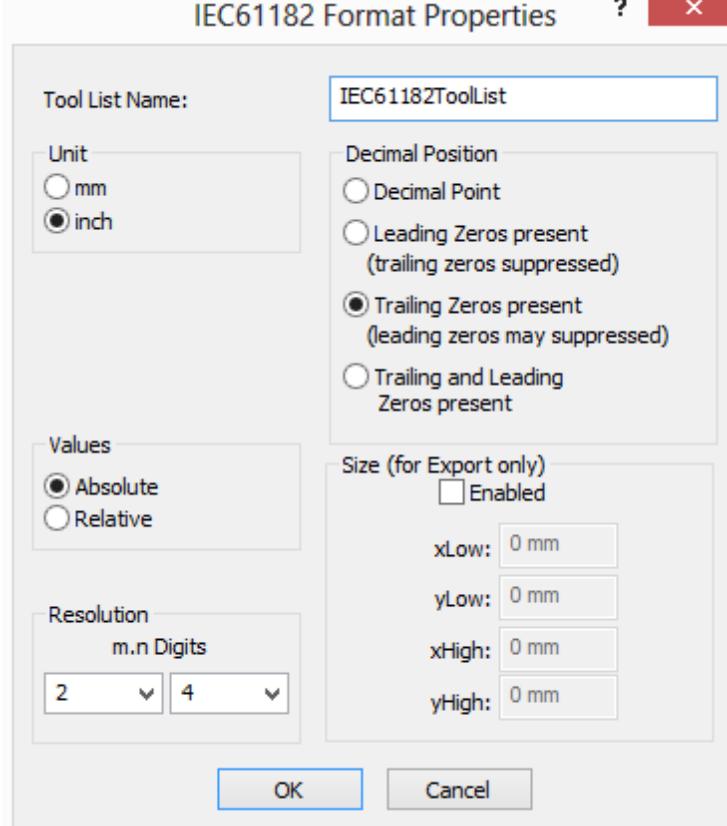
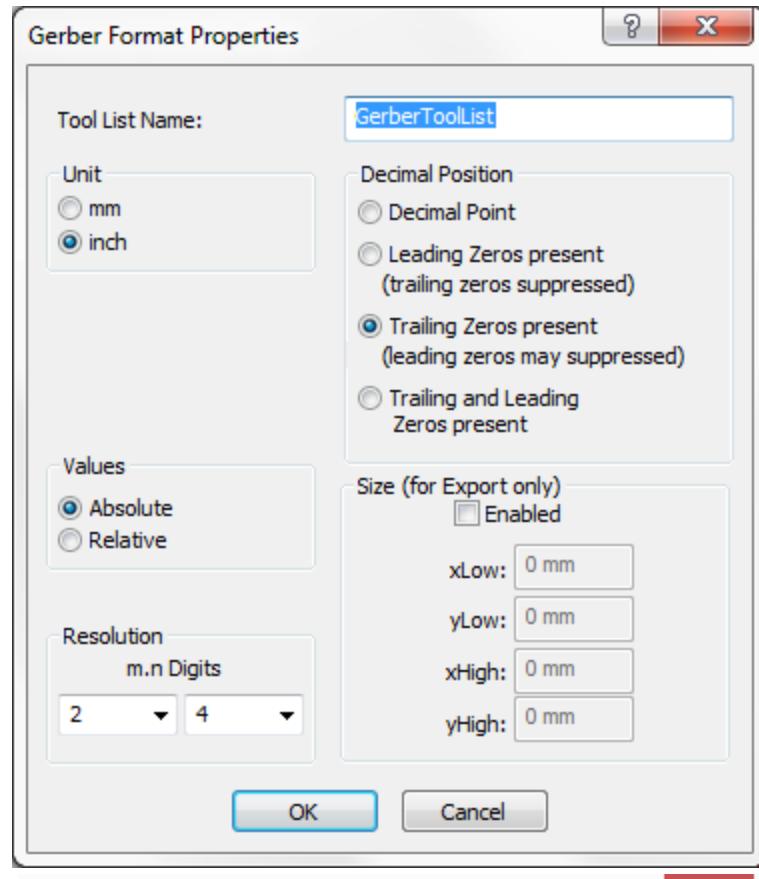


Figure 242. Gerber (Excelon / IEC61182 / Sieb & Mayer) Format Properties dialogue window

- **Unit:** Select a unit (mm or inch) which is used to represent the coordinates of all graphical objects drawn using this tool list.
- **Decimal position:** The position of the decimal point has to be determined. To do this, the **m.n** digits, and the information whether leading or trailing zeros are to be output, are used together.
- **Values:** Coordinates can be in absolute or relative (incremental) form.
- **Digits m.n:** These values are used if no decimal point format is set. "m" indicates the number of digits before the decimal point and "n" the number of digits after the decimal point.
- **Size (for Export only):** Indicates the size (bounding box) of the board.

Coordinate Format

Right click on a tool/aperture list node and select **Properties** to open a particular tool/aperture list properties dialogue window. To add a new tool list, click on the **Add Tool List** button. The **Coordinate Format** dialogue window helps to create a new aperture/tool list or change properties for already created aperture/tool list.

Tool List Name field allows the user to modify a name of the tool list.

Postscript Format Properties Dialogue

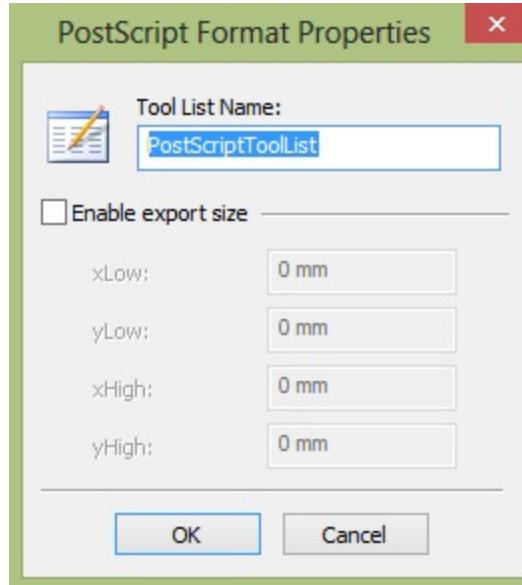


Figure 243. Postscript Format Properties dialogue

Enable export size: If **Enabled** flag is set *On* these four values specify the size of the available export area. This information is used for export only. See also **Enable export size** in General Settings under [Default Configuration](#), see also [Data Formats](#)

HP-GL/Diverse/LMD Format Properties Dialogue

The same **data format dialogue** window is used for **HP-GL**, **Diverse** and **LMD** configurations.

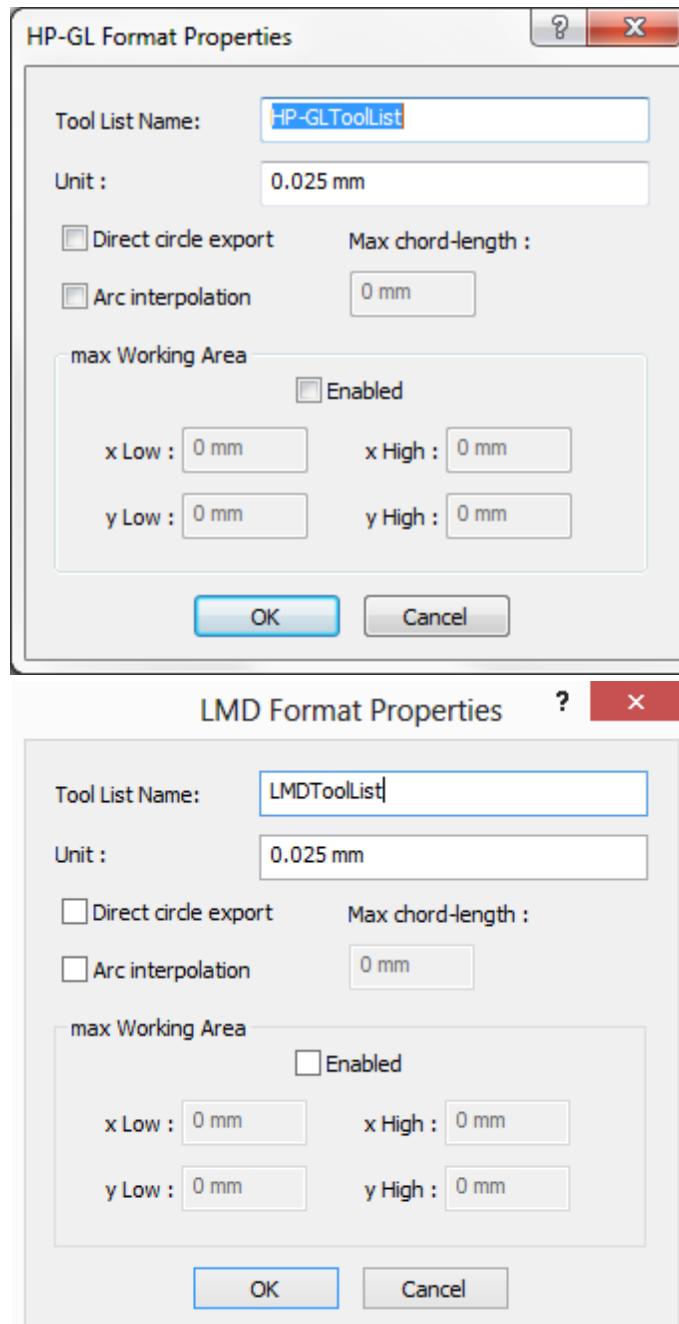


Figure 244. HP-GL (Diverse / LMD) Format Properties dialogue window

- **Unit:** The resolution can vary from plotter to plotter. The HP7475 plotter from *Hewlett Packard* (one of the first with the **HP-GL** language) has a resolution of 0.041 mm.
- **Direct circle export:** If flag is *On*, the **HP-GL** exports simple circle. Otherwise (circle like shapes) are exported as arc segments.
- **Arc interpolation:** If flag is *On*, then an arc in closed/opened path and a circle in closed path are exported as a set of lines. Interpolation step depends on values from "Unit" and "Max chord-length".
- **Max chord-length:** Specifies the maximum chord length during arc interpolation.

- **Max Working Area:** If **Enabled** flag is set **On**, these four values specify the size of the available plot area. This section is used for export only.
- see also [Data Formats](#)

DXF Format Properties Dialogue

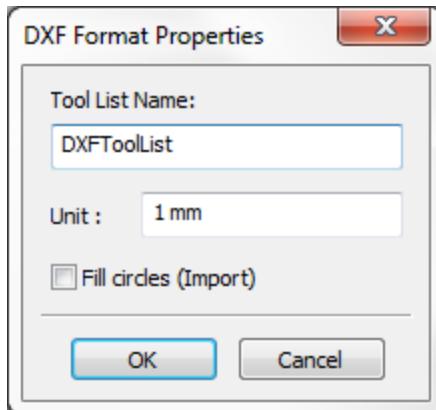


Figure 245. DXF Format Properties dialogue window

- **Unit:** Shows a unit value, which is used to represent the coordinates of all graphical objects drawn using this tool list. For import this setting is only used as default and can be overwritten by the appropriate commands.
- **Fill Circles:** If flag is set *On* an imported circle is filled. Otherwise the circle is assumed as circle in Closed-path (a contour with zero path width).
- see also [Data Formats](#)

Barco Format Properties Dialogue

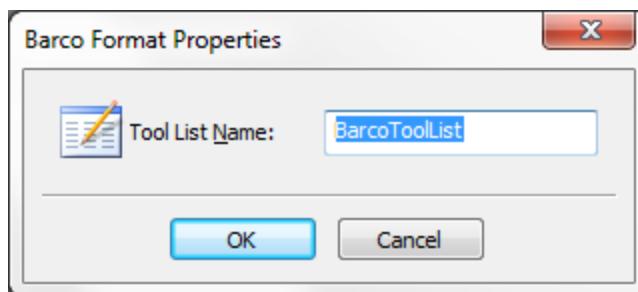


Figure 246. Barco Format Properties dialogue window

This empty dialogue means that there is no possibility to change this format's parameters, see also [Data Formats](#).

G-Code Format Properties Dialogue

For G-Code export the assumption is that the PCB is in the x-y plane of the machine. Several values for the Z-axis have to be provided for rapid movement, milling, and drilling. The value depends on the absolute Z-position of the machine in relation to the work piece.

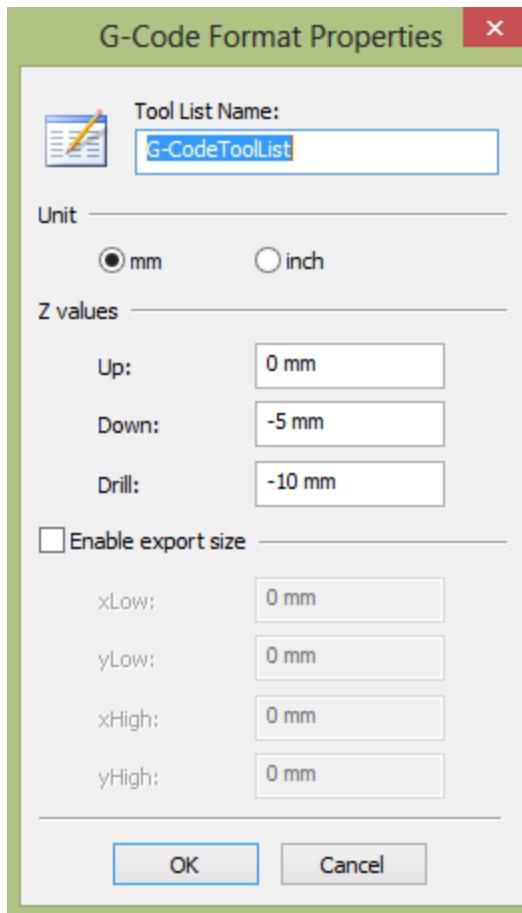


Figure 247. G-Code Format Properties dialogue window

- **Unit:** Select a unit (mm or inch) which is used to represent the coordinates of all graphical objects drawn using this tool list.
- **Up:** Defines the Z-value for rapid movements without milling/drilling.
- **Down:** Defines the Z-value for milling operations.
- **Drill:** Defines the lowest Z-value for drilling.
- **Enable export size:** If Enabled flag is set *On* these four values specify the size of the available plot area. This section is used for export only.

see also [Data Formats](#)

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.0.0.2 Tools Properties

A **tool**, also called **aperture** or **configItem** is an object with geometric properties, that can be used to create flashes or to draw paths. The properties of a tool/aperture are specified in the aperture/tool properties dialogue. The main property of an aperture/tool is its type. Various dimensions of the tools, are expressed in the letters "**a, b, c, d, e, f, g, h, i, j**". The following types are supported;

- Rectangle: A rectangle is defined by two opposing points, and it is oriented parallel to **x** and **y** axis, the parameter **a** defines **x** direction value, **b** defines **y** direction value.
- Circle: A circle is defined by a centre point, and a diameter. the parameter **a** defines the diameter.
- Oval:

- Square: A square consists of four equal sides, and four 90 degrees angles, and it is oriented parallel to x and y axis, the parameter a defines the side length.
- Rounded Rectangle:
- Bevelled Rectangle:
- Octagon:
- Marker:
- Undefined:
- Special:
- Off:

The **Aperture/Tool Properties** dialogue window helps to create a new aperture/tool, or change properties for the already created aperture/tool.

To open Aperture/tool properties dialogue, select tool from tool list, using your right mouse button, click on selected tool then properties. See also [Active config-item](#), [Library View](#).

The **Name** field specifies an aperture/tool name.

Flash Attributes

Shows the tool shape and size for the flash mode (as shown below and is set by default):

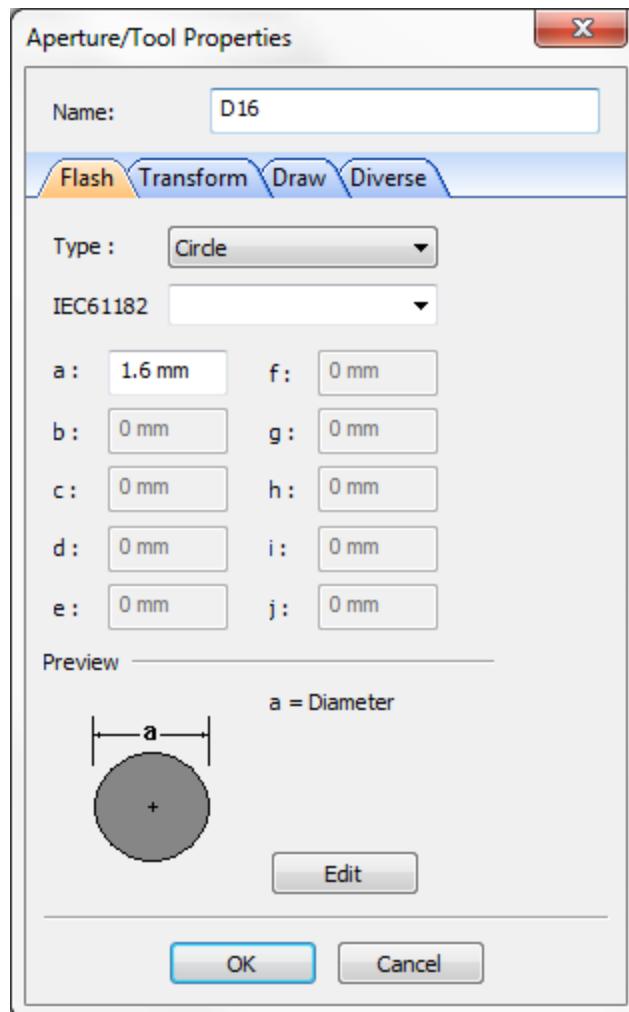


Figure 248. Aperture/Tool Properties dialogue window, Flash Attributes panel

- **Name:** Specifies an aperture/tool name.
- **Type:** Specifies the shape of an aperture/tool for the flash mode. To change the type (shape) to *circle/square/oval/.../Marker/Special*, click on the drop down menu next to the field, and edit the fields **a,b,c,...,i,f** if needed.
Select the *Off* option, to switch off the usage of this tool in a flash mode (when the tool is needed only for drawing).
- **IEC61182:** Specifies the shape from the special shape library. To set a shape from the special shape library, select the *Special* line and select a type from **IEC61182** combo box (or type from the keyboard).
- **a, b, c, ..., j:** These fields describe the shape dimensions. E.g.: **a** value is the diameter of a circle or the width of the rectangle/oval; **b** – the height of the rectangle; **c** value sets a bevel or a radius for a bevel/rounded rectangle.
- **Edit:** Opens a new dialogue, for graphical editing in tool properties, the graphical editing of the aperture/tool layout can be ended by either saving the changes or discarding them. It can either be ended by choosing *Save/Discard* from the **Item Edit** window shown below, or by accessing the menus; **Edit ---> Item Edit ---> then choosing Save or Discard.**

The bottom part of the dialogue "**Preview**" shows the graphical view of the selected tool/aperture, and explains the meaning of fields (**a/b/c/.../i/f**).

Transform

Shows the origin point **xOff/yOff** according to how the tool is drawn by using dimensions from the (**a/b/c/.../i/f**) fields, how it is rotated, and what scale is used. These fields are only available for flash apertures which does not belong to the *Special* group.

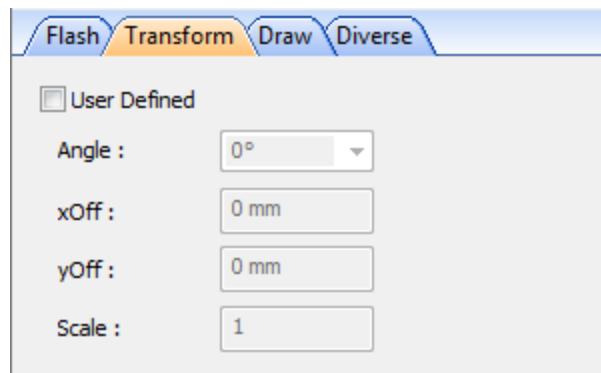


Figure 249. Aperture/Tool Properties dialogue window, Transform panel

If the **User Defined** check box is switched *On*, it is possible to modify the displayed values.

Draw

Shows the tool size and some additional features for the *draw* mode.



Note: The tool shape is the same as in **Flash** page.

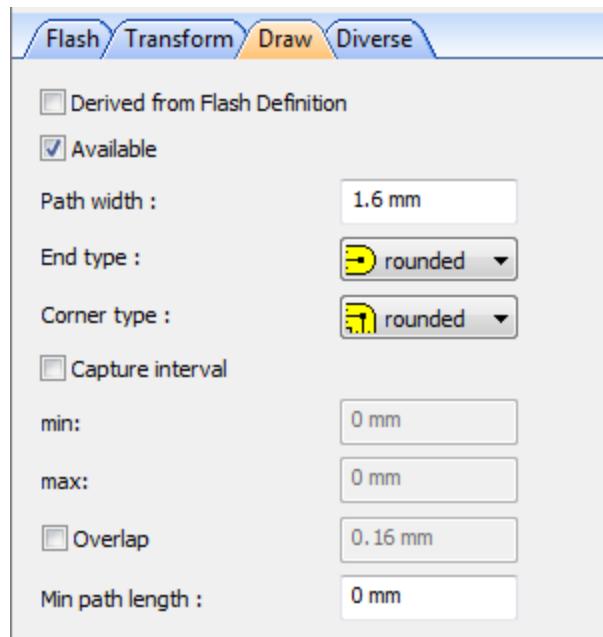


Figure 250. Aperture/Tool Properties dialogue window, Draw panel

- If the **Derived from Flash Definition** check box is switched *Off* and the **Available** check box is *On*, it is possible to modify these values. Otherwise they are set from the flash attributes.
- Path width:** defines the thickness of lines for the path drawing.
- End type:** the type of the start and end points of the path.
- Corner type:** specifies the corner's type of the path.
- Capture interval:** specifies the **min** and **max** values, which are used to select an applicable drawing tool in export operations. The size of selected tool must be inside the capture interval ($\min < \text{pathWidth} > \max$). If **Capture interval** flag is *Off*, default min and max values are used: **min** = $\text{pathWidth} * 0.95$, **max** = $\text{pathWidth} * 1.05$.



Note: The capture interval option is used only for drawing. The capture interval for flashing is defined in the **Diverse** page.

- Overlap value:** (when flag **Overlap=On**) is used:
 - in *Export* task to specify the overlap value in shapes filling mode;
 - in *Export* task to store an initial overlap value of scanner fields.
- Min path length:** defines the length value of the shortest path, which participates in the following conditions:
 - in *Export*: $\text{pathWidth} > \text{Min path length}$;

Diverse

When the **User Defined** check box is switched *on*, shows and allows modification of the tool capture interval for flashing:

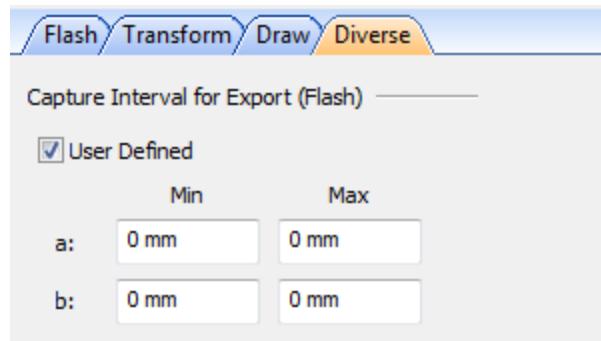


Figure 251. Aperture/Tool Properties dialogue window, Diverse panel

To output a pad as a flash, the tool size (**a/b/c/.../i/f**) must be inside the specified **Min** and **Max** values. If **User Defined** flag is *Off*, default min and max values are used (e.g.: for **a: Min = a * 0.95, Max = a * 1.05**).

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.0.0.3 Tool Edit Mode

Tools/apertures of type special, do not have parameters, and thus can only be edited graphically. Tools/apertures of other types can also be edited graphically, but they are then automatically converted into special tools.

Graphical editing of a tool is accessible through the **tool properties**, under **Flash** tab, when clicking the **Edit** button, the current shape of the tool is converted into a special temporary layout, with a unique name e.g.: `@@@_Item_Edit_GerberDefault1_D13`. When finishing the graphical editing of a tool, the special layout is removed.

While graphically editing a tool/aperture, all normal editing operations of a layout are available, with the exception that only one predefined temporary layer can be used.

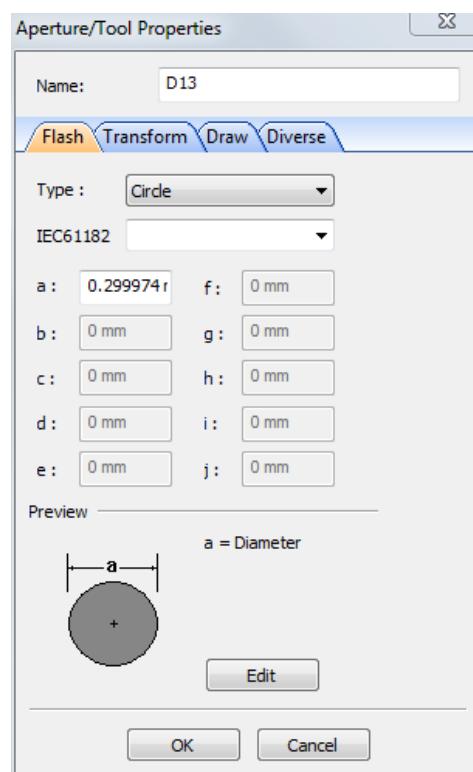


Figure 252. Tool properties window.

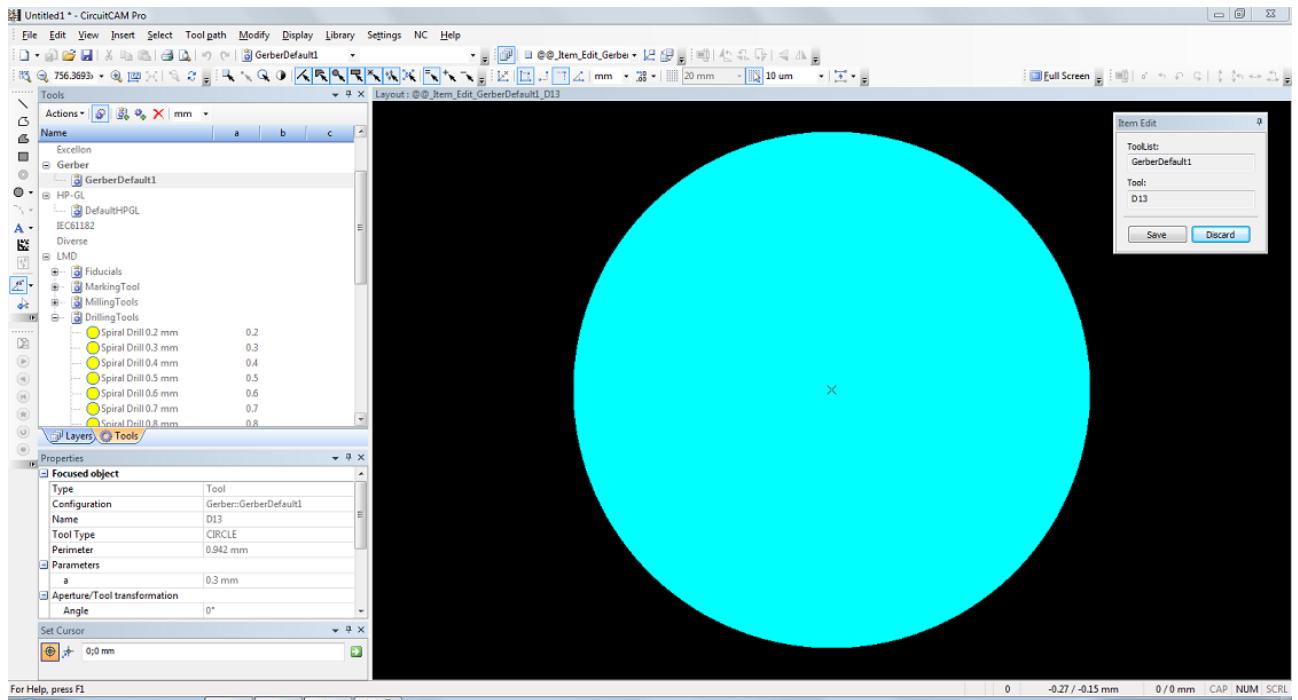


Figure 253. Tool Edit mode.

Editing a tool works exactly like the graphical [editing](#) and [insertion](#) operations; you can modify shape or size, you can add another path or polygon, you can create a cutout.....etc.

After finishing the desired editing, the edit mode can be closed by choosing either to save or discard the changes.

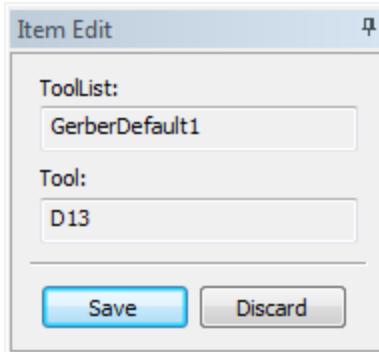


Figure 254. Ending the item edit mode by selecting Save/Discard changes.

Editing mode can be ended also by accessing the menu; **Edit ---> Item Edit ---> then choosing save or discard.**

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.0.1. Jobs and Quick Access

The **Jobs** view contains a list of available jobs, which can be of type:

- [Stencil Frame generator](#),
- [Export](#),
- [Scripts](#).

The details of a job can be defined once stored in a CAM or CAT file, and can be executed many times.

The **Jobs** view panel can be activated in the following ways:

- Select **Jobs** in the **View** menu;
- or by pressing and holding **Ctrl** + tapping on **Tab**, then select **Jobs** in the dialogue window.

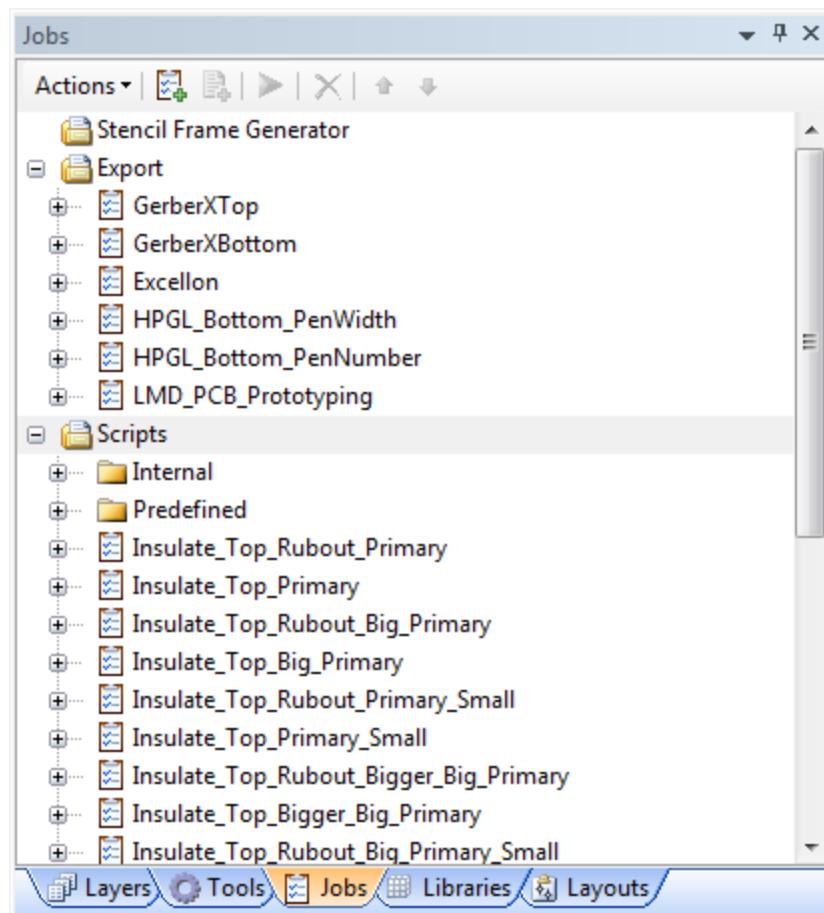


Figure 255. Jobs view

In case the **Show in quick access** option in script **Properties** window is *checked*, the selected job is included into the **Quick Access** view.

Various jobs properties:

- [Stencil Frame generator](#) job properties;
- [Export Job](#) and [Export Task](#) properties.
- [Script](#) properties.

Context Sensitive Menu (Right Mouse Button Menu)

When clicking the right mouse button in the client area of the **Jobs** panel, one of the following *context sensitive menus*) will appear:

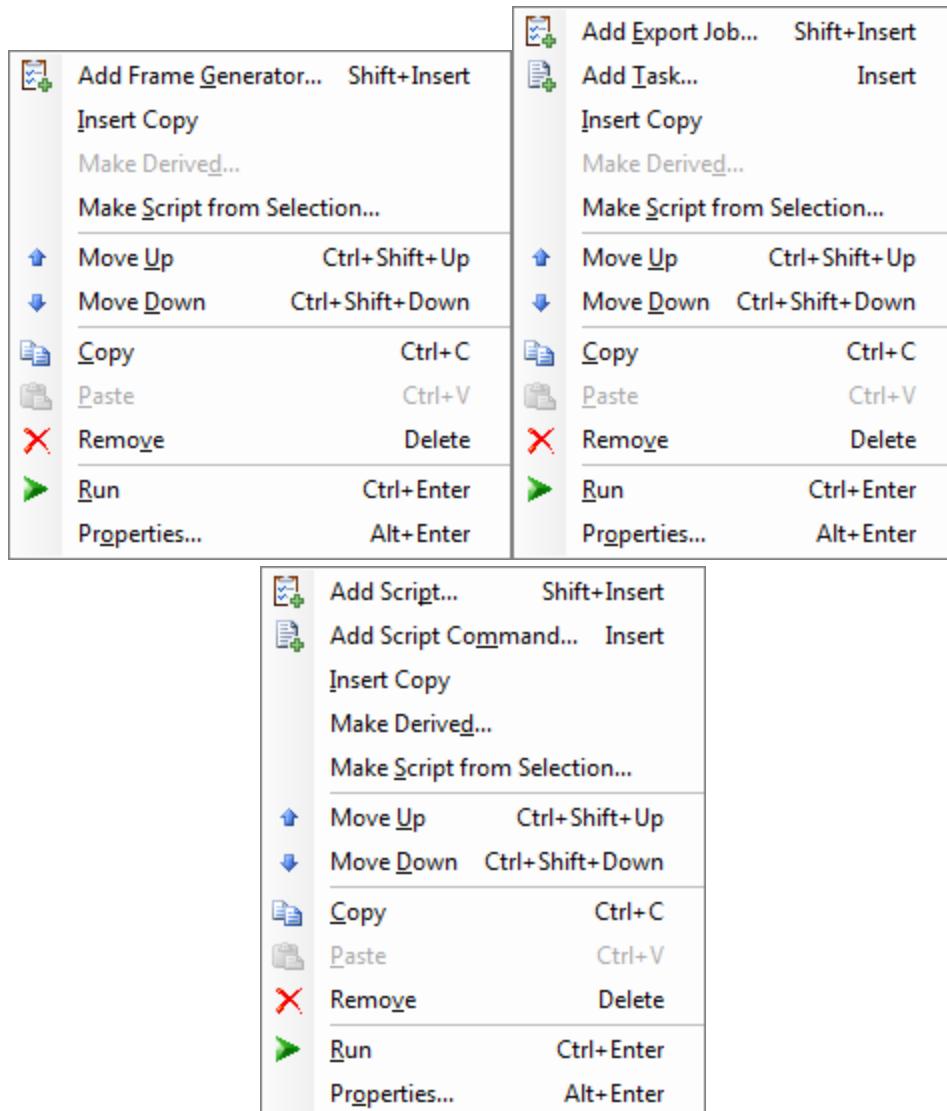


Figure 256. Jobs view context sensitive menus

Icon	Function	Description
	Add Frame Generator...	Create a new Frame Generator job.
	Add Export Job...	Create a new Export job.
	Add Task...	Create a new task under the selected Export Job .
	Add Script...	Create a new Script .
	Add Script Command...	Create a new command under the selected Script .
	Insert Copy	Create a copy of the selected job/task.
	Make derived...	Create a new script, with parameters derived from the selected script. (This function is only applicable with Scripts)

Icon	Function	Description
	Make Script from Selection	Create a new script from selected jobs.
	Move Up	Change the order of the Job/Command by moving it up.
	Move Down	Change the order of the Job/Command by moving it down.
	Copy	Copy the selected Job/Command to the standard clipboard.
	Paste	Insert Clipboard contents.
	Remove	Remove the selected Job or Command.
	Run	Run the selected Job.
	Properties...	Edit the selected Job or Command.



Note: Paste dialogue will appear if items have same names but items are different to choose whether to "Override, Not override, Override all, Not override all, Change names".

Toolbar

The **Jobs** view window has its own toolbar; this toolbar contains the same functions accessible from the context menu.



Figure 257. Jobs view toolbar

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.0.1.1 SMT Stencil Frame Generator

The SMT stencil frame generator creates the outer geometry objects of a stencil:

- A rectangular shape.
- A rounded-rectangular shape.
- Only the corners of the rectangular/rounded shape with middle segments.
- Rectangular/rounded shape with a user specified gaps.

The created frame is displayed as a path with width 0 on the destination layer.

In addition the frame generator has a possibility to create and add a pattern of fixing holes in the generated frame.

SMT Stencil Frame Generator Dialogue Box

To open the dialogue box, select an existing job under *Stencil Frame Generator* branch or *add a new job* in the **Jobs** view.

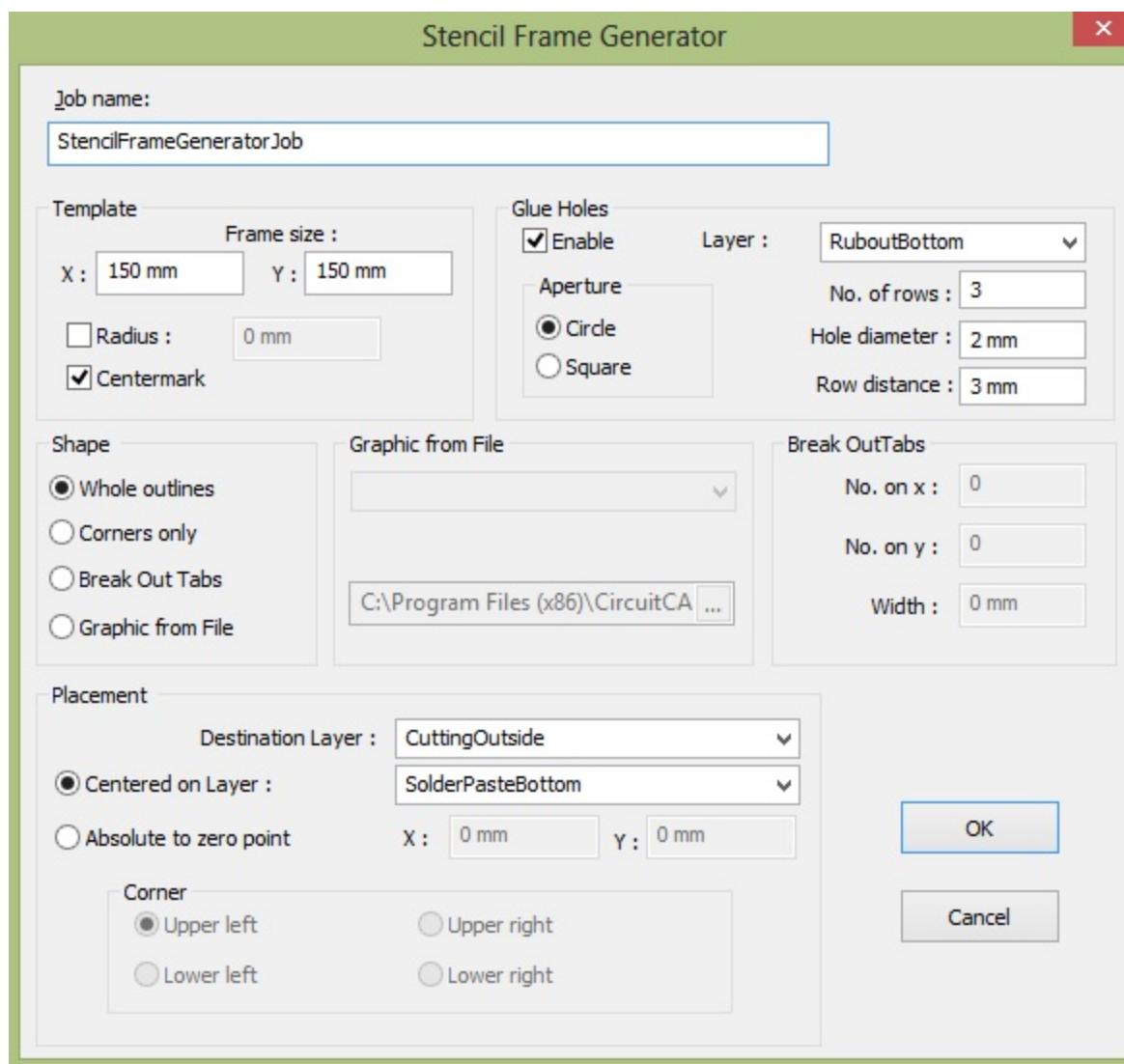


Figure 258. Stencil Frame Generator dialogue window

- **Job name:** Allows the user to enter/edit a job name.
- **Frame size:** Defines the frame-box size; switched *On* check-box.
- **Radius:** Indicates that the rounded shape will be drawn. The field of the radius (not diameter) size is available; in case the **Radius** check-box is switched *On*.
- **Centermark:** Is switched *On*, a special marker (like a “V”) with a width of 2 and a height of 3 mm at all sides is generated.
- **Whole outlines:** Means that a rectangular shape or a rounded rectangular shape will be created.
- **Corners only:** Creates either the corners [15 mm] of the rectangular shape with middle segments [20 mm] or the corners [90 degree arc + 10 mm line extensions] of the rounded rectangular shape with middle segments [20 mm] if the **Radius** check-box is switched *On*.
- **Break Out Tabs:** Means that a rectangular shape or a rounded rectangular shape with specified number of gaps will be created. Parameters of such gaps are controlled in the special **Break Out Tabs** group.
- **Graphic from File:** Copies the whole layout from the selected file on the destination layer. To choose the file a special **Graphic from File** tab is used.



Note: If the size **X/Y** in the **Whole outline** mode, is smaller than 50 mm, the middle segments are skipped. If the size **X/Y** is smaller than 30 mm, the corners are connected.

- **Break Out Tabs** group: **No on x** defines a number of gaps in **x** direction; **No on y** defines a number of gaps in **y** direction; **Width** defines a width of the gap (the same in both directions).



Notes:

1. Group is enabled only if **Break Out Tabs** shape is set.
2. In case of even number of gaps the *centermark* is allowed.

- **Destination Layer:** Specifies a layer on which the generated frame is placed. It can be chosen from the list or entered as a new one.
- **Centred on Layer** mode: The frame is centred and placed around the sum of all objects on one of *source* layers. The *source* layer can be chosen from the list or entered as a new one.
- **Absolute to zero point:** Absolute to zero point mode places the frame as absolute, i.e.: one of the corners of the generated frame is at the given **X,Y** coordinates. This corner is chosen from the **Corner** group.
- **Glue Holes:** The glue (fixing) holes will be generated only if **Enable** flag is switched *On*.
- **Aperture:** Specifies a type of the glue hole. Only two types of flashes are allowed - circle and square.
- **No of rows:** Specifies the number of rows of fixing holes. The number is the same in both *x* and *y* directions. If this number is zero the flag **Enable** will be switched to *Off* automatically.
- **Hole diameter:** Specifies the diameter of the hole. All holes have the same diameter.
- **Row distance:** Specifies the distance between two rows. To get a regular placement of holes, the distance can be recalculated and be different in *x* and *y* directions.
- **Layer:** Specifies the layer where generated fixing holes will be placed. It can be chosen from the list or entered as a new one.

The figure below shows a frame with circular glue holes generated with parameters that were set in the dialogue box.

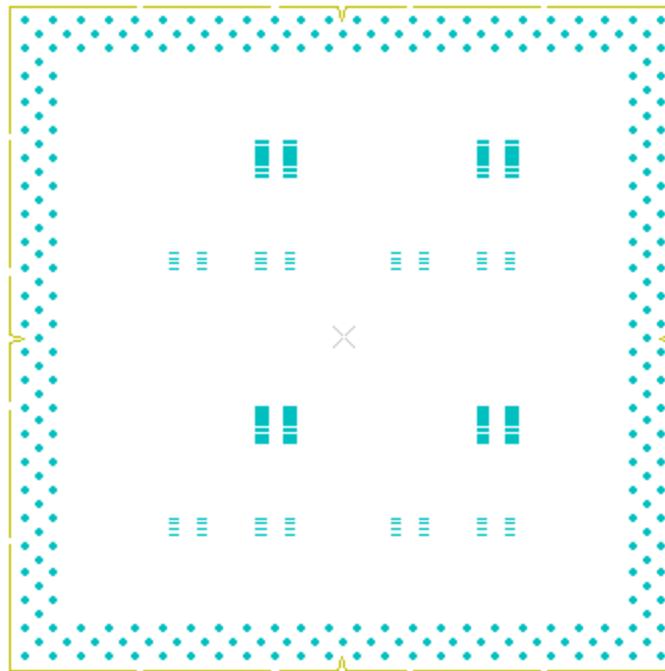


Figure 259. Frame with circular glue holes

- **OK** button saves all the values in the job.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.0.1.2 Export Job

With export job it is possible to convert the contents of the current layout into formats other than the CAM format. All supported formats are graphical formats, except for [layout instances](#), which only provides a list of layout instance names. For all other formats an export job is divided into smaller *export tasks*, which allow exporting of individual layers with different parameters. Available export jobs are listed in the [Jobs](#) view and also under **File -> Export** menu. Export jobs can be executed either by the **Run** button in the **Jobs** view or via **File -> Export -> ...** menu.

To create a new export job, activate the **Jobs** view: select the **Jobs** submenu in the **View** menu or click the **Jobs** tab on the left side of the *CircuitCAM* window. Then select **Export** job type in the list, press on the **Add Script** button in the **Jobs** view toolbar or right click on the **Export** branch and select the **Add Script** menu in the context sensitive menu. The **Export Job Properties** dialogue appears.

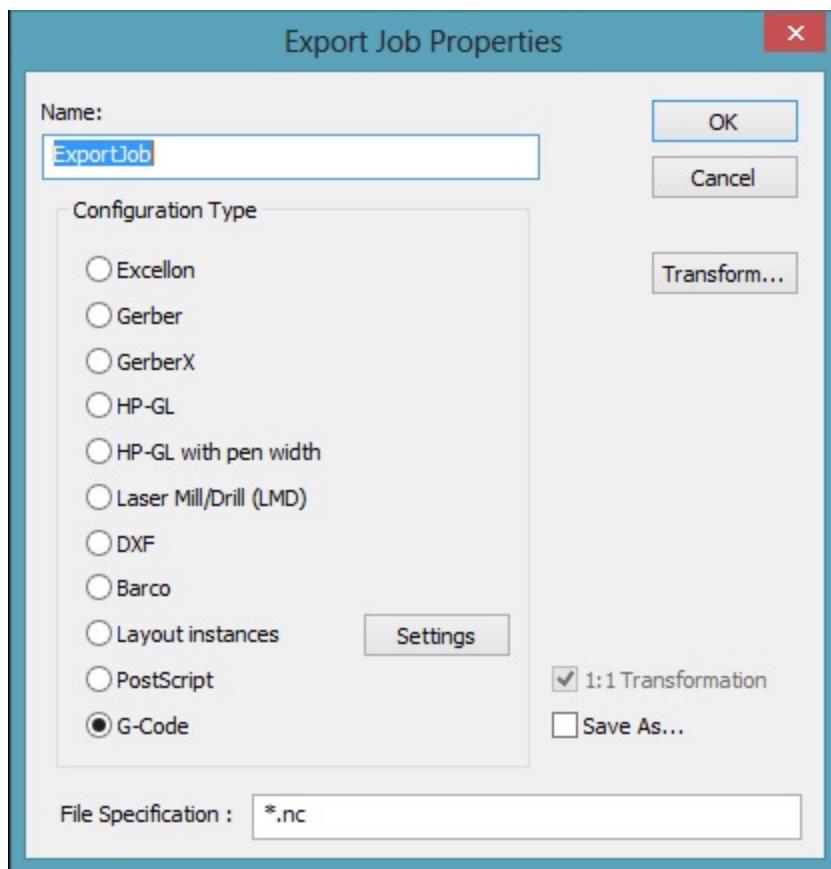


Figure 260. Export Job Properties dialogue window

- **Name:** specifies the name of the job.
- **Configuration Type group:** provides a list of selection buttons to select a format type. For more information about data formats, see [Data Formats](#).
- **File Specification:** unless otherwise specified, the file specification is relative to the directory where the CAM file is stored. In the case that a new CAM document was created (Untitled N) a Save As dialogue shows up for storing the CAM file in a directory to choose. The file specification may contain the character "*" that is then during export replaced by the name of the CAM file (without its extension). Examples:
 - Assuming that the open CAM file is *c:\test\example.cam*
 - "*xyz.abc*" will result into an exported file *c:\test\xyz.abc*,
 - "**.abc*" will result into an exported file *c:\test\example.abc*,
 - "*New_*5.abc*" will result into an exported file *c:\test\New_example5.abc*,
 - "*e:\mydata\xyz.abc*" will result into an exported file *e:\mydata\xyz.abc*,
 - "*e:\mydata*.abc*" will result into an exported file *e:\mydata\example.abc*.



Note: If no tool is defined for the export job an Export Error Message window will appear.

- **Transform** button allows a user to set a new origin point, new scale and rotation of all data by the defined angle. See [Export Transforms](#).
- **1:1 Transformation:** This flag indicates whether further scaling is applied in the transformation menu, if scaling is applied checking this box will restore the scale to 1.

Running an Export Job

To run an export job:

- Select the desired export job, then press the  **Run Job** button in the **Jobs** view toolbar.

The run job processing window appears, indicating the progress of each job/task steps.

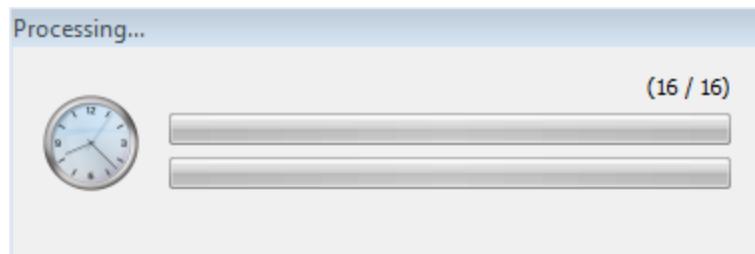


Figure 261. Job processing window

After a job is finished successfully, a confirmation window appears, this window also displays the output path of the exported file, (*the path of the exported file is the same path of the CAM document*);

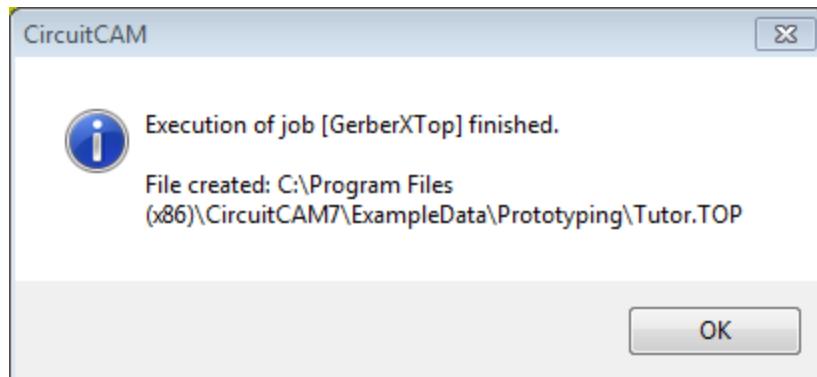


Figure 262. Job finished window

If a job was erroneous; an indication window appears, this window displays the number of warnings and errors that were found during the execution of the job.

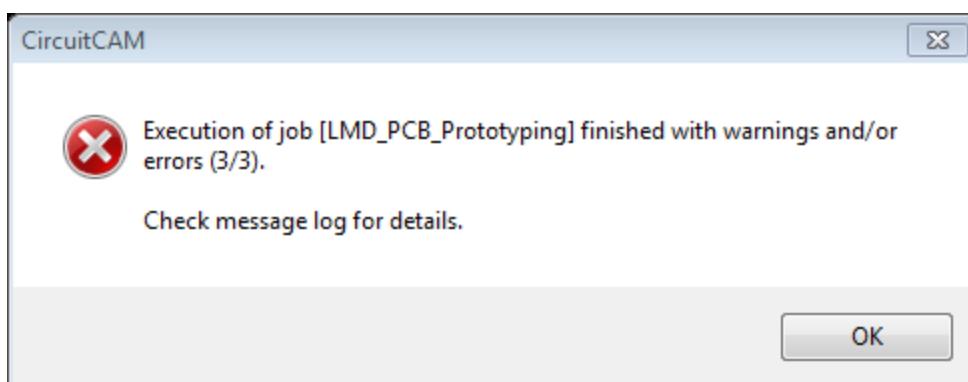
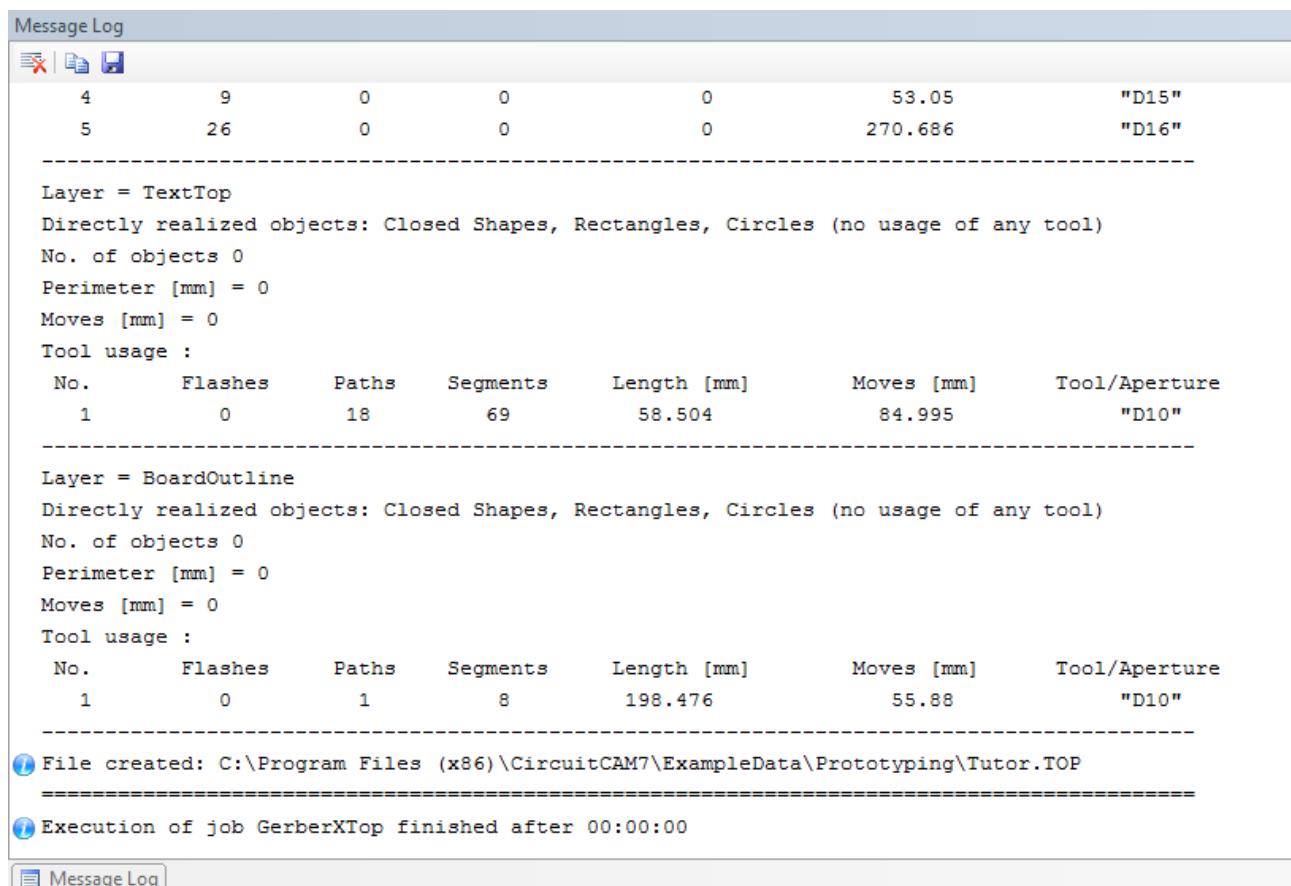


Figure 263. Job finished with errors window

Examples for some of the export errors:

- Inappropriate tool(s) defined.
- Export size (area) not compatible with the design area.

All the job progress is displayed in the [Message Log](#).



The screenshot shows the 'Message Log' window with the following content:

```

Message Log

4         9         0         0         0       53.05      "D15"
5        26         0         0         0     270.686     "D16"

-----
Layer = TextTop
Directly realized objects: Closed Shapes, Rectangles, Circles (no usage of any tool)
No. of objects 0
Perimeter [mm] = 0
Moves [mm] = 0
Tool usage :
  No.    Flashes    Paths    Segments    Length [mm]    Moves [mm]    Tool/Aperture
    1        0        18        69        58.504        84.995      "D10"

-----
Layer = BoardOutline
Directly realized objects: Closed Shapes, Rectangles, Circles (no usage of any tool)
No. of objects 0
Perimeter [mm] = 0
Moves [mm] = 0
Tool usage :
  No.    Flashes    Paths    Segments    Length [mm]    Moves [mm]    Tool/Aperture
    1        0         1         8        198.476        55.88      "D10"

-----
File created: C:\Program Files (x86)\CircuitCAM7\ExampleData\Prototyping\Tutor.TOP
=====
Execution of job GerberXTop finished after 00:00:00

```

Figure 264. The Message Log showing the job execution details

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.0.1.3 Export Task

To open the **Export Task** dialogue window: select an export job under the **Export** node (in the **Jobs** panel) and click left mouse button on it. Press right mouse button on the existing task and press the button or create a new task by using the button . Export task characteristics will be shown.

When having more than one export task in an export job, the different export tasks maybe based on different configurations/tool lists of the same type (e.g.: G-Code, LMD). In this case the format details of configuration of the first task will be used for the whole exported file. For the following tasks only the specified tools/config-items are considered, but not the format details.

Common Export Tasks Properties

- **Layer:** Graphic data on the selected layer will be exported. In the selection list there are filters; "*-- All --*" and "*-- All visible --*". Selecting "*-- All --*", will export all layers, selecting "*-- All visible --*" will export all visible layers.
-  **Note:** When selecting "*-- All --*", empty layers will be ignored, but invisible layers will be exported.
- **Use job transformation:** If *ON*, the current task uses transformation properties of the related job. If *OFF*, the **Transform** button is enabled and a user can set individual transformation for the current task in the [Export - Transform dialogue](#).

Common Export Task Properties except PostScript

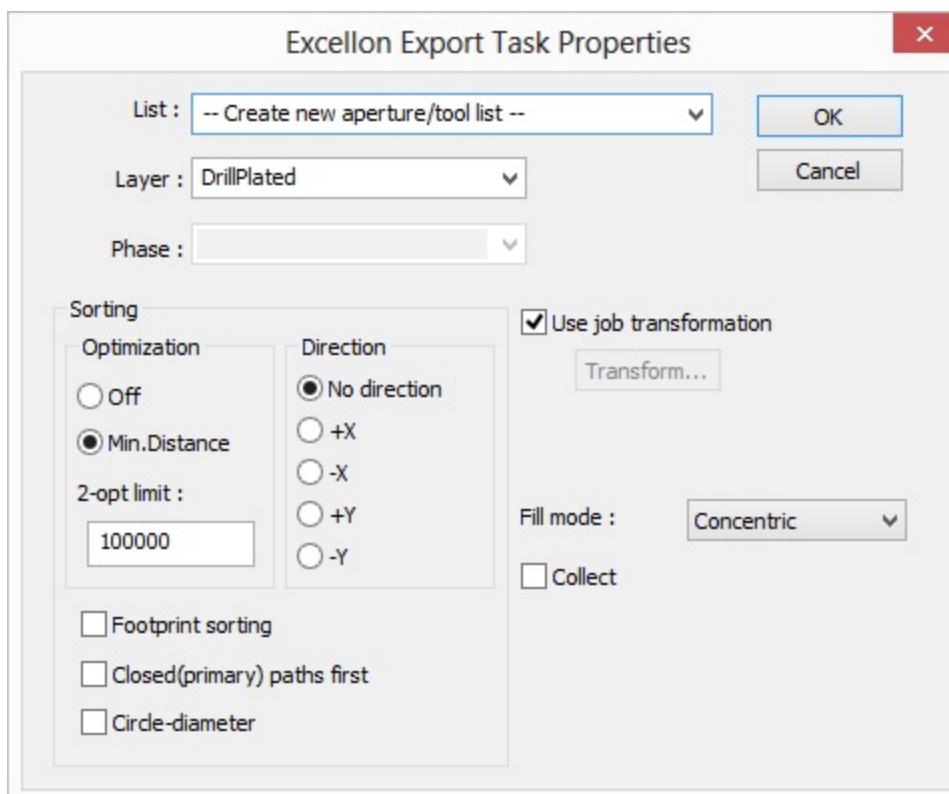


Figure 265. Excellon Export Task dialogue same as (G-code, DXF, Barco)

- **List:** A name of the aperture list, it can be configured for all export jobs except **GerberX** type. Either an existing configuration/tool list is chosen, or a new one is generated, based on the [default configuration properties in general settings](#). Included in the aperture list is the desired export size, in case no size is specified, the size specified in the general settings will be used.
- **Sorting:** A group of flags to define how the order of the output objects will be done. **Direction** specifies the sorting direction. Enable the **Min.Distance** flag to switch-*ON* sorting, flag **Off** is used to switch-*Off* the sorting at all. If flag **Closed (primary) paths first** is enabled, the closed paths have higher priority to compare with the rest of objects. **Circle-diameter** flag groups circles according to the diameter size.

- **Fill mode:** allows filling of closed shapes during an output. The default filling is *concentric*.
- **Collect:** This flag extends a scope of actions on flashes and circles. E.g.:, the flag *Polygon* in *Gerber* export, outputs all flashes and circles as *Gerber* polygons too (with G36 and G37 commands); if arc interpolation is set it is used for flashes and circles too and so on.

Special Properties for GerberX Export Task

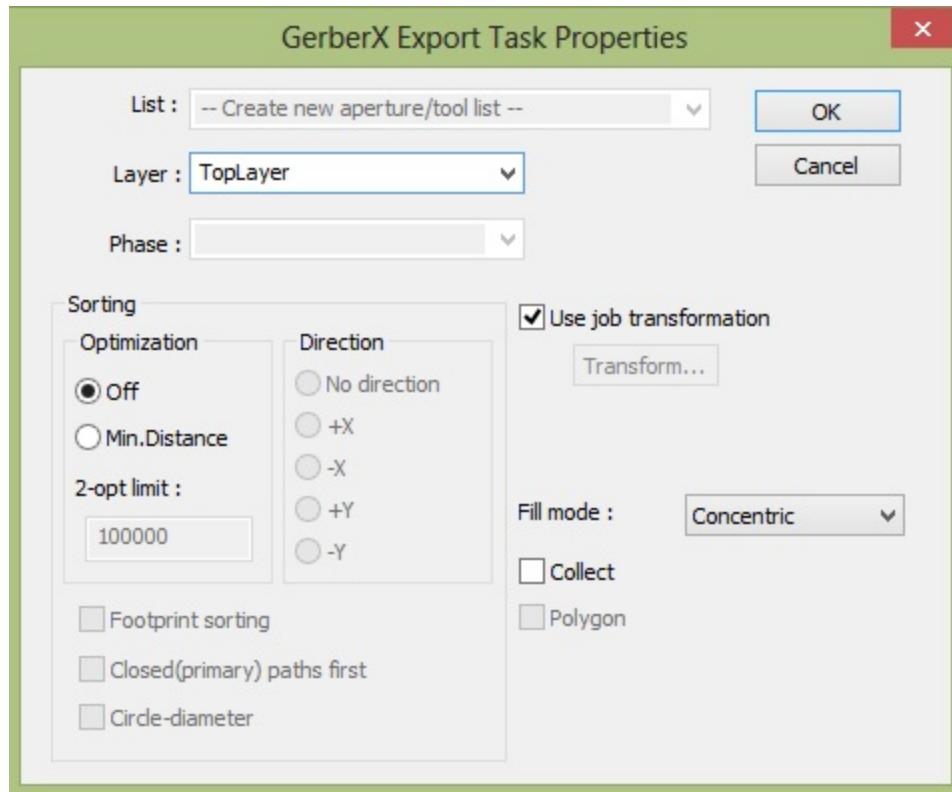


Figure 266. GerberX Export Task dialogue

- **Polygon:** The flag is used only for *Gerber* to output the *Gerber* polygons (with G36 and G37 commands).

Special Properties for HP-GL and LMD Export Task

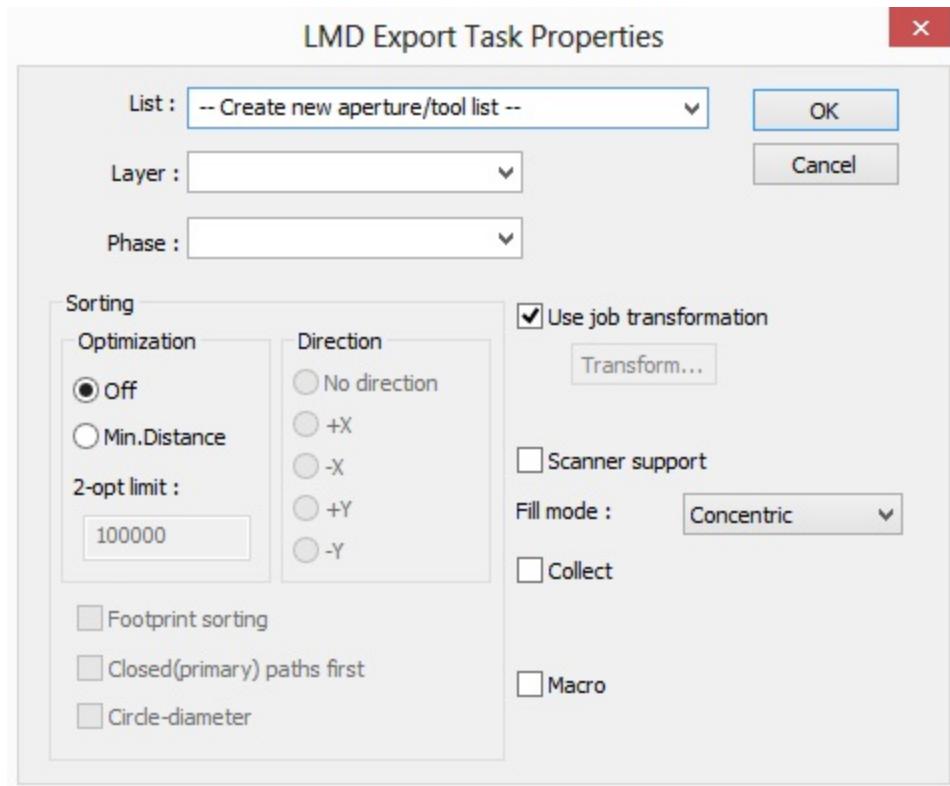


Figure 267. LMD Export Task dialogue

- **Macro:** This LMD-Format specific functionality allows minimizing the amount of data to transfer, by sending a complex configItem only once, and then only sending the transformations for each flash. Main usage of this functionality is for SMT Stencil Laser applications..
- **Scanner Support:** In the case that scanner fields are defined and a flash fits completely within a scanner field, then the flash will become part of the scanner field. If a flash does not fit completely into one scanner field it will be processed outside of the scanner fields.

Special Properties for HP-GL Export Task

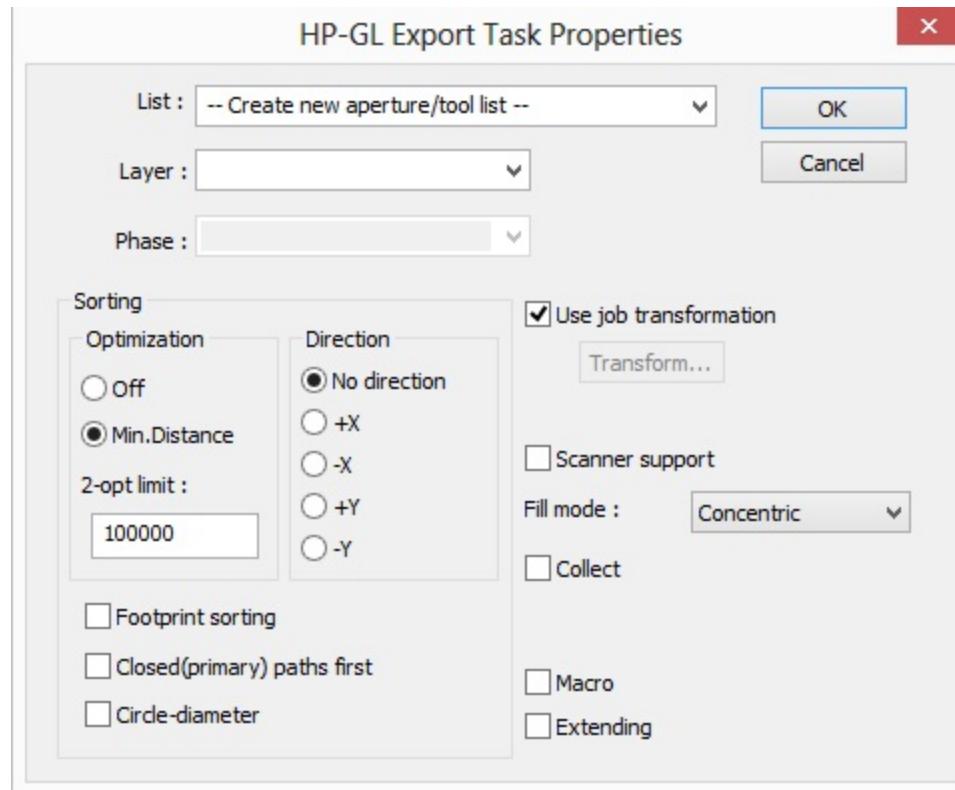


Figure 268. HP-GL Export Task dialogue

- **Extending:** To do swing-in and swing-out. The swing-in and -out path is a short line vector, which is tangential to the start and end-point of open paths. Closed paths, circles and polygons are not supported this way. For the swing-in and -out lines a special artificial tool (PEN) with number (last_used + 1) is used. The distance for swing-in and swing-out (one) is specified in corresponding edit field.

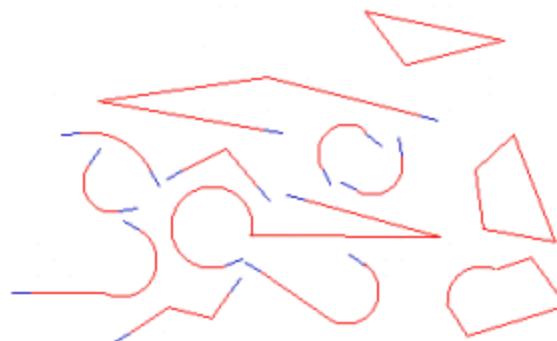


Figure 269. Swing-in and -out paths (blue lines)

Special Properties for PostScript Export Task

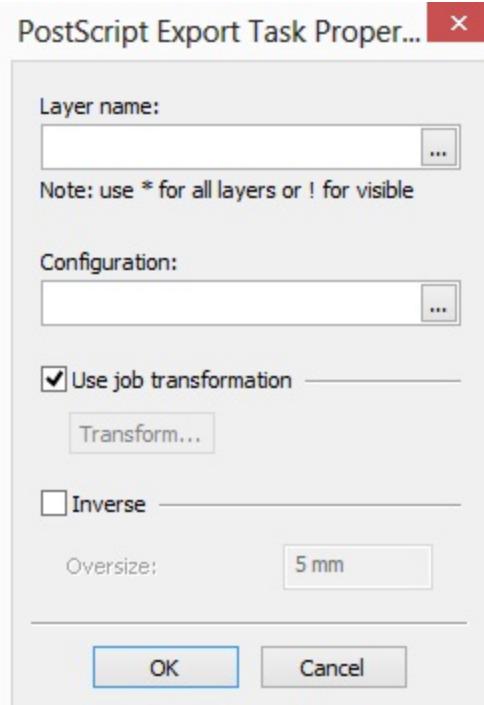


Figure 270. Post Script Export Task dialogue

- **Configuration:** Drop down menu to choose a preset configuration for the export size, in case no configuration chosen, default configuration from the default configurations in the general setting menu is selected
- **Inverse:** If *ON* the **Postscript** layer is exported in negative color way. **Oversize** option is enabled when the **Inverse** flag is *ON*, it create a bounding box based on the value provided.



Note: Typically when working for PostScript, one file per layer is needed. To achieve these several export jobs, each with a single task for one layer is needed.



Note: PostScript files "*.ps" are viewable by applications such as;

- [Adobe Acrobat](http://www.adobe.com/products/acrobat.html) "http://www.adobe.com/products/acrobat.html"
- [GSview](http://www.ghostscript.com/) "http://www.ghostscript.com/"

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.0.1.4 Export Transforms

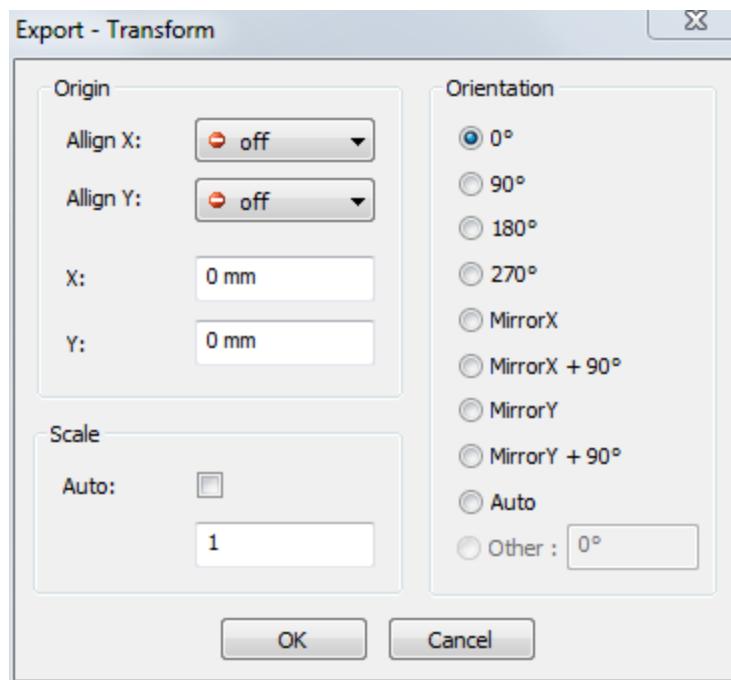


Figure 271. The Export Transform window.

When clicking the **Transform** button in the **Job Properties** or **Task Properties** windows, the above window appears with the following selection groups:

▪ **Orientation Group:**

- **Fixed Angles:** Rotates the orientation of the design on the specified working area with a fixed angle; (0, 90, 180, 270)
- **Mirroring:** Mirrors the orientation of the design either on X-axis or Y-axis.
- **Fixed Angle and Mirroring:** Combines various values of fixed angles and mirroring.
- **Auto:** This selection selects the best orientation for the working area, either normal or rotated 90 degrees.
- **Other:** Rotates the design according to an arbitrary angle; the *Auto* function is only available for LMD, HP-GL, Excellon, and G-Code exports.

▪ **Origin Group:**

- **Align X:** This menu has 4 variants; Off: **No alignment** on X-axis, **left** alignment, **centre** alignment, and **right** alignment.
- **Align Y:** This menu has 4 variants; Off: **No alignment** on Y-axis, **top** alignment, **centre** alignment, and **bottom** alignment.
- **X:** Specifies a value for an offset on X-axis.
- **Y:** Specifies a value for an offset on Y-axis.

▪ **Scale Group:**

- **Auto Checkbox:** Automatically fits the design to the whole working area.
- **Value field:** Specifies the value to scale the design. E.g.: If we enter the value **2**, the size is doubled.

The transformation settings can either be set in the [Job Properties](#), or [Task Properties](#). If a transformation is set in the job properties, this transformation can be applied to all tasks falling under that job. However in the task properties it can be chosen whether to use the general job transformation or specify a special transformation for that task.



Note: For the following formats rectangles are not supported while export job transform is used with a free angle (LMD, HP-GL, Excellon, GCode).



Note: BARCO and DXF formats have no sizes.

Examples on Export Transforms

Transformations

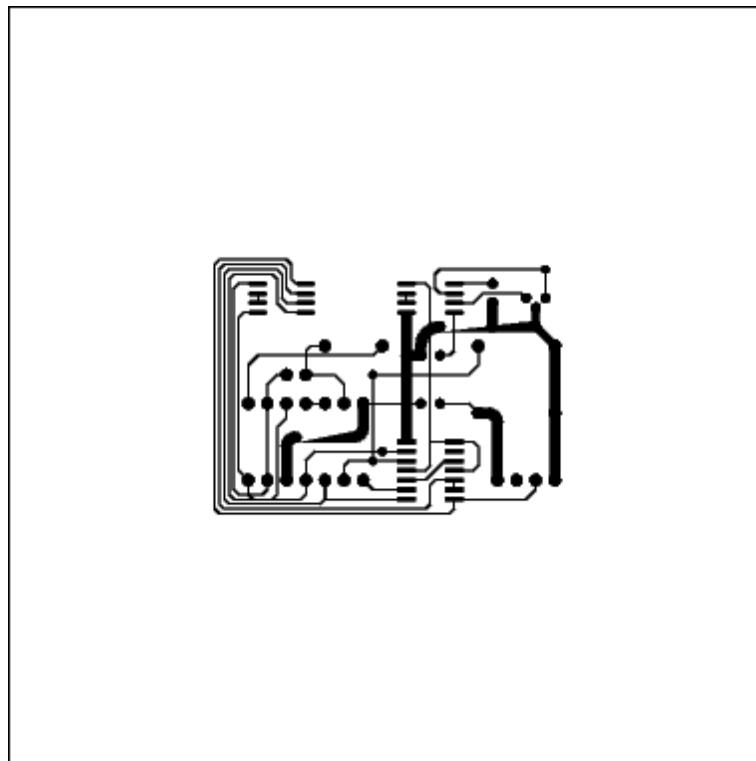


Figure 272. Alignment is set to centre on X and Y axis

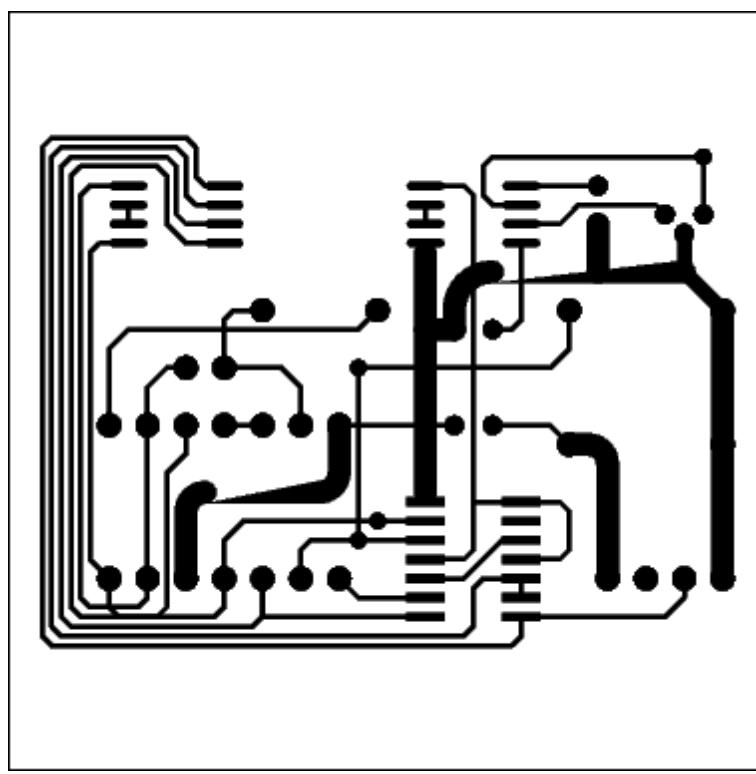


Figure 273. Alignment is set to centre on X and Y axis, and scale is set to "2" - Double

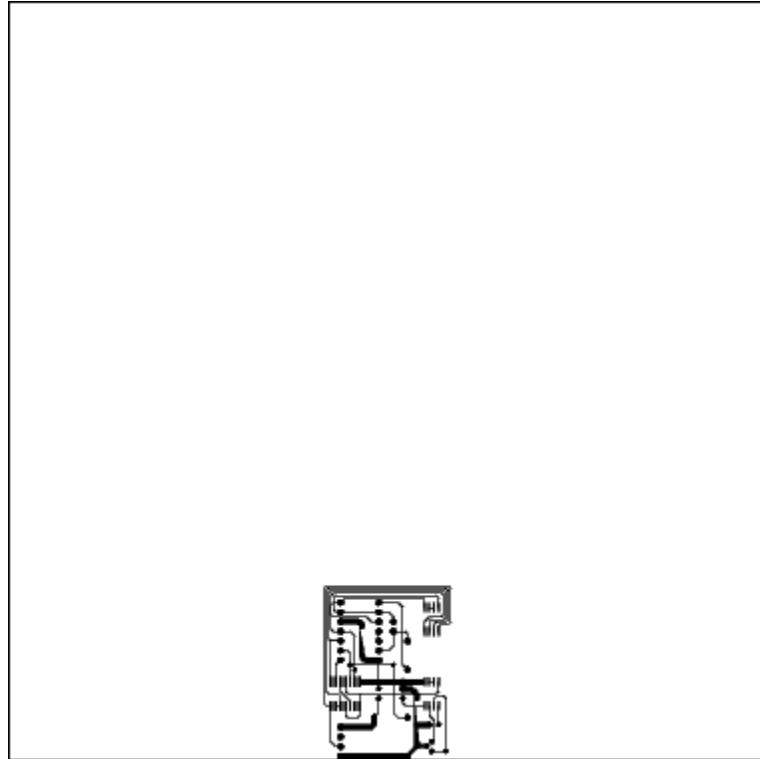


Figure 274. Alignment is set to centre on X axis, and bottom on Y axis, and scale is set to "1/2" Half, and 90 degree rotation

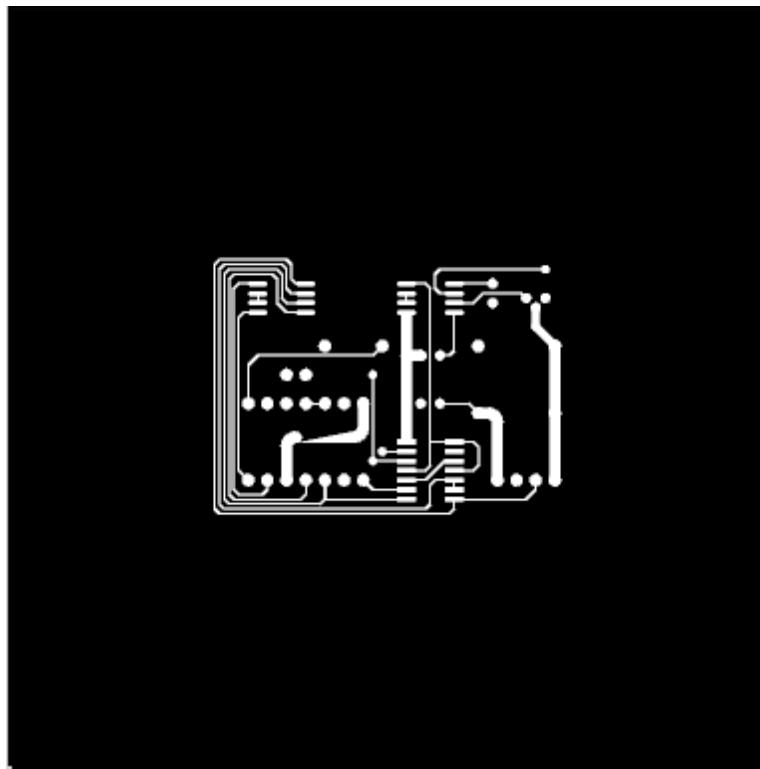


Figure 275. Alignment is set to centre on X and Y axis, and the inverse flag is on

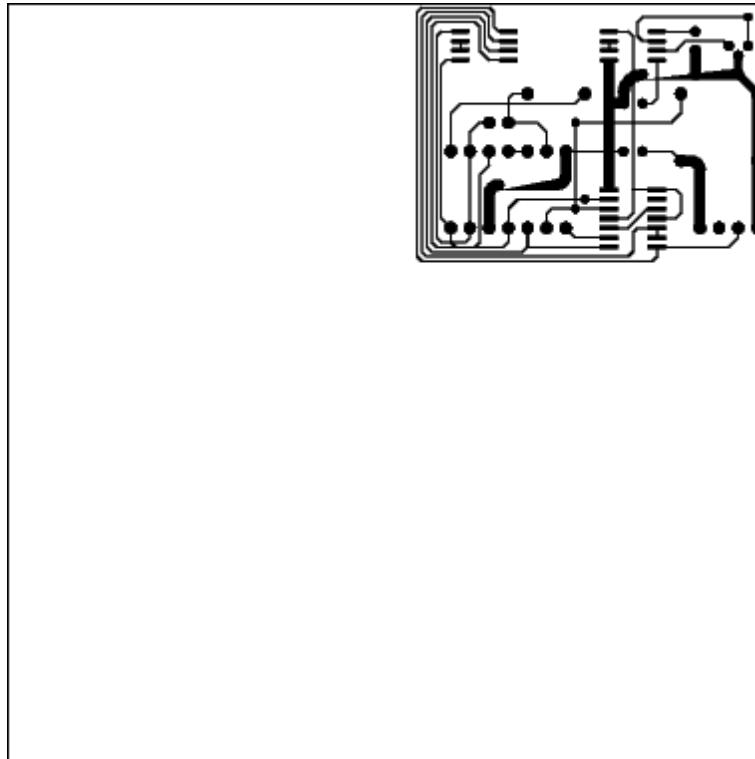


Figure 276. Alignment is set to right on X axis, and centre on Y axis

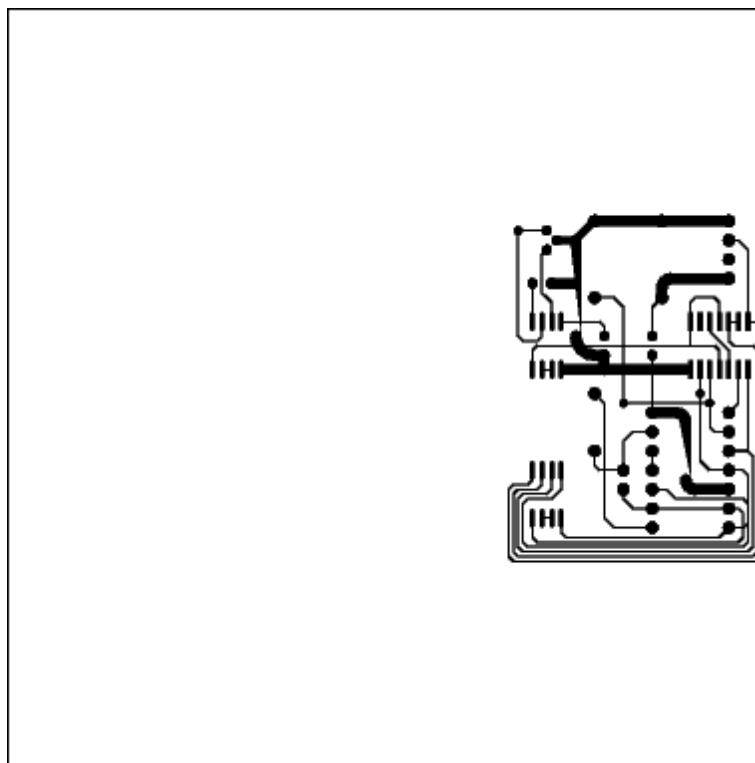


Figure 277. Alignment is set to right on X axis, and centre on Y axis, and rotation is 270 degrees

Auto Origin

For the auto origin function we have an example of a drawn figure, *left* figure shows export without using auto origin function, *right* figure shows export file while using auto function.

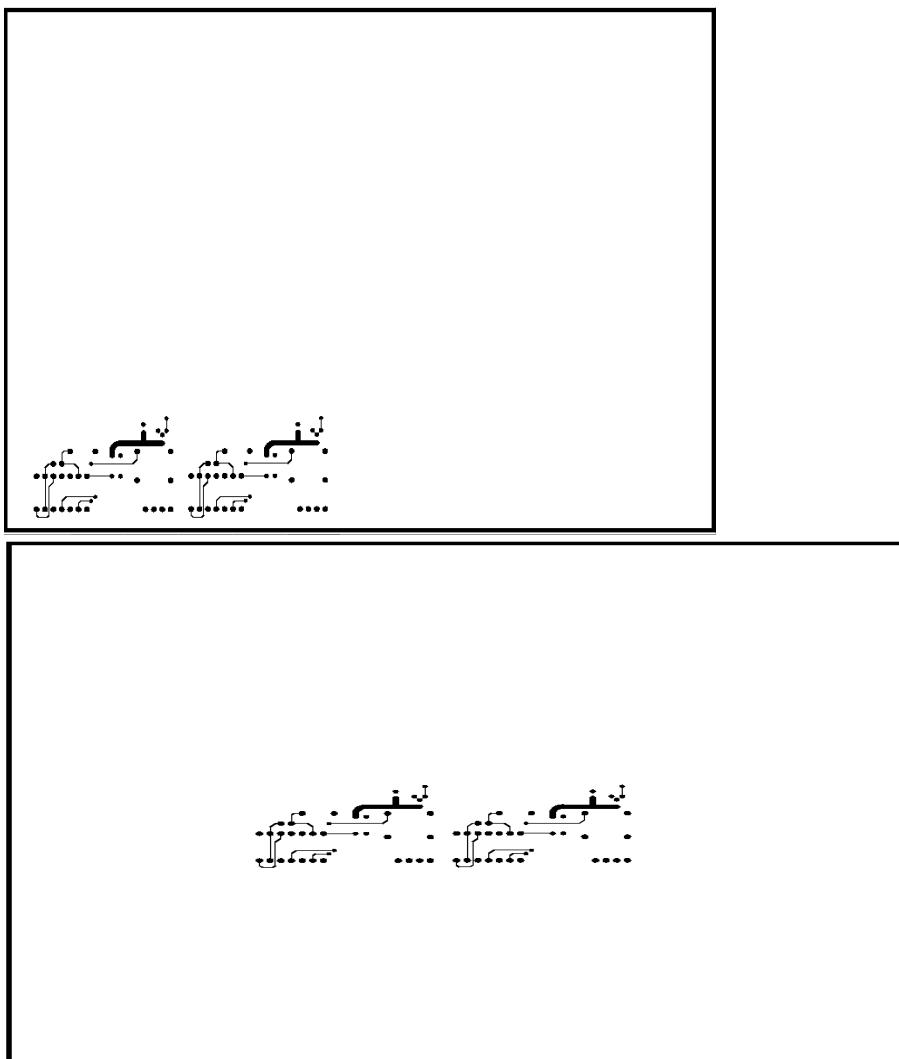


Figure 278. left figure shows the auto origin off and right figure shows the auto origin on

Auto Scale

For the auto scale function we have an example of a drawn figure, **left** figure shows export without using auto scale function, **right** figure shows export file while using auto scale function.

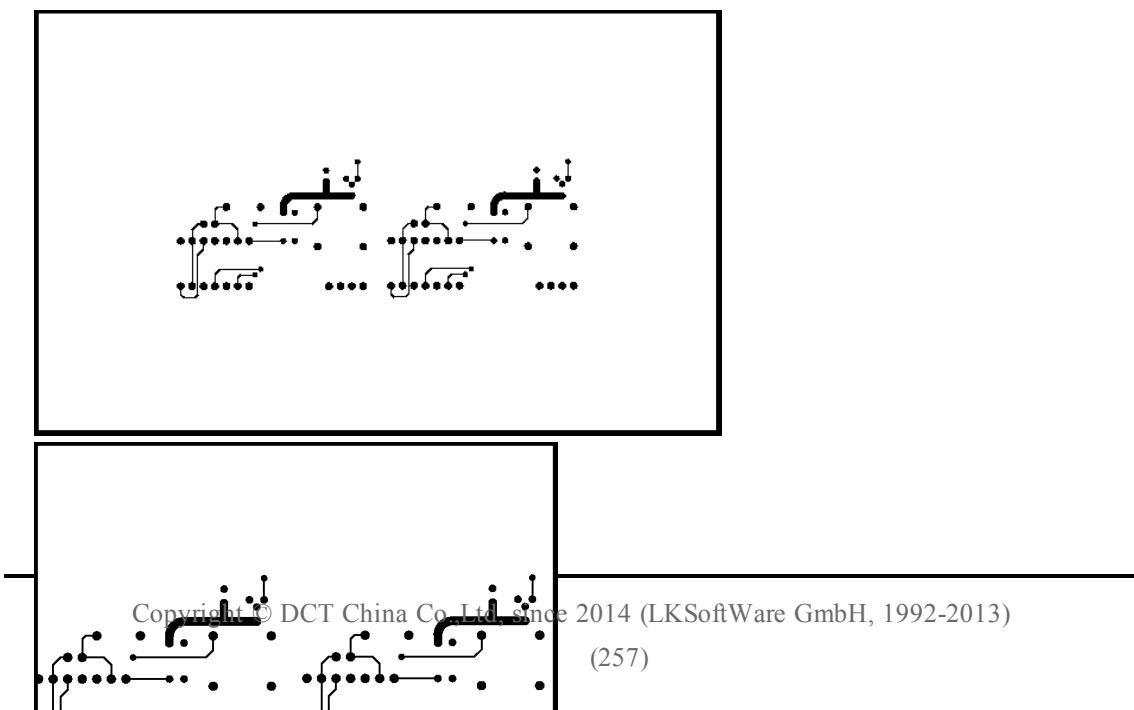


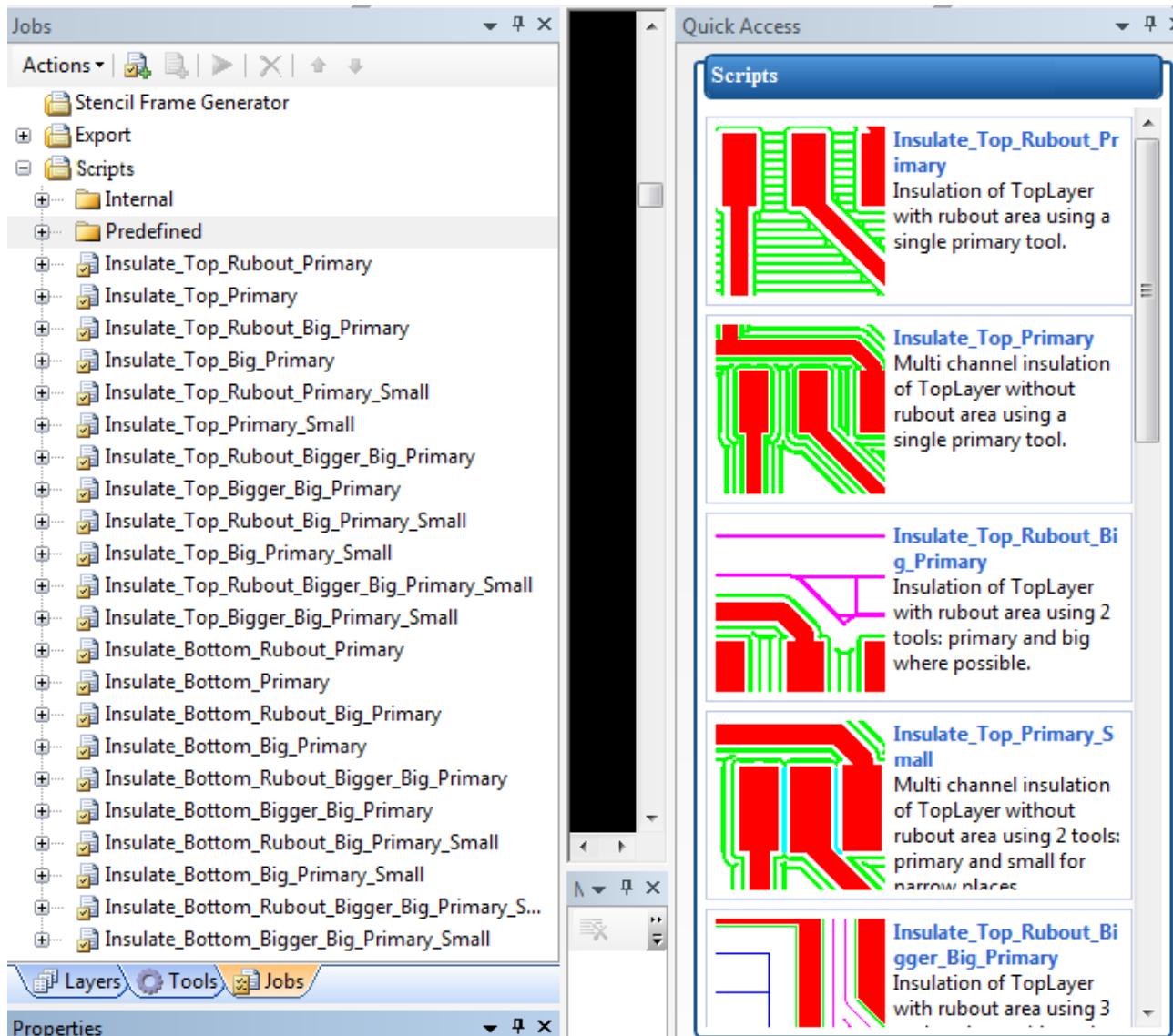
Figure 279. left figure shows the auto scale off and right figure shows the auto scale on

Note: Auto origin function can be selected with the auto scale function.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.0.1.5 Scripts

Scripts are jobs which provide the capability to execute a sequence of tasks without further user intervention. Script commands are either invocations of other jobs or normal tasks. Scripts appear under the **Script** branch in the **Jobsview**. Scripts can be assigned to categories of arbitrary names.

**Figure 280.** Jobs view, listing available Scripts and Quick Access view with included Scripts

Add Script

To create a new Script click on  **Add Script** button in the **Jobs** view toolbar. The **Script Properties** dialogue window appears on the screen.

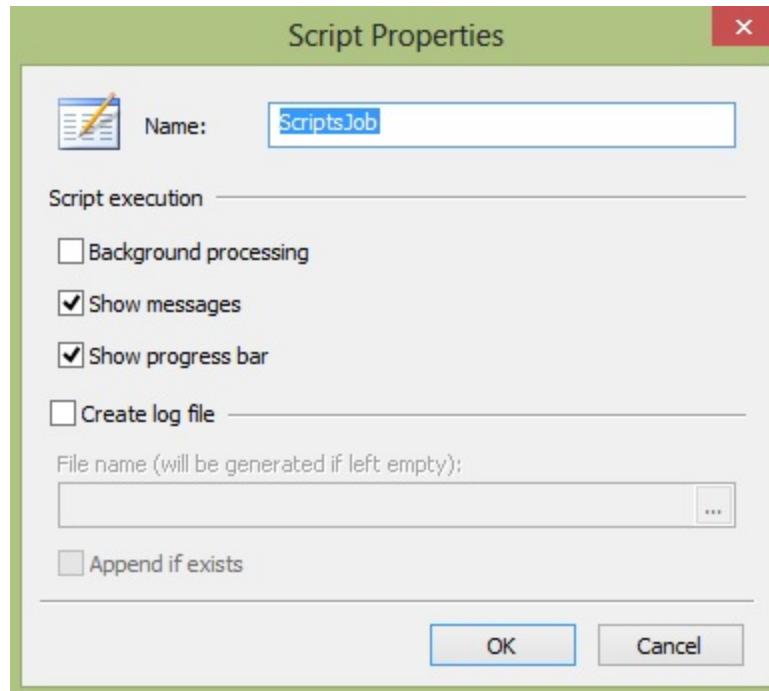


Figure 281. Script Properties dialogue window

Name	Shows/uses the Script name
Background processing	<i>CircuitCAM</i> executes the Script in minimized mode. No messages, any questions come up in the screen (except a file open dialogue for import task in <i>Open File dialogue</i> mode).
Show messages	If <i>Off</i> , no messages, any questions come up in the screen (except a file open dialogue for import task in <i>Open File dialogue</i> mode).
Show progress bar	If <i>Off</i> , the progress bar does not show.
Create log file	If <i>On</i> , creates a log file in a specified place.
Append if exists	If <i>On</i> and a log file already exists with the same name, it appends the info at the end.

Multiple export tasks can be grouped by choosing the option make Script from selection, this option allows the user to choose different script jobs to be run at once, also can be added to [Quick Access](#).



Note: If warning or error message appeared for one of the grouped scripts, it will be shown immediately and referred for later to be viewed in the [Message Log](#), to disable the warning or error message uncheck the show message flag in the [Quick Access](#) properties.

Script Properties

To see the current parameters of a particular job, select the job or press right mouse button and select **Properties** menu item. The **Properties** view with detailed data of the selected job appears on the left bottom section of the *CircuitCAM* window.

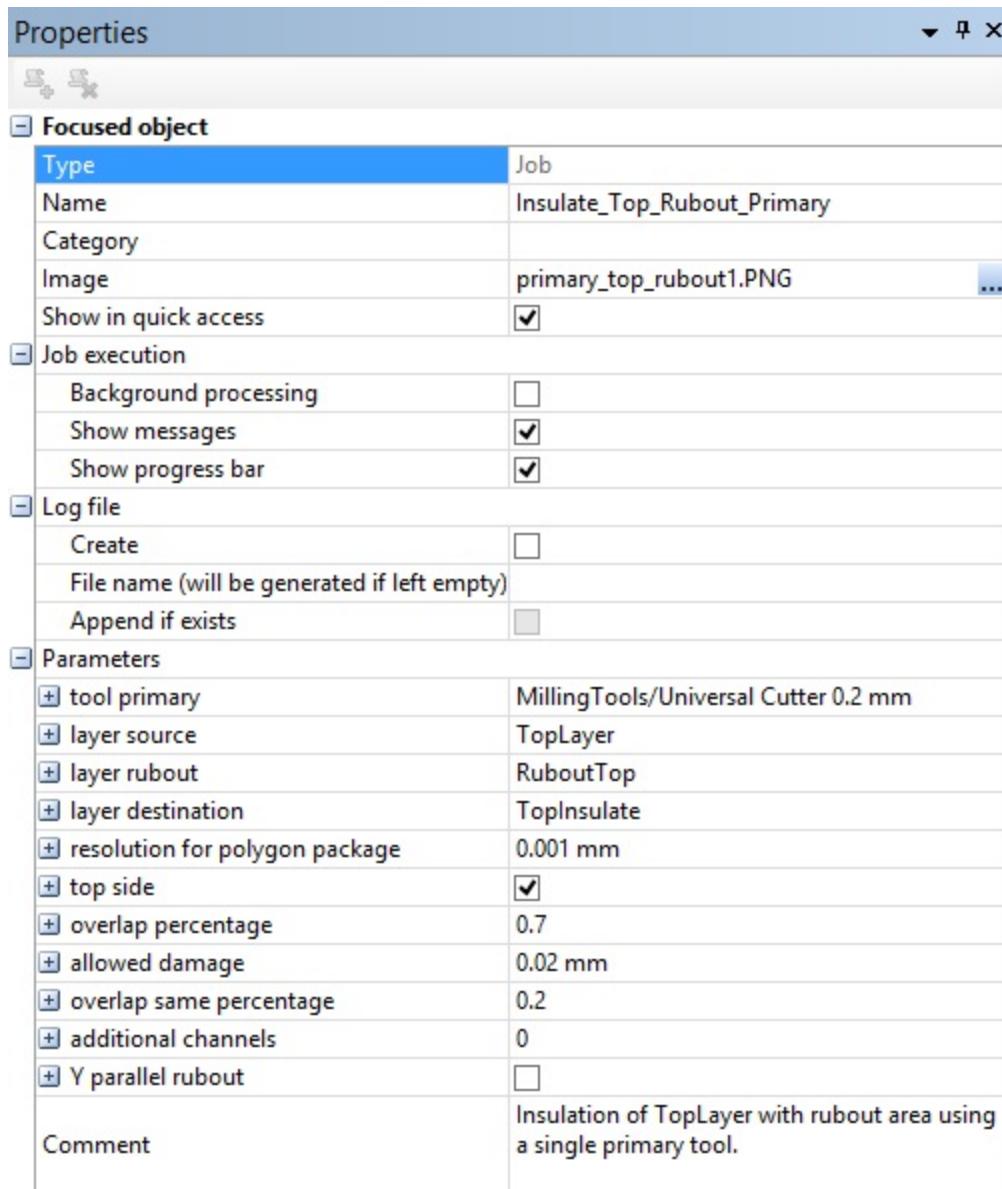


Figure 282. Properties view of selected Script

Focused object		
	Type	Specifies the type of the focused object - <i>Job</i> .
	Name	Shows/uses the Script name.
	Category	Specifies the category the selected job belongs to.
	Image	A static picture that is shown along with the Script in quick access that should illustrate the capabilities of the script. All pictures are stored in the sub-folder img under the installation directory,

		e.g.: C:\Program Files (x86)\CircuitCAM7\img. User can place here further pictures as needed. when pressing on "..." an open dialogue shows up to choose a new picture for the current job. The recommended size of images is 300 x 300 pixels in PNG¹ format.
	Show in Quick Access	If <i>On</i> the <i>Job</i> will be viewed and added in the <i>Quick Access View</i>
Job execution		
	Background processing	<i>CircuitCAM</i> executes the Script in minimized mode. No messages will display, any questions come up in the screen (except a file open dialogue for import task in <i>Open File</i> dialogue mode).
	Show messages	If <i>Off</i> , no messages, any questions come up in the screen (except a file open dialogue for import task in <i>Open File</i> dialogue mode).
	Show progress bar	If <i>Off</i> , the progress bar does not show.
Log file		
	Create	If <i>On</i> , creates a log file in specified place.
	File name (will be generated if left empty)	Specifies the name and the location of the log file.
	Append if exists	If <i>On</i> and a log file already exists with the same name, it appends the info at the end.
Parameters		Parameters for a particular job.

Adding Scripts to quick access

A Script is selected, then select an image file to be assigned to the Script by clicking on image tab found in properties panel (See below; This image folder is found in the Program files\ CircuitCam7 \img), In the comment tab user may write a comment to be reminder of the Script descriptions.

The Quick access view displays predefined and user defined Scripts for direct execution. Each entry in the quick access view consist of:

- Name of the script
- The description of the script
- A picture indicating the script function

¹A bitmap image file format that supports lossless data compression

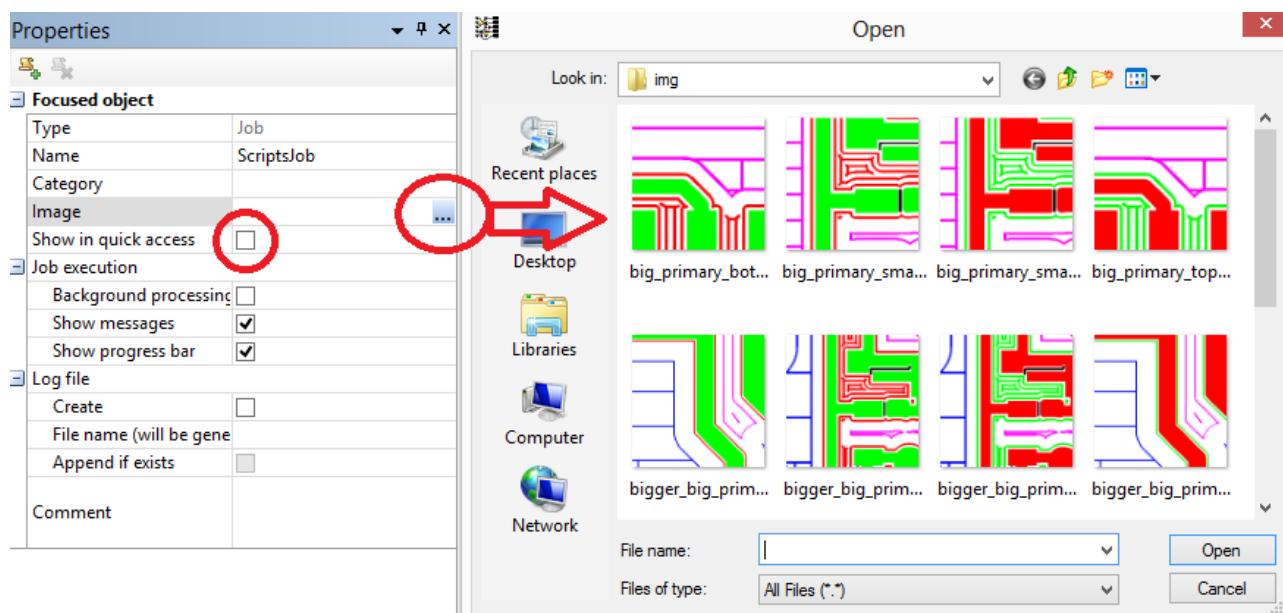


Figure 283. Assigning image to the corresponding job

The image chosen in the property panel will appear in the quick access panel along with the comment that was added in the properties panel.



Note: If no picture is chosen, only the name of the Script will appear.



Note: If no comment is added nothing will appear with Script name.

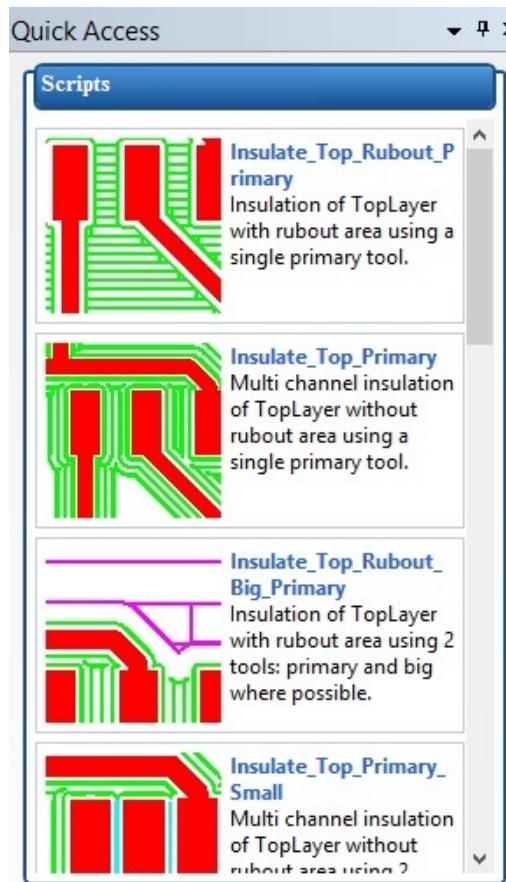


Figure 284. Script added with image and comment is written below the job name

Run Script

To run a Script:

- Select the desired script, then press the  **Run Job** button in the **Jobs** view toolbar.
- Or
- Press on an applicable job in the **Quick Access** view.

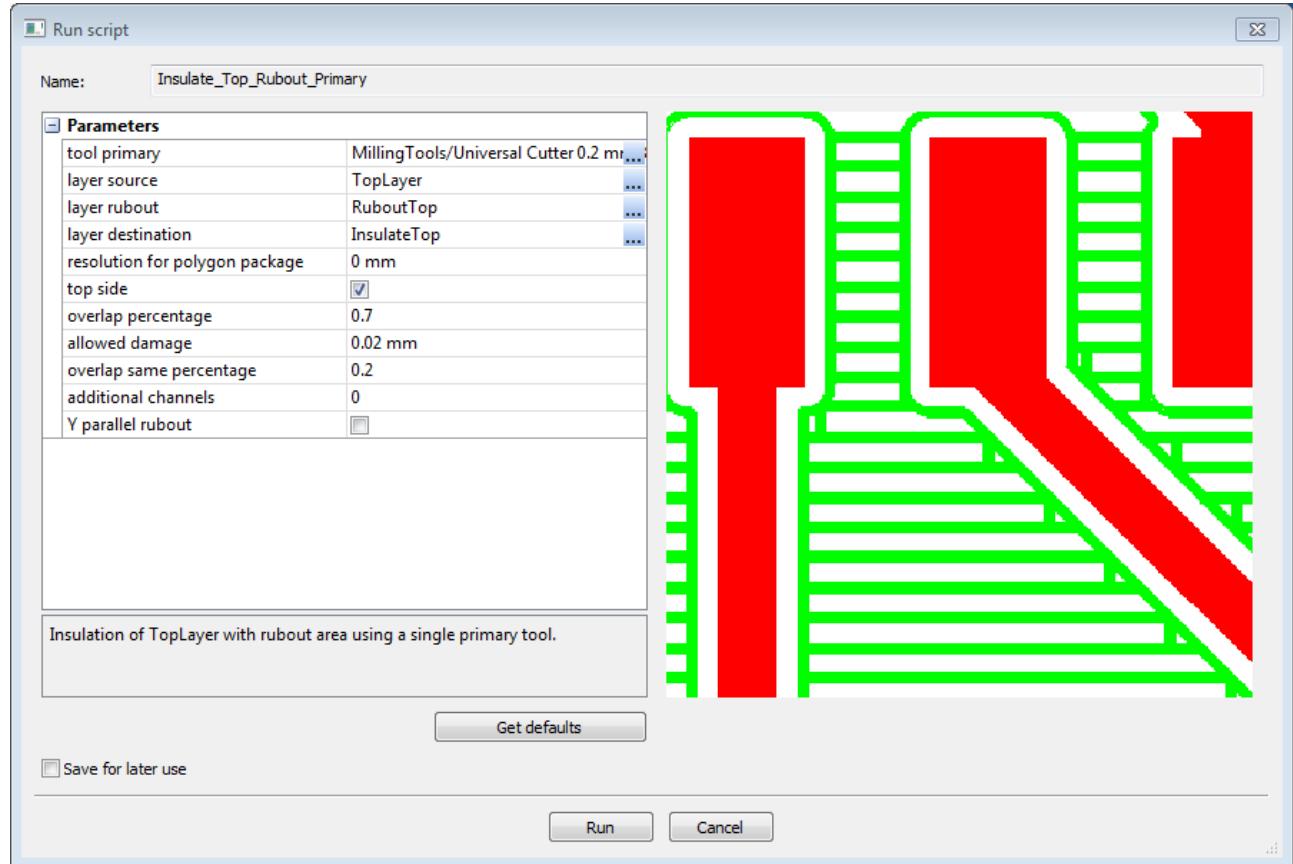


Figure 285. Run Script dialogue window

The **Run** job dialogue window appears on the screen. For each parameter a user can specify **Name**, **Type**, **Value**, and **Comment**.

Name is a sequence of any alphabetical characters.

Type of parameter can be selected from the list of supported types in *CircuitCAM*. The types of parameters are:

- String,
- Integer,
- Real number,
- Layer name,
- Tool name - it is used to specify a tool name with or without a tool list name in the front, e.g.: "MillingTools/Universal Cutter 0.2 mm" - "MillingTools" is tool list name and "RF Cutter 0.25 mm" is a tool name.
- Length - a distance in units, specified in *CircuitCAM* settings.

Value: Value is dependent on the specified type of parameter.

Comment: This field is used to describe the current parameter (meaning, usage, restrictions and etc.). This comment will be displayed in parameters dialogue window which is displayed before job execution. Values that can be added to the comment field can be; free text, single parameters, e.g.: "*Layer \$Layer_name is empty.*" or "*Layer \$src_layer was copied to \$dst_layer.*"

Save for later use: When checking the "Save for Later Use" box, the chosen parameters will be saved within the CAM file so the user can apply them again later. If the box is not checked, the changed parameters will only be used for this single execution of the script.

Get Defaults button: If the user wants to get the original parameters of the script, clicking the **Get Defaults** button will do the following; check the underlying template file ***.CAT**, and compare the current values and restore the original values.

Run Script Parameters

The following parameters are from the script "**Insulate_Top_Rubout_Primary**", the included parameters may differ from one script to another;

- **Tool Primary:** The primary that will be used to mill the circuit board.
- **Layer Source:** Specifies the source layer for the insulation.
- **Layer Rubout:** Graphical objects on this layer define additional areas which have to be cleared by the isolation process.
- **Layer Destination:** Specifies the layer for output data.
- **Resolution for Polygon Package:** Specifies the grid to be used.
- **Top Side:** Specifies whether the job will be executed on the top or bottom side of the circuit board.
- **Overlap Percentage:** Specifies the overlap between two insulation lines created by different tools.
- **Allowed Damage:**
- **Overlap Same Percentage:** Specifies the overlap between two insulation lines created by the same tool.
- **Additional Channels:** Number of additional insulation channels to be created around pads and tracks.
- **Y-Parallel Rubout:** Directions of serpentines.

Make Derived Script

This functionality allows creating a new derived script based on a selected script that already exists. For this all parameters and their assigned values are copied, but the content of the selected script is not copied. Instead of this a "**Call job script task**" which calls the selected script is created. After the parameter values are modified according to the needs, a new script is saved under the desired name.

When selecting "**Make Derived**" from the context menu, the following window appears; this window is similar to the run script dialogue but it allows creation of a new script based on the modified parameters.

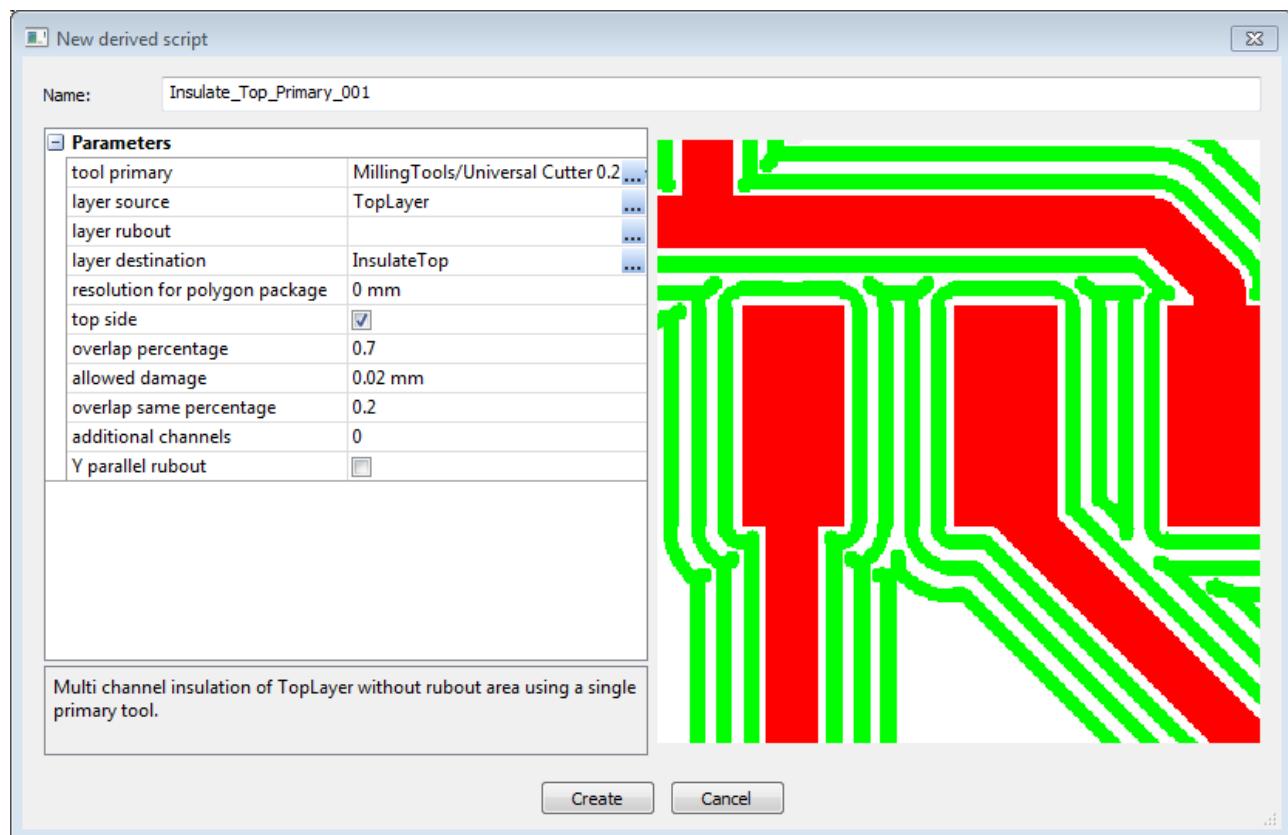


Figure 286. New Derived Script Dialogue

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.0.1.6 Script Tasks

Add Script command

To add a new task in the Script, press **Add Task** button either from the **Jobs** view own toolbar or context sensitive menu. Select a task from the tasks list and press **OK** button:

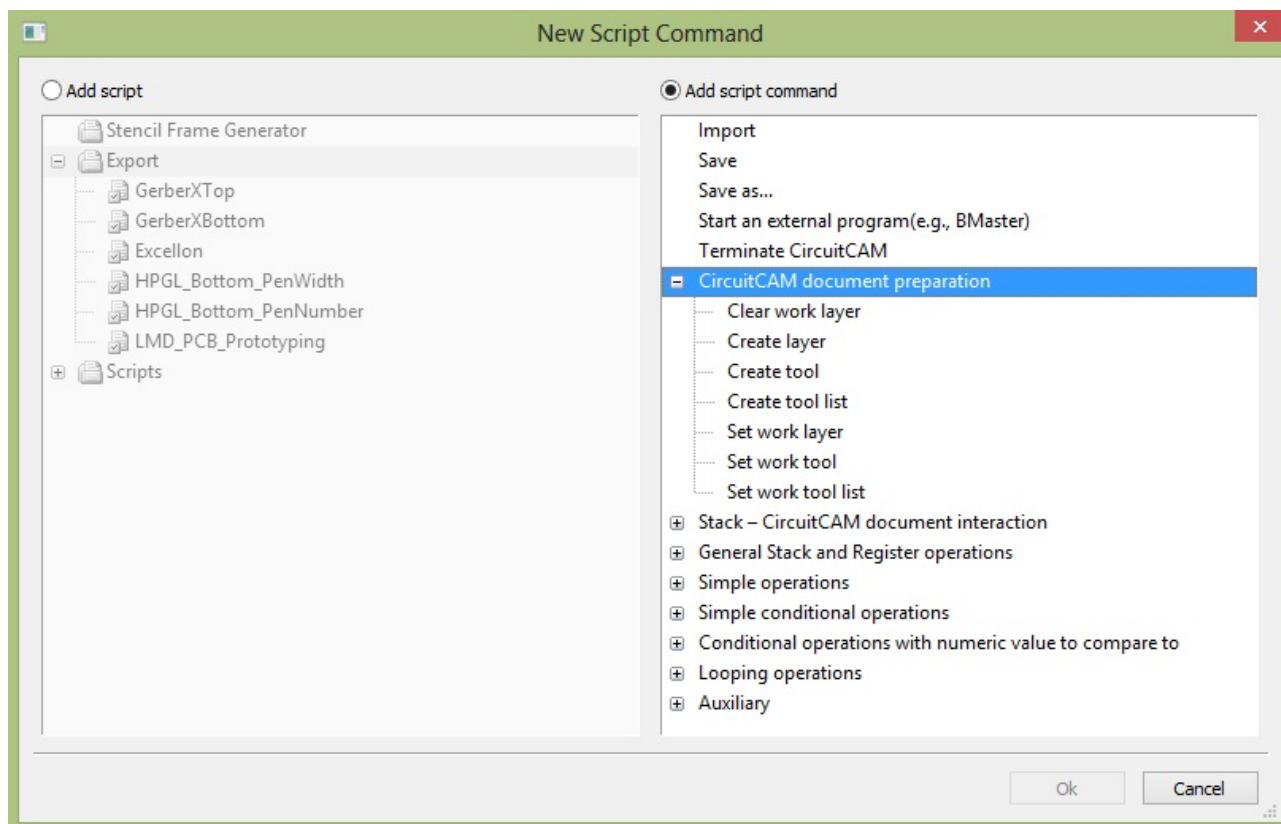


Figure 287. New Script command dialogue window

There are two types of the tasks in the tasks list - **Jobs** and **non-Job actions**.

- In **Add job**, group you can see all predefined *CircuitCAM* job groups (**SMT Stencil Frame generator**, **Export**, **Script**) and jobs, which are created in the current CAM document. Export task additionally saves output file by the name, given by *Save* or *Save As* commands. If there were no such tasks in the job, it opens the standard *Save As* dialogue, where the last imported file is suggested.
- **Add non-job action** group allows the user to add a new task as a function, which has no execution jobs, for example *Import*, *Create Outline*.

Script command Parameters

To see the current parameters of a particular task, click on a task or press right mouse button and select **Properties** menu item, the particular **Properties** view will appear.



Note: The **Properties** window differs according to the type of the task.

Any task has the following properties: **type**, **name**, and a **comment** property. The string in the **name** property is displayed in the **Jobs** view. If it is empty, the type of the operation is displayed instead.

Some tasks have some additional parameter fields. For example, the **Oversize** task, from the Script commands group has these fields:

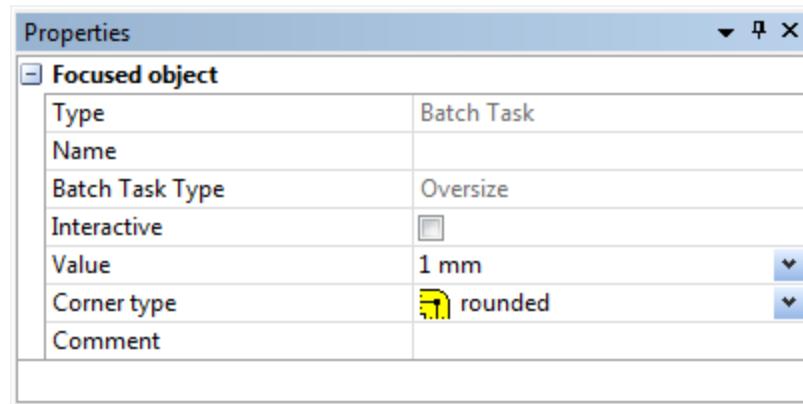


Figure 288. Properties view, Script command of type Oversize properties

Value	A positive value for stretching, or a negative one for constraining the objects in the stack.
Corner type	Combo-box to select one of proposed types - rounded (Rounded), sharp (Extended) or cut (Truncate).
Interactive	Allows interactively set oversize parameter via input dialogue. Otherwise (if this flag is <i>Off</i>) takes oversize value from the Value field.

Changing of property values is also done through the **Properties** view.

Available script commands

Import

Imports selected files to CircuitCAM.

Settings:

- a) **File to import.** This parameter can be set to "Open file dialog" or "From command line" mode.

If "Open file dialog" is set, the file selection dialog box is opened, otherwise the file(s) for the import must be specified as command parameters.

- b) **File type recognition.** This parameter can be set to "By import assignment" or "Automatic".

"By import assignment" value indicates that CircuitCAM will use "Import assignments" settings to detect types of input files. If CircuitCAM will not be able to detect all file types - import dialog box will be shown.

"Automatic" value indicates that CircuitCAM will use internal automatic recognition mode to detect types of input files. If CircuitCAM will not be able to detect all file types - import dialog box will be shown.

Save

Saves current document.

Settings:

None.

Save As

Saves current document with different name.

Settings:

a) **Interactive**. This parameter has two states - ON or OFF.

If value is "ON", the "Save as..." dialog will be shown.

If value is "OFF", value from "File name" as new name.

b) **File name**. This parameter is enabled only if "Interactive" parameter is OFF.

Full path and file name must be specified.

Start an external Program

Runs external program.

Settings:

a) **Program name**. This parameter holds the full path to the executable file.

b) **Parameters**. This parameter holds parameters for the specified program.

c) **Hidden**.

d) **Hold script until program closes**. This parameter has two states - ON or OFF.

Terminate CircuitCAM

Stops CircuitCAM execution.

Settings:

None.

see also script commands of the [Configurable Geometry Engine](#)

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.0.1.7 Configurable Geometry Engine

CircuitCAM is able to perform complex operations on graphic objects by the so-called "**Set Operations**", also known as **2D** “Constructive Solid Geometry” or **CSG**¹ operations. In *CircuitCAM* the classical **CSG** operations are extended in various ways to include operations on areas and paths, and to support infinite areas and more.

A **Polygon/Path** variable can contain either a **Polygon** or a **Path**. A **Polygon** can be completely empty or completely full (infinite flag), and it can contain any number of non-intersecting closed inner and outer boundaries which consist of line and circular arc segments. An outer boundary defines the outline of an area. An inner boundary defines a hole within an area. A **Path** can contain zero, one or many single paths, each consisting of any sequence of line and circular arc segments. A single path may be closed or open; in the latter case it has a dedicated start and end point.

¹Constructive Solid Geometry

Polygon/Path variables can be stored either on the stack or in a register. All operations that consumes, generate or modify **Polygon/Paths** work on the stack; similar to the programming language **Forth**, see [http://en.wikipedia.org/wiki/Forth_\(programming_language\)](http://en.wikipedia.org/wiki/Forth_(programming_language)).

The stack can contain zero, one or more **Polygon/Path** variables. The Register contains a fixed set of 21 **Path/Polygon** variables, numbered from 0 to 20. *CircuitCAM* ensures that at the beginning of each **Script** the register variables are empty, the stack however may be filled by a previous manual **Set Operation**.

Like all other Script commands, the **Stack Operations** are executed in the sequence in which they are defined. Conditional and looping operations contain inner **Stack Operations** that are only executed when a defined condition is met.

Stack Operations are created through the **New Script command** dialogue. Once created, the properties (also called parameters) of the selected **Stack Operations** are displayed in the **Properties** view. Changes of property values are made through the **Properties** view. All **Stack Operations** have a name and a comment property. The string in the name property is displayed in the **Jobs** view. If it is empty, the type of the **Stack Operation** is displayed instead. Some **Stack Operations** require further properties, indicated below.

CircuitCAM Document Preparation

Clear work Layer

Removes all objects from the active **Work** layer.

Create Layer

Creates a new layer (if not exist already) and makes it the **Work** layer.

Create Tool

Creates a new tool (if not exist already) and makes it the **Work** tool.

Create Tool List

Creates a new tool list (if not exist already) and makes it the **Work** tool list.

If layer exist

Checks if layer with specified layer name exist in current CircuitCAM document.

Set Work Layer

Sets the new working layer for follow up operations. Before the first invocation of this operation, all *Stack – CircuitCAM document interaction* operations (e.g., **Move to Stack**, **Copy to Stack**, ...) will work on the selected graphic objects, afterwards - on the objects of the work layer.

Set Work Tool

Sets a new working tool for follow up operations. Before the first invocation of this operation the currently active tool is used.

Set Work Tool List

Sets a new working tool list for follow up operations. Before the first invocation of this operation, the currently active tool list is used.

Stack - CircuitCAM Document Interaction

Move to Stack

Calculates the merged outline(s) (outer and inner) of the selected graphic object(s), and *pushes* it/them onto the **Polygon** stack. The graphic objects from **Work** layer are removed.

Move to Stack as Path

Calculates the merged outline(s) (outer and inner) of the selected graphic object(s), and *pushes* it/them onto the **Path** stack. The graphic objects from **Work** layer are removed.

Copy to Stack

Calculates the merged outline(s) (outer and inner) of the selected graphic object(s), and *pushes* it/them onto the **Polygon** stack. The graphic objects remain on **Work** layer.

Copy to Stack as Path

Calculates the merged outline(s) (outer and inner) of the selected graphic object(s), and *pushes* it/them onto the **Path** stack. The graphic objects remain on **Work** layer.

Pull from Stack

Pulls the result of a set operation from the **Polygon** stack. The result of this action is always presented in the form of the *CircuitCAM* polygons (even if no operations were performed on the Polygon stack). Resulting **PP-Polygon** will be placed on **Work** layer in **CAM** polygon and path.

Pull from Stack as Path

Pulls the result of a set operation from the **Path** stack. The result of this action is always presented in the form of the *CircuitCAM* paths (even if no operations were performed on the Path stack). Resulting **PP-Path** will be placed on **Work** layer in **CAM** path.

Pop Stack

Pulls out and destroys last element from the stack.

General Stack and Register Operations

Clear Register

Clears the **Register**.

Clear Registers

Clears all the **Registers**.

Clear Stack

Clears the **Polygon/Path** stack.

Copy Register to Stack

Copy the **Register** to the **Polygon/Path** stack.

Copy Stack to Register

Copies the **Polygon/Path** stack to the **Register**.

Create Polygon

Creates empty polygon (an infinity for a new polygon is set via parameter) and pushes it to the stack.

Duplicate

Makes a copy of the object on the stack and pushes it onto the stack.

If Register Has Value

Checks if specified Register has **Polygon/Path**.

Move Register to Stack

Moves the **Register** to the **Polygon/Path** stack.

Move Stack to Register

Moves the **Polygon/Path** stack to the **Register**.

Swap

Replaces the content of the top two stack operations.

Simple Polygon/Path Operations

Combine Closed Path

Combine a set of neighbour **Paths** into the single path. Initially this task checks and closes all open paths in incoming set of **Paths**. The distances between closed paths are checked and paths are combined if this distance is less than the value specified by the user.

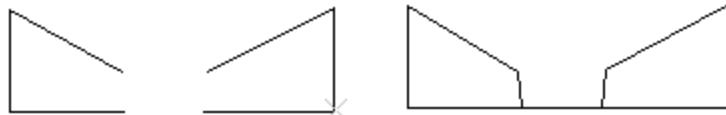


Figure 289. Combine closed path initial data (left) and result (right)

Combine Path

Combines the ends of single paths if the distance between the two ends is smaller than a specified value. Depending of the mode, the direction of single **Paths** may be inverted while doing so.

Covert to Path

Convert **Polygons** to **Paths**.

Convert to Polygon

Convert **Paths** to **Polygons** with the given thickness and corner/end types.

Create mark

Creates circular polygon on the stack with specified diameter parameter at the beginning of the path which must exist on the stack.

Create Needle from Concentric

The first **Polygon** on stack is used as base, the second **Polygon(s)** are used for needle ends.

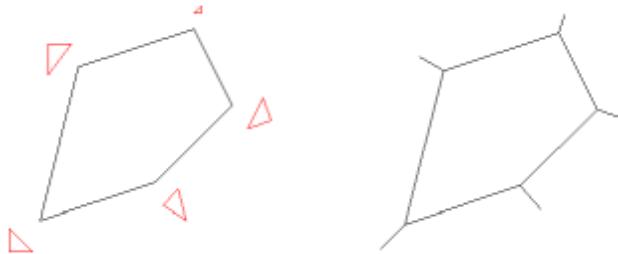


Figure 290. Create needle from concentric initial data (left) and result (right)

Centre points of needle (*Secondary*) polygons are calculated and connected with the nearest points on base polygon.

Create Serpentine

Creates serpentine milling tracks. The direction of milling tracks can be set by the *X-parallel* parameter, (when *On* – horizontally oriented milling paths will be created, when *Off* – vertically). The distance between parallel lines is defined in the field "Distance between scan-lines".

Cut Path at Path

The first **Path** initially could be closed path. After *Cut path at path*, it becomes open.

Cut Path at Polygon

Cuts the **Path** at the position, indicated by the **Polygon**. A path initially could be a closed path. After *Cut path at polygon* it becomes open.

Cut Stripes

Cuts the **Path** at the position, indicated by the **Path**.

Denest Polygon Cutout

The cutouts (inner figures) are extracted out of polygon cutout and become separated areas.

Difference

The top object on a stack is subtracted from the second object on a stack. The top object must always be a **Polygon**. The second object and the result is either a **Polygon** or a **Path**.

Extend path to nearest object

Extends start/end point of the first **Path** to the second **Path**. **Distance** parameter must not be exceeded while extended points.

Find common path

Finds common borders of two **Polygons** on the stack.

Find Lead-in

The first **Polygon** is a contour, the second **Polygon** is the basis (undersized).

The result is a **Path** from the basis contour.

Parameters for this task:

- *LeadIn Distance* – distance of Lead-In path;
- *Angle* – defines angle between LeadIn Path and contour;
- *Counter clock-wise*;

- *Tangential approach* – if is ON arc is the result of this task, otherwise – result is line segment;
- *Use midline*;
- *Inside* – indicates where to create the LeadIn path.



Figure 291. Find lead-in initial data (left) and result (right)

Get Centre Point

Calculates the centre point of **Polygon/Path** on the stack. A new **Path**, containing two same points having the same coordinates as the centre point, is created and placed on top of stack.

Get Mass Point

Calculates the mass point of the **Polygon/Path** on the stack. A new **Path**, containing two same points having the same coordinates as the mass point, is created and placed on top of stack.

Get start point

Gets the start point of the **Polygon** on the stack. A new **Path**, containing two same points having the same coordinates as the start point, is created and placed on top of stack.

Get Sub Path by Length

The single **Path** has to be traced for this task. This task creates new **Path** which is sub part of original **Path**. The *Distance* specifies the length of the new path. A sign of the *Distance* value indicates the beginning of a new path -the positive sign means that the beginning of the new path will be from the start of the original path, if negative, then, from the end of the original path.



Figure 292. Get sub path by length initial data (left) and result (right)

Get width and height

Calculates the bounding box of the **Polygon** on the stack. A new **Path**, containing two same points having x coordinate as width and y coordinate as height, is created and placed on top of stack.

Hatch

Fills a **Polygon** with parallel lines in the parallel distance *Distance 1*. The direction is specified by *Mode 1*. Hatch lines are extended, truncated or turned around by *Distance 2* as specified by *Mode 2*.

Intersection

At least one stack value must be a **Polygon**.

Intra minimum space

Checks if **Polygon** on the stack has narrow places defined by the **Distance** parameter.

Inverse

For **Polygons**, inverts all boundaries so that inner boundaries become outer boundaries and outer boundaries become inner boundaries. By this also the infinity flag is changing. For **Path** the direction of each single path is inverted.

Midline

Calculates a midline of the **Polygon** on the stack.



Figure 293. Midline initial data (left) and result (right)

Outline Diff

Calculates an outline of the second **Polygon** on the stack, which is not covered by the first **Polygon** on the stack, and converts the results into a **Path**.

Outline Shift

Calculates the outline of a **Polygon** which is shifted by a given value, and provides the result as a **Path**.

Oversize

In case of **Path** the **Path** is widened to an area.

Prepare heatings and extend stripes

Generates heating lines and extends stripes to the contour.

Remove Holes

Removes all holes and further nested objects inside.

Remove Outlines

Removes all outlines on the top level, in case the infinity flag is not set. In the result, the infinity flag is set.

Remove Small Holes

Removes all holes and possible further objects inside that are smaller than a given value.

Remove Small Paths

Removes **Paths**, which have the smaller length than a specified parameter.

Rotate

Either rotates or mirrors in fixed 90 degree steps, or rotates by an arbitrary angle.

Shift

Shift all objects by the given x-y value.

Smooth path

Removes points from the **Path** which are on the same line.

Sort Path

Sorts the order of single **Paths** in the given **Path** according to the specified mode. If the inverse flag is set, the direction of the single **Paths** may be changed.

Striping and Stripping

Generates cutting and heating lines within the **Polygon**.

Union

Both stack parameters are either **Polygons** or **Paths**. The result is of the same type. In case of **Path**, the **Path** on the top of the stack is appended to the 2nd path on stack.

XOR

Calculates the exclusive intersection XOR between the two **Polygons** on the stack.

Simple Conditional Polygon/Path Operations

Has Boundary

Starts inner operations if condition is met.

Has Data

Starts inner operations if condition is met.



Note: If level with a given name cannot be found, level will have no graphical objects, error message will be generated.

Is Infinite (Pp Infinity)

Starts inner operations if condition is met.

Conditional Polygon/Path Operations with Numeric Value to Compare to:

If Area

Starts inner operations if condition is met.

If Bounding Box

Calculates bounding box values of **Polygon** on the stack, and performs comparison of bounding box values with a specified value according to the *Mode* parameter. The *Mode* parameter can have a predefined value.

- X – width of bounding box is compared with the specified value;
- Y – height of bounding box is compared with the specified value;
- $\max X, Y$ – the higher value between width and height is found and compared with the specified value;
- $\min X, Y$ – the lower value between width and height is found and compared with the specified value.

If Circular Segment

Checks if a **Path** on the stack has a circular segment. A **Path** element must be on the stack.

If Full Circle Segment

Checks if a **Path** on the stack has a single circle segment. A **Path** element must be on the stack.

If Length

Starts inner operations if condition is met.

If Perimeter

Starts inner operations if condition is met.

If selected

Starts inner operations if amount of selected objects is bigger than specified **Amount**.

If Single Line Segment

Checks if a **Path** on the stack has a line segment. The **Path** element must be on the stack.

Looping Polygon/Path Operations

Area Decomposition

De-composes a **Polygon** into single areas with a single outer boundary or not, if the infinity flag is set.

Path Decomposition

De-composes a **Path** into single **Paths**.

Segment Decomposition

Iterates through single segments of a single **Path**.

Sheet Decomposition

De-composes a **Polygon** into polygons with a single inner or outer boundary.

Auxiliary

Change Parameter

Modifies the current **Parameter** by a specified value. Currently supported modes for changing **Parameter**'s value are: adding, subtraction, multiplication and division.

Compare Parameters

The specified **Parameter** is compared with an explicitly defined value or with other **Parameter**. Comparison modes are: less, more, equal and not equal.

Group

This command is only as placeholder for set of script commands which are intended to perform single task.

Polygon Package Resolution

Modifies polygon package resolution.

Quit Job

Stops the execution and exits the current job.

If **Quit job** parameter is *On*, the execution of the job is stopped.

Report Error

Displays an error message in a dialogue box.

Set Parameter

Sets a new value for the specified parameter, defined by the *Parameter name*. The *Value* can be set explicitly or specifying another parameter name by *Value*.

Set parameter from tool property

An active tool must be set before calling this task. The specified tool property of the active tool is read and set for the specified parameter by the *Value*.

Set Parameter by Tool Width

An active tool must be set before calling this task. A width value of the active tool is read and set for the specified parameter by the *Value*.

Skip

Empties operation, which purpose is to display some comment text (if specified).

Task Types

	Name	Read Stack	Write Stack	Parameters	Error Conditions
CircuitCAM document preparation					
	Clear work layer				9210
	Create layer			Layer Name	9200
	Create tool			Tool Name or combination of tool list and tool name, Diameter	9202, 9211
	Create tool list			Tool List name	9201
	If layer exist			Layer name	9200
	Set work layer			Layer Name or Active layer	9200, 9220, 9223
	Set work tool			Tool Name/Active tool or combination of tool list and tool name	9202, 9211, 9225
	Set work tool list			Tool List name or Active tool list	9201, 9221, 9224
Stack – CircuitCAM document interaction					
	Move to stack		Polygon++		9001, 9008
	Move to stack as path		Path++		9001, 9008
	Copy to stack		Polygon++		9001, 9008
	Copy to stack as path		Path++		9001, 9008
	Pull from stack	Path/Poly-gon--			9000, 9210
	Pull from stack as path	Path/Poly-gon--			9000, 9210
	Pop stack	Path/Poly-gon--			
General Stack and Register operations					
	Clear register				
	Clear registers				
	Clear stack				
	Copy register to stack				9100, 9101, 9001
	Copy stack to register				9000, 9100
	Create polygon		Polygon++	Infinite	
	Duplicate	Path/Poly-gon	Path/Poly-gon++		9000, 9001

	If register has value				
	Move register to stack				9100, 9101, 9001
	Move stack to register				9000, 9100
	Swap	Path/Poly- gon-- Path/Poly- gon--	Path/Poly- gon++ Path/Poly- gon++		9002
Simple Polygon/Path operations					
	Combine closed path	Path--	Path++	mode Distance:Real	9000, 9004
	Combine path	Path--	Path++	mode Distance: Real	9000, 9004
	Convert to path	Polygon--	Path++		9000
	Convert to polygon	Path--	Polygon++	Corner type End types Oversize value	9000, 9004
	Create mark	Path	Polygon++	Length : Real	9000, 9004
	Create needle from concentric	Polygon-- /Polygon--	Path++		
	Create serpentine	Polygon	Path++	ToolDiameter: Real; Mode: Integer	
	Cut path at path	Path--, Path--	Path++		
	Cut path at polygon	Path--, Path--	Path++		
	Cut stripes	Path--, Path--	Path++, Path++		9000, 9004
	Denest polygon cutout				
	Difference	Polygon-- Path/Poly- gon--	Path/Poly- gon++		9002, 9007
	Extend path to nearest object	Path--, Path	Path++		9000, 9004
	Find common path	Polygon--, Polygon--	Path++		9000, 9003
	Find lead-in	Polygon--, Polygon--	Path++	distance of lead in angle, direction, tangential approach, use midline, inside	9000, 9002, 9005, 9007
	Get centre point	Polygon--	Path++	result is PATH with 2 same points	9000, 9003

	Get mass point	Polygon--	Path++	result is PATH with 2 same points	9000, 9003
	Get start point	Polygon--	Path++	result is PATH with 2 same points	9000, 9003
	Get sub path by length	Path	Path++	Distance: Real	9000, 9004
	Get width and height	Polygon	Path++	result is PATH with 2 same points(width and height)	9000, 9003
	Hatch	Polygon--	Path++	Distance1: real; Distance2: real; Mode 1: integer Mode 2: integer	9000, 9003
	Intersection	Path/Poly- gon-- Path/Poly- gon--	++Path/Po- lygon		9002
	Intra minimum space	Polygon	Polygon++		9000, 9003
	Inverse	Path/Poly- gon--	Path/Poly- gon++		9000
	Midline	Polygon--; Path--	min dis- tance to border; bor- der- Indication: PATH		
	Outline diff				
	Outline shift	Polygon--	Path++	Shift: Real	9000, 9003
	Oversize	Path/Poly- gon--	Polygon++	Corner type Oversize value	9000
	Prepare heatings and extend stripes	Path--, Polygon--, Polygon	Path++, Path++		9000, 90003, 9004
	Remove holes	Polygon--	Polygon++		9000, 9003
	Remove Outlines	Polygon--	Polygon++		9000, 9003
	Remove Small Holes	Polygon--	Polygon++	Size: Real	9000, 9003
	Remove Small Paths	Path--	Path++	Size: Real	9000, 9004

	Rotate	Polygon--	Polygon++	Enum: 90 degree rotation and mirroring, Arbitrary angel : Real	9000, 9003
	Shift	Polygon--	Polygon++	x, y: REAL	9000, 9003
	Smooth path	Path--	Path++	Distance: REA- L	9000, 9004
	Sort Path	Path--	Path++	Inverse: bool- ean, Mode: Integer Direc- tion: integer	9000, 9004
	Striping and Stripping	Polygon--, Polygon			9000, 9003
	Union	Poly- gon/Path-- Poly- gon/Path--	Poly- gon/Path+- +		9002, 9005
	XOR	Polygon-- Polygon--	++Polygon		9002
Simple conditional Polygon/Path operations					
	Has boundary	Polygon		inverse flag	9000, 9003
	Has data	Poly- gon/Path			
	Is infinite (PpInfinity)	Polygon		inverse flag	9000, 9003
Conditional Polygon/Path operations with numeric value to com- pare to					
	If area	Polygon		Size: Real, Bigger or smaller: bool- ean	9000, 9003
	If bounding box	Polygon		Size: Real, mode: x, y, xy max, xy min	9000, 9003
	If circular segment	Segment			
	If full circle segment	Segment			
	If length	Path		Size: Real, Bigger or smaller: bool- ean	9000, 9004
	If perimeter	Polygon		Size: Real, Bigger or smaller: bool- ean	9000, 9003
	If selected			Amount: int	

	If single line segment	Segment			
Looping Polygon/Path Operations	`				
	Area decomposition	Polygon – (at end)	Polygon++ (for inner loop)		9000, 9003
	Path decomposition	Path – (at end)	Polygon++ (for inner loop)		9000, 9004
	Segment decomposition	Path –			9000, 9004
	Sheet decomposition	Polygon – (at end)	Polygon++ (for inner loop)		9000, 9003
Auxiliary					
	Change parameter			Value: dou-ble/string	9300, 9301, 9302
	Compare parameters			Value: dou-ble/string	9300, 9301, 9302
	Group				
	Polygon package resolution				
	Quit job				
	Report error				
	Set parameter			Value: dou-ble/string	9300, 9301, 9302
	Set parameter from tool property			Property name Parameter name: string	9212, 9300, 9301,
	Set parameter by tool width			parameter name	9301, 9302
	Skip				

Abbreviations, used in the table:

1. ++ : push something to stack.
2. -- : remove something from stack.

ERROR Codes:

9000: Stack is empty

9001: Stack overflow; Stack can hold at max 10 objects

9002: Expect at least two objects on stack

- 9003: Top object on Stack is required to be a Polygon, but it is Path
 9004: Top object on Stack is required to be Path, but it is Polygon
 9005: Top 2 objects on Stack must be both either Path or Polygon, but they are of different type
 9006: Second top object on Stack is required to be Path, but it is Polygon
 9007: Second top object on Stack is required to be Polygon, but it is Path
 9008: Nothing to move into stack, nothing selected or no work layer defined
- 9100: Register index is out of range (allowed 0 to 20)
 9101: Indexed register is empty, can't be moved/copied to Stack
- 9200: Invalid work layer name
 9201: Invalid tool list name
 9202: Invalid tool name
 9210: No work layer defined
 9211: No tool list defined
 9212: No tool defined
 9220: Specified layer not found
 9221: Specified tool list not found
 9222: Specified tool not found
 9223: Active layer not set
 9224 : Active tool list not set
 9225: Active tool not set

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.0.2. Layouts

The **Layouts** view presents a list of layouts. The **Properties** view shows details of the selected layout or layout components.

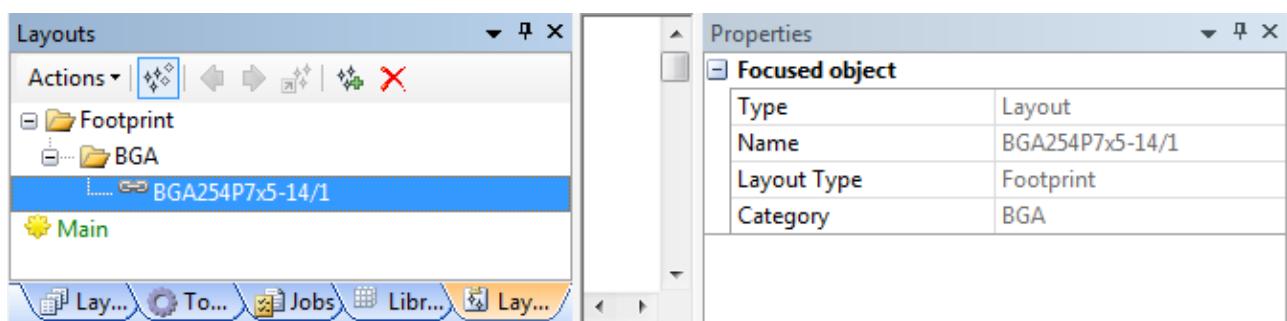


Figure 294. Layouts and Properties view

 **Note:** If the **Layouts** view is not visible on the screen, go to the menu **View --> Layouts** to make it visible.

Layouts in **Layouts** view are grouped by:

- layout type,
- layout category for layouts with same type.

Defined layout types in *CircuitCAM* are:

- Undefined/None,
- Land Template,
- Package Terminal,
- Padstack,
- Footprint,
- Package,
- Footprint Occurrence,
- GMC Shape,
- Package Footprint.

A *Layout* contains any type of graphic objects: paths, polygons, flashes and etc.. Layouts can have references to other layouts (which are called sub-layout instances).

A user can freely specify an applicable categories for layouts.

The following special notations are used in **Layouts** view:

- **Red** color - indicates that this layout is selected in the main graphic window.
- **Bold** - indicates that this layout is active (sub-layout instances to this layout can be inserted in to layout which is displayed in the main graphic window).
- - indicates that this layout is currently displayed in the main graphic window.
- - used only for **FootprintOccurrence** layouts and indicates that footprint occurrence doesn't have footprint information (no reference to **PackageFootprint** layout).

Context Menu

While using the right mouse button in the client area of the **Layouts** view, the following popup menu (mouse *context sensitive menu*) will appear:

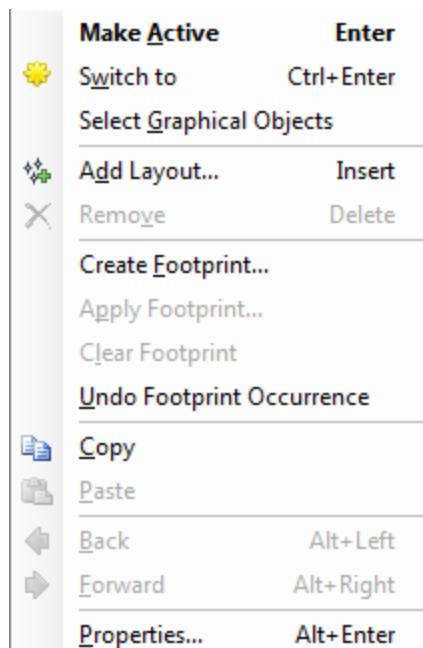


Figure 295. Layouts view context sensitive menu

	Make Active	Sets the selection in Layouts view as active, and it can be inserted into current layout.
	Switch to	Displays graphical data of selected layout in main graphic screen.
	Select Graphical Objects	Selects graphical objects in current layout.
	Add Layout	Adds new layout after current selection in Layouts view.
	Remove	Removes currently selected layout.
	Create Footprint	Creates footprint from currently selected layout (this can be performed on Footprint Occurrence or Package layout – for more details see below).
	Apply Footprint	Applies footprint for currently selected layout (this can be performed on Package layout – for more details see below).
	Clear Footprint	Removes footprint information from currently selected layout (this can be applied to Package Footprint layouts – for more details see below).
	Undo Footprint Occurrence	Removes footprint information and restores recognized objects (this can be applied to Footprint Occurrence layouts – for more details see below).
	Copy	Copy data of current layout to the standard clipboard.
	Paste	Pastes data from clipboard to current document.
	Back	Switches to previous layout.
	Forward	Switches to next layout.
	Properties	Displays properties dialogue box for currently selected layout.

Toolbar

Layouts view contains a toolbar.



Figure 296. Layouts view toolbar

	The list of available actions in the Layouts view. This list contains same items available in the context sensitive menu.
	Show All Layouts Shows/hides all layouts which are not used in the current layout
	Back

	Switches to previous layout
	Forward Switches to next layout
	To Main Switches to Main layout
	Add Layout Displays dialogue box for creation of new layout
	Remove Layout Deletes layout (only not used layout is available for delete)

See [Footprint](#) for the functions:

- *Create Footprint*,
- *Apply Footprint*,
- *Clear Footprint*,
- *Undo Footprint Occurrence*.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.0.3. Libraries

The **Libraries** view, presents a list of available libraries. The available libraries are stored in the installation directory under the **Libraries** folder. A user can change this folder in the [General Settings -> Directories](#) page. The **Properties** view shows details of the selected library.

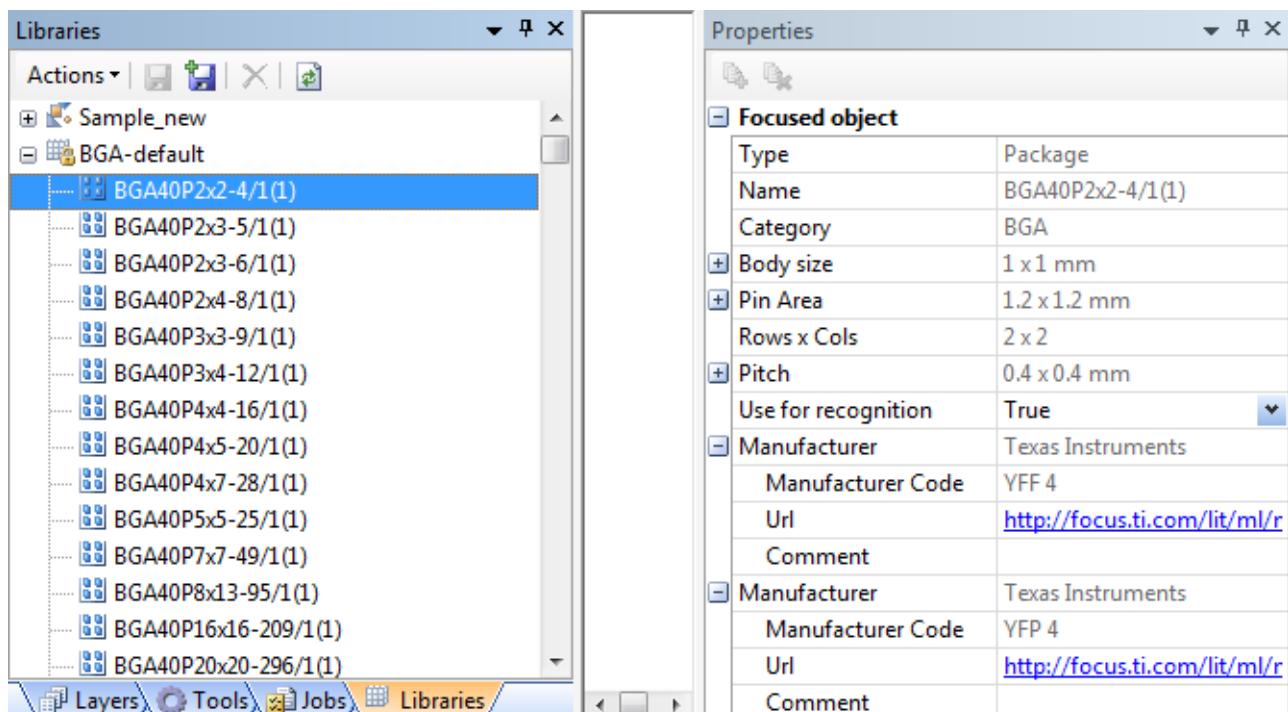


Figure 297. Libraries and Properties view



Note: If the **Libraries** view is not visible on the screen, go to the menu **View -> Libraries** to make it visible.

Types of Libraries

A user can create a library from the following three types of libraries:

- **Package** library.

A **Package** library file has the extension “*.cam-pkg”. **Package** libraries are used for [Package recognition](#). A new **Package** library entry can be created from scratch or from selected items. Packages are used to identify components such as BGA, PLCC and etc. on the current layout. A default “**BGA-default.cam-pkg**” library is provided with the *CircuitCAM* installation in the default libraries location (see [General Settings](#)).

- **Footprint** library.

A **Footprint** library file has the extension “*.cam-ft”. As a result of *Package recognition*, the current footprint in the layout can be replaced by a footprint of the **Footprint** library. A new **Footprint** library entry can be created in the **Layouts** view from layouts, which are either of type **Package** or **Footprint Occurrence** (see [Create Footprint](#)). Footprints can be applied only for *Packages* (see [Apply Footprint](#)). No default **Footprint** library is provided with the *CircuitCAM* installation.

- **GMC** shape library.

A **GMC** library file has the extension “*.cam-gmc”. [Until CircuitCAM version 6.2, GMC library had the extension “*.gmc” and can be converted to the new format](#) (see [Import Libraries](#)). A new **GMC** library entry can only be edited through the **GMC** dialogue (see [GMC](#)). GMC library shapes can be applied on any layout (see [GMC Compare](#)). A default “**Sample_new.cam-gmc**” library is provided with the *CircuitCAM* installation in the default libraries location (see [General Settings](#)).

States of Libraries and Library Components

The sub-icon indicates that a library is read-only and cannot be modified.

User can specify which components from which library to use for package recognition. This can be performed by double clicking on a single package in the **Library** view (the indicates, that the package will not be used for package recognition). This will include/exclude the package from the recognition procedure.

The sub-icon indicates that the item is excluded from recognition/exchange total library as well.

The current state of included and excluded packages is stored in the file **PkgRList.xml**, in libraries directory. This file is loaded when *CircuitCAM* starts, and is saved on exit.

After a package state is changed, the library is indicated as modified. It is necessary to *Save library* modifications.

Context Menu

While using the right mouse button in the client area of the **Libraries** view, the following popup menu (mouse *context sensitive menu*) will appear:

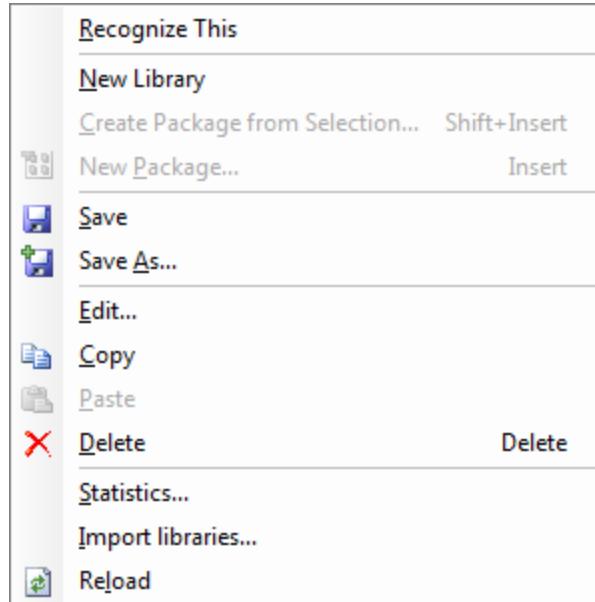


Figure 298. Libraries view context sensitive menu

Recognize This	Recognize the current selection in the library.
New Library	Create a new library.
Create Package from Selection	Create a new package from selected objects.
New Package....	Create a new package.
Save	Save modifications to the focused library.
Save As...	Save selected library with a new name.
Edit...	Edit library item. See also Tool Properties
Copy	Copy the selected element to the standard clipboard.
Paste	Extract the objects from the clipboard.
Delete	Delete library or library item.
Statistics...	Show statistics about focused library.
Import libraries...	Import old format libraries.
Reload	Reload libraries.

Toolbar

The **Libraries** view contains a toolbar.



Figure 299. Libraries view toolbar

Action	The list of available actions in the Libraries view. This list contains same items available in the context sensitive menu.
Save library	Saves modifications to the focused library.
Save As	Saves selected library with a new name.

	Delete Deletes library (only libraries that are not used/active are available for delete).
	Reload Reloads libraries.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.0.4. Packages and Footprints

Data formats (like *GERBER*, *GERBERX* and etc.), which can be imported to *CircuitCAM*, have only “raw” data “*pads and tracks*”. Users need to identify components on imported or created designs. For that purpose package library and package recognition can be used.

Default package library (contains BGA's packages) is delivered with *CircuitCAM*. The directory **Libraries** is created in *CircuitCAM* installation directory. The library file name is *BGAdefault.cam-pkg*. This library can be extended with new packages by the user (see [Library](#)).

Components from package libraries are used to scan through *CircuitCAM* data and identify components. A user can extend a default package library. New components can be added to library via **Library** menu or by context menu in **Libraries** view (see [Libraries](#)). New components can be created from scratch or from currently selected objects.

Package – is a layout with a type of *Package*. Pins are defined as instances to (*Package Terminal*) layouts. The placement points of these instances are used when the package is used for recognition.

Other information (like courtyard, pin-area and etc.) are placed in level objects on appropriate levels (“Courtyard”, “PinArea” and etc.)

- Each package entry in library contains:
 - Set of pins and centre points,
 - Pin area,
 - Courtyard,
 - Body size.

The names of packages are constructed in the following way:

- Package name.
- Pitch.
- Rows and columns.
- Pin number.

Example: BGA40P16x16-209 – means that this is BGA package; pitch is 40 mm, has 16 rows and 16 columns and contains 209 pins.

All packages have the “/x” and “(y)” post fix. “/1” and “/2” indicate different pin arrangements while “(1)” and “(2)” will indicate different alternatives of the same pin arrangement, but with different pin area, package area, courtyard and etc.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.0.5. Create Package from Selection

To create a package from selection, first of all *CircuitCAM* will check how many not read-only libraries exist in specified libraries directory. Possible situations:

- No libraries available for creating new packages – warning message will appear;
- One library for creating new packages – **Create Package from Selection** dialogue box will appear;
- Several libraries for creating new packages – the dialogue with detected libraries will appear and user must check one library to proceed further:

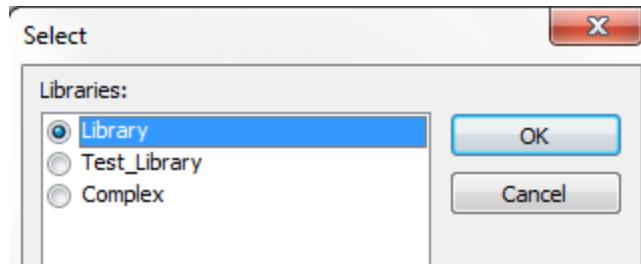


Figure 300. Select libraries dialogue window

Then the main dialogue window comes up with the package preview.

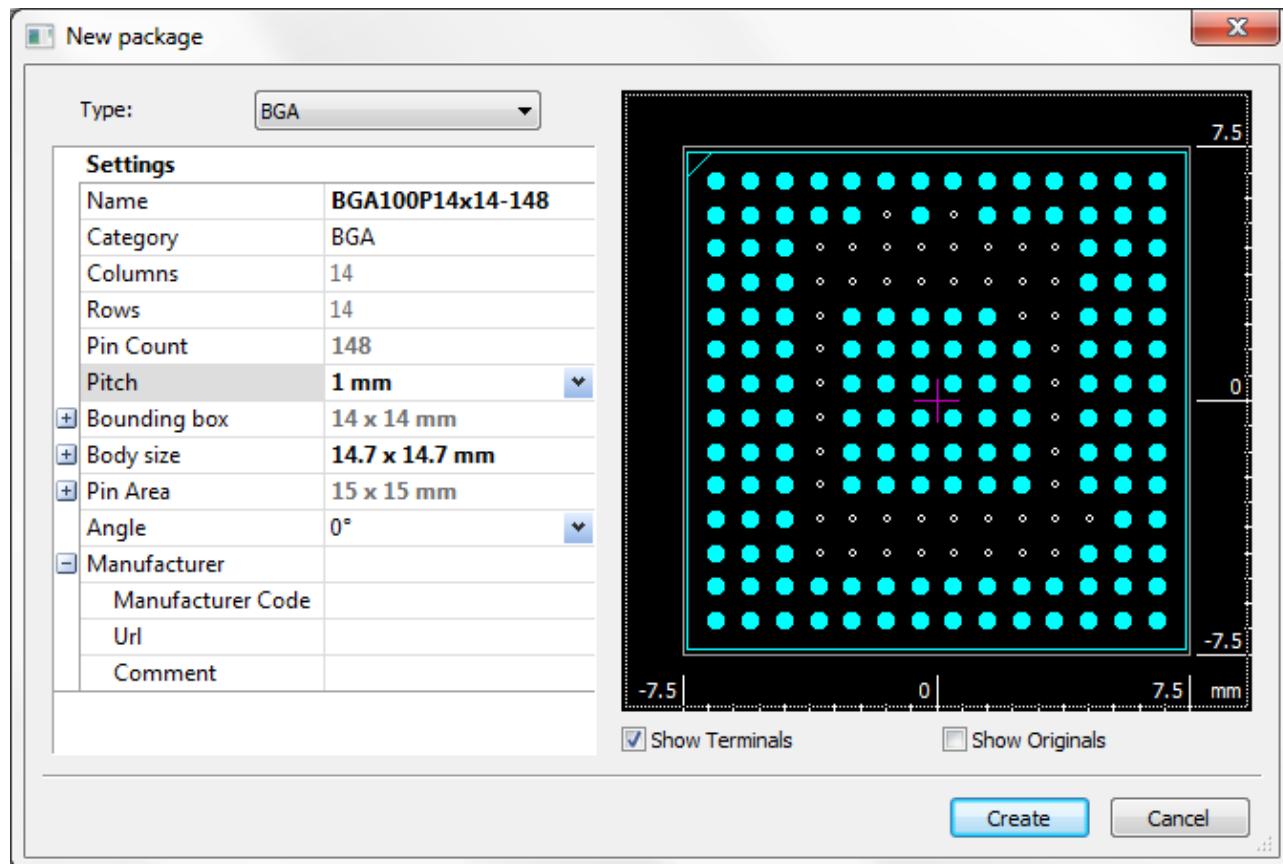


Figure 301. New package dialogue window

For newly created BGA packages all pads are circles with diameter half of pitch. Note, that you create only a package, but not a footprint here. So, these circles do not define the land areas to be used for the footprint. They are only for indication of the position and visualization, and are shown with the flag **Show Terminals**. Meanwhile the flag **Show originals** shows the original shapes of selected pads.

For all BGA types the pads are always on a fixed grid defined by a pitch (or pitch-x and pitch-y). But for other package categories they may have pads that are outside of the main grid.

The **0/0** point always is the centre of the package. For a 5x5 BGA this is the pad in the centre, but for a 4x4 BGA it is between the centre pads. All pads are either a full pitch or a half pitch away from the centre (in x and in y separately).

In the **General Setting -> Predefined Pitches** list a user can add some pre-defined values of typical pitches:

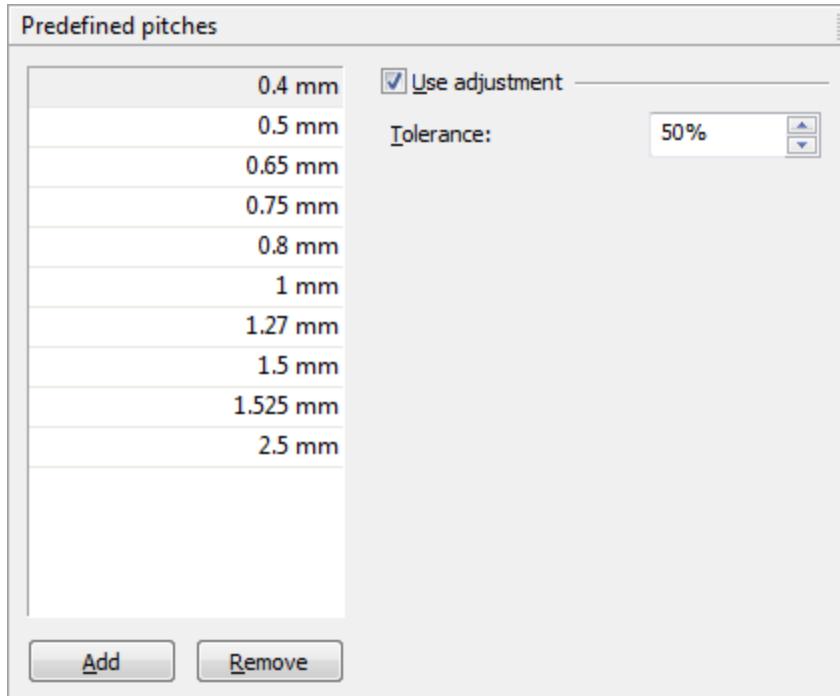


Figure 302. Predefined pitches panel

The combo box lists some pre-defined values of typical pitches, but a user can add pitches as well. The initial pitch values are taken from the current library. In general, the calculated pitch may be slightly different to some the ideal value from the predefined pitch list. In such case a pitch should snap to the nearest pre-defined value. For this the **Predefined Pitches** tab has snap-tolerance.

The **New Package** window has some information about the package to be created. Some parameters are editable (**Name**, **Category**, **Pitch**, **Body size**, **Angle**), other parameters are only informative.

The properties for BGA packages:

Name	Editable
Category	Editable
Pin	Read-only
Pitch	Editable
Bounding box	
Body size	Read-only
Pin area	Read-only
Angle	Editable
columns	Read-only
Rows	Read-only

During the creation of a new package from selection there is no direct information presented for the **Courtyard**, **Body size** and **Pin area**. So this information has to be derived from other values.

First a **Bounding-box** of all pads on layer terminal is calculated (those that will go into the package, including the final pad diameter).

The **Body size** is 5% bigger than the **Bounding box**, (2.5% in each direction).

width of the **Pin area** is number of pin columns, multiplied by pitch. The height of the **Pin area** is number of pin rows, multiplied by pitch.

The **Courtyard** is +15% of **Bounding box**, so 7.5% in each direction.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.0.6. Package Recognition

Performs automatic package recognition with enabled libraries.

Invoke the **Package Recognition** line from the **Library** menu. By mouse double-click enable/disable (remove/add the) the package library(-s) from **Libraries** view.

There are two possibilities to work with pads; on selected pads only or on the whole layout. In both cases, if pads belong to more than one layer, the user has to set layer(s) in the dialogue:

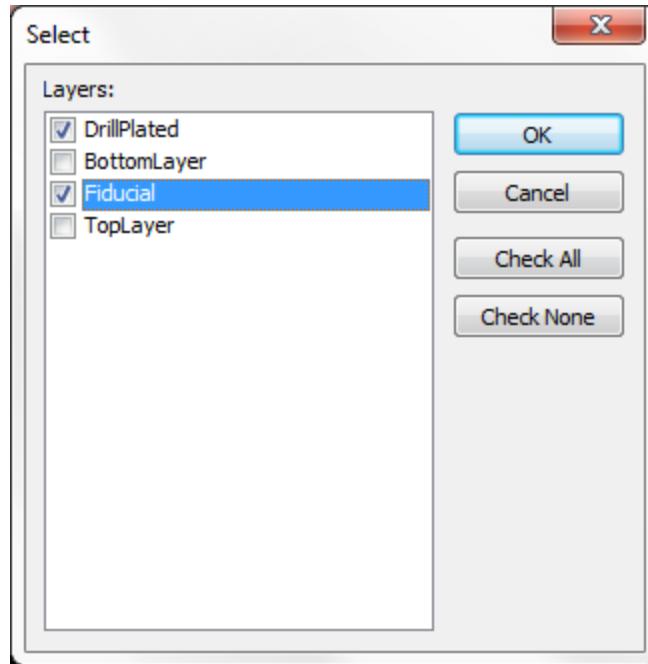


Figure 303. Select layers dialogue window

Automatic recognition process goes according to descending number of pins - starts from packages with maximum number of pins and ends with packages with minimum number of pins. The recognition ensures that there are no additional elements inside the courtyard. Otherwise the package is not recognized.

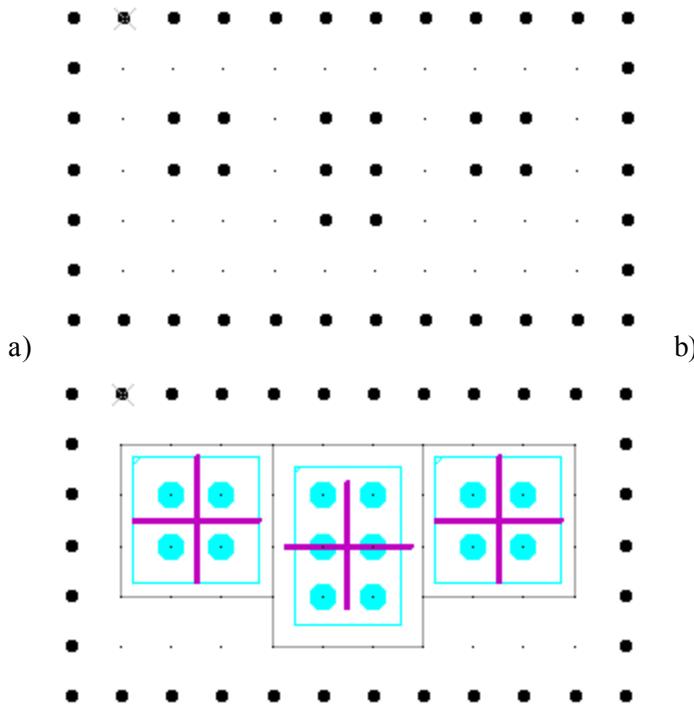


Figure 304. a) Initial data for recognition, b) recognized data

If a package is recognized and still does not exist in *CAM* file, the **Package** and its **Package Terminals** are copied from the **Package Library** to the *CAM* file.

A **Footprint Occurrence** is always created when package is recognized.

The **Footprint Occurrence** is a layout with type *FootprintOccurrence*.

Footprint occurrence is created in current *CAM* file: all relevant land objects (level objects with draws and flashes) are moved from the working layout into the new created layout. A sub-layout instance to the new created footprint occurrence is created, instead of moved land objects. Also the sub-layout instance to corresponding package is added inside the footprint occurrence. The name of the footprint occurrence is the name of the recognized package appended with a dot and sequence number.

If a user does not agree with the recognition result, he can simply denest the **Footprint Occurrence** with menu command **Modify -> Denest Flashes/Instances** and have back the original state.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.0.7. Footprints

Create Footprint

This command provides ability to extend footprints library by creating new footprints. This command is enabled only when a **Footprint Occurrence** or **Package** layout is selected.

The user needs to specify footprints library for creation of a new footprint. If more than one library exists in the **Libraries** directory, the dialogue box will appear and user will be prompted to specify library for storing of footprint.

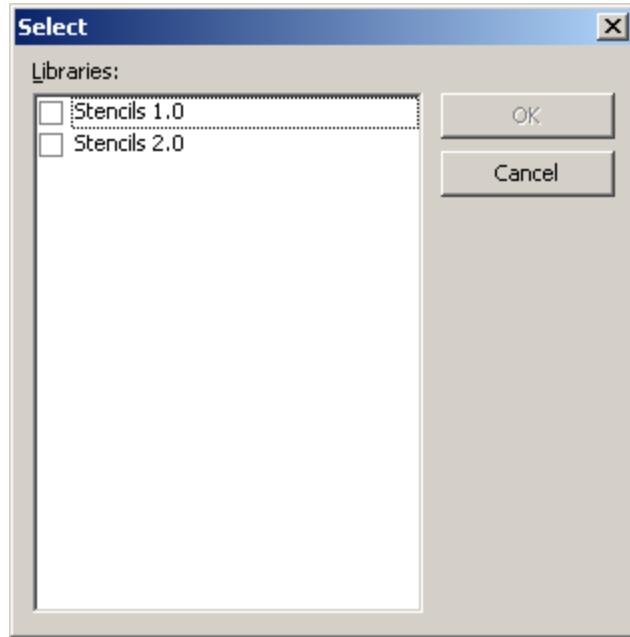


Figure 305. Select dialogue window to select libraries

If no *Footprint libraries* exist in specified libraries directory, the user should create a new footprint library before creating a footprint.

Data for footprints is located on the layer *ModifiedObj*.

The creation of a footprint from ***Footprint Occurrence*** is made as follows:

- All level objects from specified layout, are copied onto the layer named *ModifiedObj*;
- Other data (like sub-layout instances) are skipped.

The creation of a footprint from ***Package*** is made as follows:

- All sub-layout instances of packages are denested, and all level objects are moved to the *ModifiedObj* layer;
- The proposed name for new footprint will be similar to the selected package (information between "(...)" will be removed).

After the footprint is created, the user has the possibility to edit it. The newly created footprint is displayed in main graphic screen and is prepared for editing.

Apply Footprint

This command works only on selected ***Package*** layouts.

CircuitCAM searches all enabled footprints libraries (see [Library view](#)) to find the corresponding footprint for selected package.

Possible cases:

Case 1: A corresponding footprint was not found.

No actions are performed.

Case 2: One corresponding footprint was found.

First, all level objects are deleted from the selected ***Package*** layout. After this, all level objects on *ModifiedObj* level from corresponding footprint will be copied into the selected layout. All copied level objects will be moved on the layer, on which current ***Package*** layout was found.

Case 3: A few corresponding footprints were found

The dialogue box with the list of all corresponding footprints will appear and user will need to specify which concrete footprint to use. When user specifies, which footprint to use, all other actions are same like in case 2.

If the apply operation was successful, the target layout type is changed from **Package** to **PackageFootprint**.

Clear Footprint

This command can be used only for selected **PackageFootprint** layouts.

CircuitCAM will remove all footprint information from selected **PackageFootprint** layout: remove level objects from corresponding level (level on which **Package** layout was found) and layout type is changed from **PackageFootprint** to **Package**.

Undo Footprint Occurrence

Removes footprint information and restores recognized objects

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.0.8. Geometric Manipulation Centre

The **Geometric Manipulation Centre** (GMC) is a group of toolbars and dialogues to analyse systematically all pads in a layout and to modify them as needed. The resulting modified pads are primarily used to establish a paste mask. To optimize the solderability of a PCB typically modifications of the solder paste mask are needed. In general the GMC can be used to make any kinds of mass modifications on graphic objects that are provided in patterns.

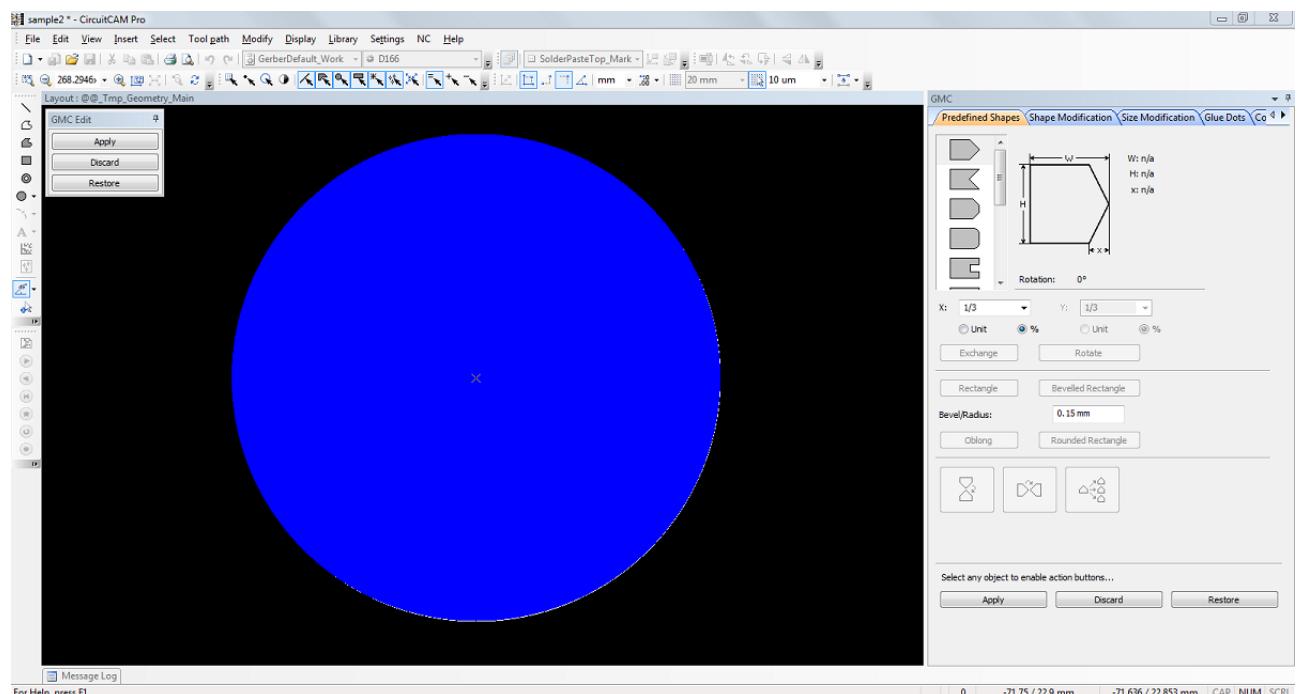


Figure 306. GMC main overview

The GMC maintain libraries of modified pads, group of pads or complete footprints for later re-use. The requirements on how to modify pads are dependent on the used PCB technology or

customer requirements. Thus several libraries with alternative modifications are typically needed.

To organize a systematic review of all pads, several additional layers are used.

The following sub-layers for each layer *Xxx* (e.g.: *SolderPasteTop*) are used throughout the GMC process:

Xxx: original layout;

Xxx_Work: objects not yet processed;

Xxx_Mark: objects to be modified;

Xxx_Modified: objects (after modification) waiting for assertion to be ready;

Xxx_Ready: objects ready for any operation/calculation.

GMC functionality

The GMC functionality consists of:

- The GMC view for detailed modifications of pads and building up a GMC library.
- The GMC toolbar with the function
 - 1.  (to open the GMC view)
 - 2.  to move selected objects from *Xxx_Work* to *Xxx_Mark*
 - 3.  to move selected objects from *Xxx_Mark* to *Xxx_Work*
 - 4.  to move selected objects from *Xxx_Modified* to *Xxx_Mark*
 - 5.  to move all objects from *Xxx_Modified* and *Xxx_Mark* to *Xxx_Ready*
 - 6.  to move selected objects from *Xxx_Ready* to *Xxx_Work*
 - 7.  to copy all objects from *Xxx* to *Xxx_Work*. The original objects on the *Xxx* layer become invisible. All other specific GMC layers are empty.

Toolbar and workflow

Workflow with GMC View and toolbar with no or incomplete GMC library:

1. The GMC-Init  **Copy all to WORK** operation copies all objects from *Xxx* to *Xxx_Work*, and indicates that all objects are waiting to be processed.
2. The  button moves selected elements from the *Xxx_Work* to the *Xxx_Mark* layer. The opposite direction is available with the  button.
3. Press on **GMC** () to start the GMC view that switches the complete *CircuitCAM* behaviour into the special GMC mode.
4. There is a shortcut for the steps {1} to {3}: select some pads and press the **GMC** button.
5. While in GMC mode, we see only the pads on the *_Mark* layer and their actual modification, if available. User can now modify the pads by using any combination of predefined shape, shape modification, size modification or glue dots.
6. When modification is finished, the resulting pad or group of pads, or complete footprint can be stored in a GMC library under some user-defined name
7. The performed modification can now be applied to all other similar pads or group of pads that remain on the *Xxx_Work* layer.

8. Till this point all modifications are temporal. Only when the **Apply** button is pressed, the modified objects are removed from the *Xxx_Mark* layer and placed on the *Xxx_Modified* layer, and the special GMC mode ends. Discard also ends the GMC mode, but all modifications are ignored.
9. After additional manual verification, all modified objects on *Xxx_Modified*, including the objects on *Xxx_Mark*, are moved to *Xxx_Ready* by the operation **Modified-to-Ready** (). Nevertheless, an incorrect modified objects can be send back to *Xxx_Mark* with **Modified-to-Mark** ().
10. Steps 2 to 9 are repeated till all objects from *Xxx_Work* are processed. Therefore, no pad remains on this layer, and the complete result is on the *Xxx_Ready* layer. This ends the GMC process
11. In the rare case that afterwards some incorrect modified object are detected on *Xxx_Ready* they can be selected and send back to *Xxx_Work* by the operation **Ready-to-Work** ().

The  **Init** button has an additional function:

The faster and more convenient way to modify size of a flashed pad is to change its size in the *CircuitCAM* aperture list, instead of using the *Size Modification* in the *GMC*. But, to see modifications in the graphic, these pads must be moved to the *Xxx_Modified* layer. Therefore *CircuitCAM* checks if the *Init* button has been pressed before an aperture was modified. If yes and the size or shape of the aperture was changed, all objects drawn or flashed with this aperture and located on the *Xxx_Work* layer are automatically moved to the *Xxx_Modified* layer. The aperture could be modified either from the main toolbar aperture list or from the *List* view aperture list or from the aperture graphical editor.

GMC view details

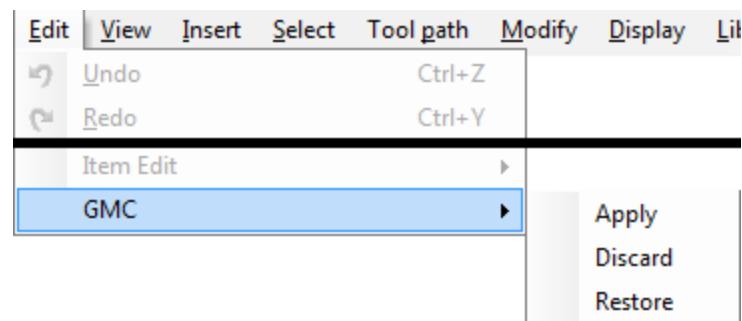
The GMC view is divided into six tab pages:

- [Predefined Shapes](#)
- [Shape Modification](#)
- [Size Modification](#)
- [Glue Dots](#)
- [Compare](#)
- [Lib](#)

Three buttons are common on every tab:

1. **Apply**— press this button to apply changes to main graphic. *CircuitCAM* will close *Geometry Manipulation Centre*, and will replace originally opened objects by modified ones.
2. **Discard**— press this button to discard changes made in GMC. *CircuitCAM* will close *Geometry Manipulation Centre*, and no modifications to main graphic screen will be made.
3. **Restore**— press this button to restore original shapes after modifications in GMC.

Those three options, are also accessible through the menu; **Edit ---> GMC ---> then choosing the desired action**.



Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.0.9. Predefined Shapes

The operations in the **Predefined Shapes** panel in most cases are executed on a pair of pads. The user has to select a pair of pads and open the GMC **Predefined Shapes** page.

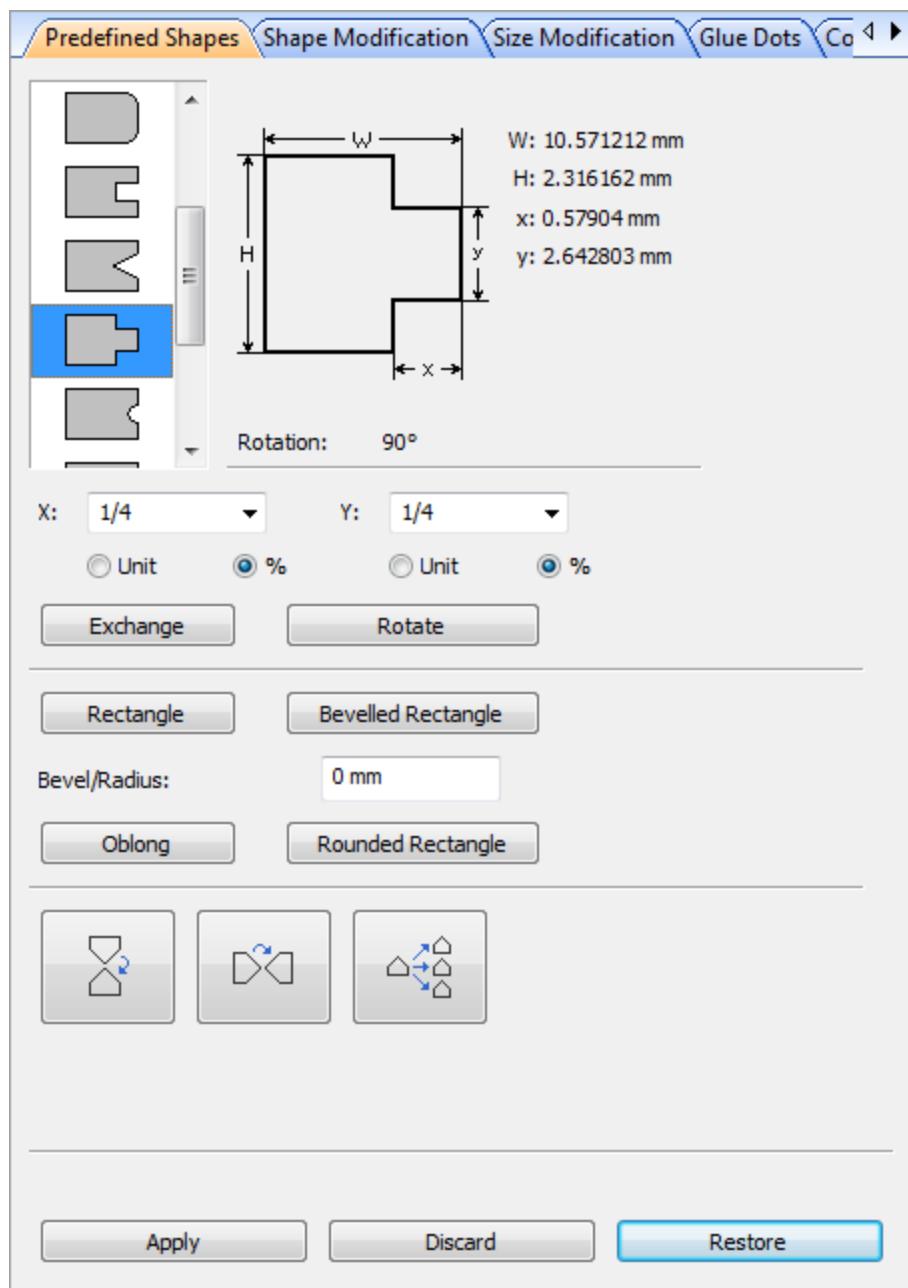


Figure 307. GMC, Predefined Shapes panel

Set of predefined shapes is represented in the list of predefined shapes. The **X** and **Y** values can be set directly in *Unit* mode or calculated automatically according to the shape length and width in parentage mode **%**.

Exchange: press this button to replace selected object(s) with the shape, which is currently selected in the list of *predefined shapes* (the size of new objects is exterminated separately for each object – using the bounding box of the object).

Rotate: Each click on this button rotates a new shape by 90° counter clockwise. At the same time the shape is recalculated to fit the bounding box.

Rectangle – converts selected object(s) to a surrounding rectangle.

Oblong – converts selected object(s) to a surrounding oblong.

Bevelled – converts selected object(s) to a bevelled rectangle (the corner value is retrieved from the **Radius** edit box). **Rounded** – converts selected object(s) to rounded rectangle (the radius value is also taken from “radius” edit box).

Flip horizontally and **Flip vertically** buttons mirror specified shape(s).

Scenario:

- 1) select modified shape;
- 2) press either **Flip horizontally** or **Flip vertically** button;
- 3) click mouse on original shapes to be exchanged, or use rectangle selection mode if more than one original shape must be exchanged.

CircuitCAM mirrors the selected modified shape and copy it onto the pads, specified by mouse. The bounding box centre point of the pad and a new shape is on the same coordinate.

The third button - **Make all shapes as selected** automatically replaces all original shapes by the selected modify shape.

4.0.10. Shape Modification

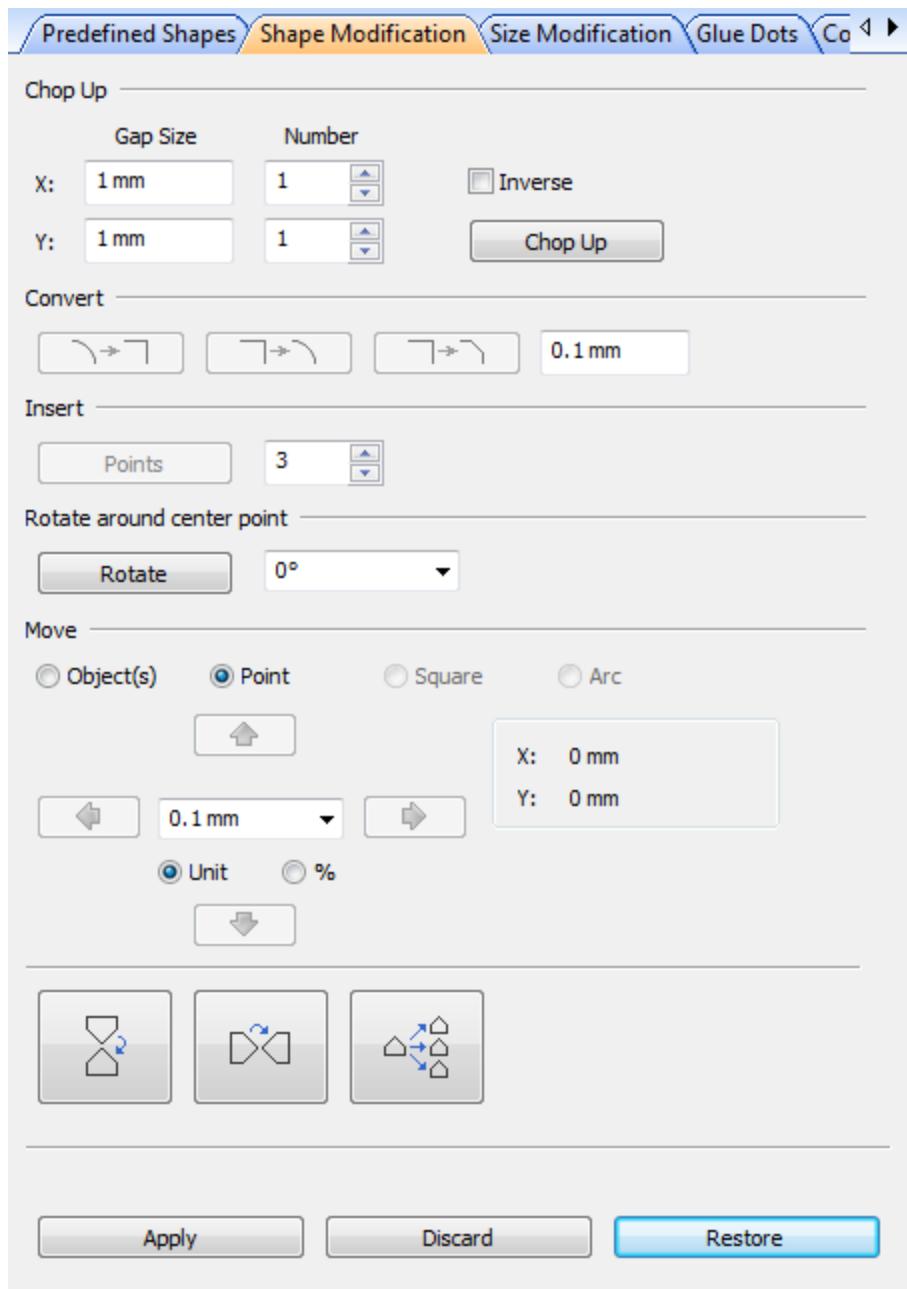


Figure 308. GMC, Shape Modification panel

Chop Up group:

Divides a pad into two or more parts.

This section works on all selected objects. Buttons in this group are enabled only if at least one object is selected.

X / Y specifies a gap size, **Number** defines amount of gaps on x/y directions.

Inverse flag - select this option to remove divided parts. The resulting pad is a union of gap(s) segments.

Example:

With the current *Chop Up* settings

**Figure 309. Chop Up settings**

a square shape is cut in this way (the right splitting is done with *Inverse* flag):

**Figure 310. Chop Up example**

Convert group:

This section works with a single object. State of buttons also depends on the current marker position

The functionality components of the group:

- **arc to corner** – converts the selected arc into a corner. The button is activated only if an arc or a part of an arc of the object is selected;
- **corner to arc** – converts selected corner into arc, the radius is specified in the “Radius” edit field. This button is activated only if the corner of the object is selected;
- **corner to bevel** – converts selected corner into bevel . This button it is activated only if the corner of the object is selected;

Insert group:

- **Points** – inserts point(s) in the selected line. Specify an amount of points in the combo box. This button is activated only after the selected point on the object is in the middle of the line;

Rotate around centre point group:

- **Rotate** – rotates/mirrors the selected object according its centre point. Select the rotation angle and mirror type in the corresponding combo box. A user is allowed to enter a rotation angle manually.

Move group:

Make your selection from the buttons to move a line, arc, point of selected pad or whole pad by the specified value:

- **Point:** moves the selected point (line start/end point, line middle point, arc start/end point, arc middle point, arc centre point)
- **Object:** moves a whole pad;
- **Square:** moves a line segment of a pad (available only if a middle point of a segment is selected);
- **Arc:** moves an arc segment of a pad (available only if a middle point of an arc is selected);

The distance a segment or a pad will be moved, is chosen from an additional combo box. The value can be either absolute (button **Unit**) or relative to the pad size (button **%**). A user can select one of the predefined values or enter value/ratio manually.

Flip group:

The group is common with **Predefined Shapes** tab and works in the same way.

Scenario:

- 1) select modified shape;
- 2) press either **Flip horizontally** or **Flip vertically** button;
- 3) click mouse on original shapes to be exchanged or use rectangle selection mode if more than one original shape must be exchanged.

CircuitCAM will mirror the still selected modified shape and copy it onto the specified by mouse pads. The bounding box centre point of the pad and a new shape will be on the same coordinate.

The third button - **Make all shapes as selected** automatically replaces all original shapes by the selected modify shape.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.0.11. Size Modification

The **Size Modification** tab in the GMC has the exact same features as in the **Graphic view, Size Modification Menu**.

The description of each feature can be found on the [Size Modification Menu](#).

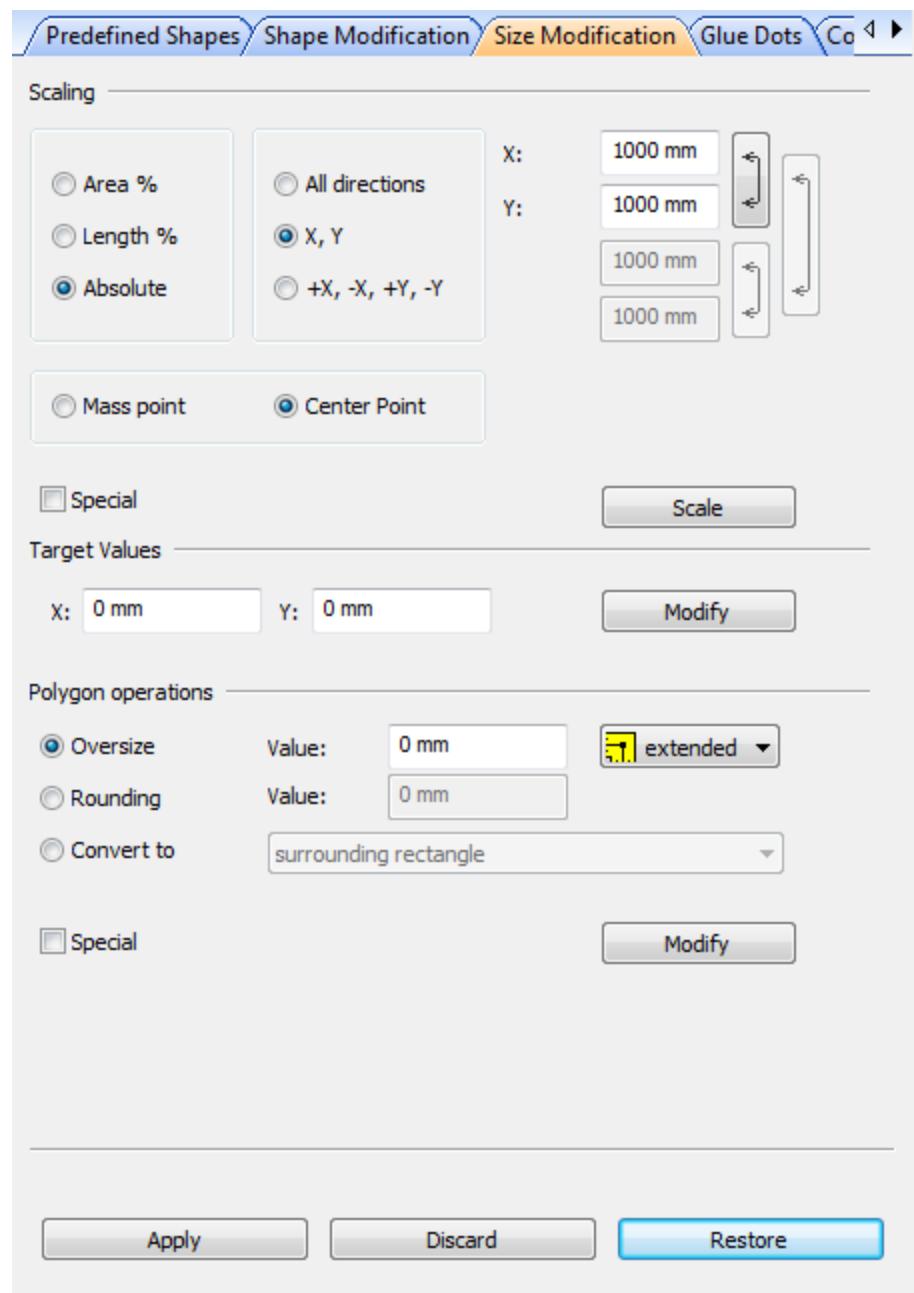


Figure 311. GMC, Size Modification panel

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.0.12. Glue Dots

This tab allows a user to place a defined number of circular dots with a defined diameter and distance to each other onto the original pads (these dots are centred on pads).

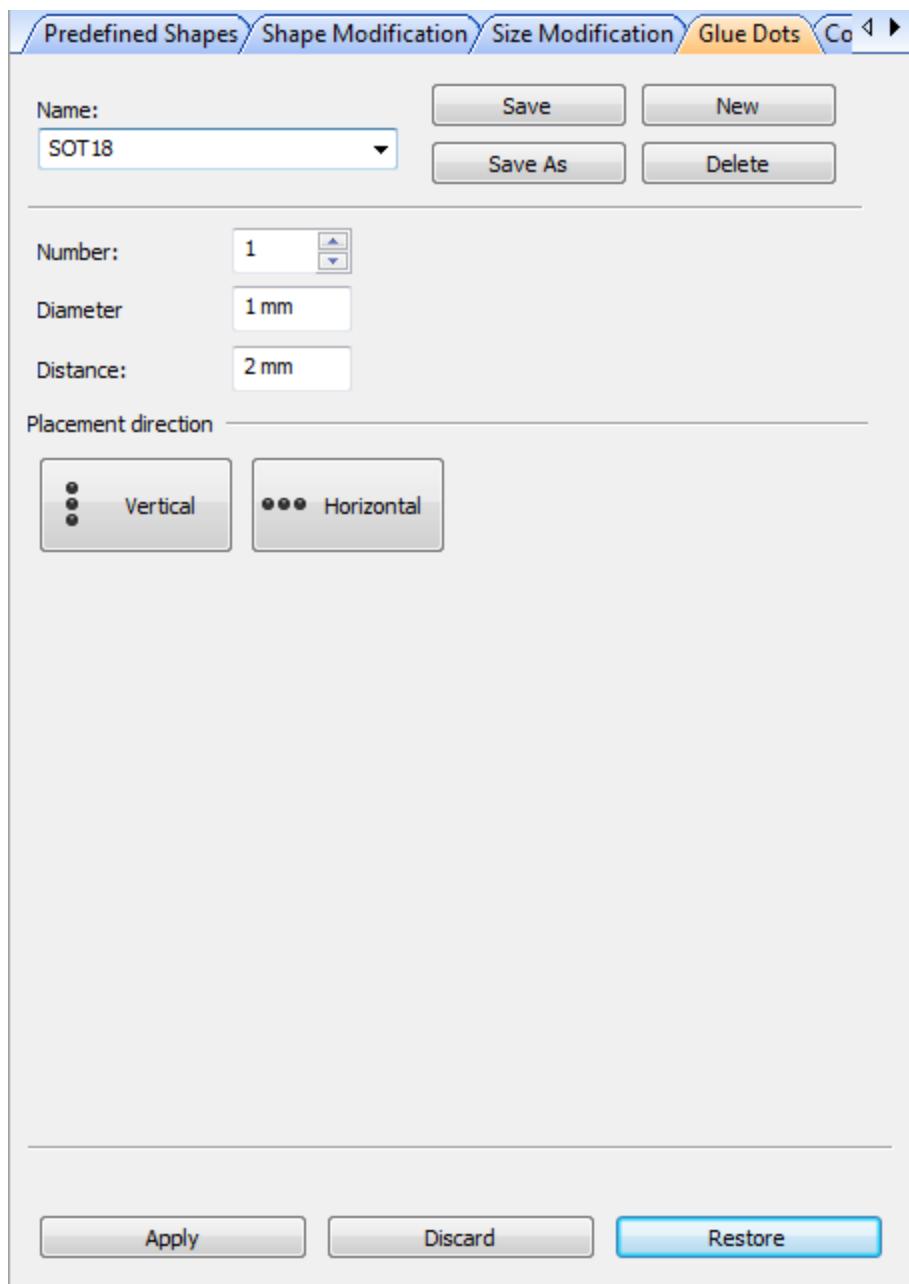


Figure 312. GMC, Glue Dots panel

The characteristics of glue dots depend on the component type. The user can save them by the component name (SOT 18 for example). These names appear in the **Name** list box, sorted alphabetically. The same name can be used only once in this list.

Each component may have a different combination of parameters **Number**, **Diameter** and **Distance**. The name and these three parameters are saved as a list in a text file with the fixed name “*Glue.dot*”. The file must be located in the **Libraries** folder. Structure of the file: SOT 18512.5SOT 14412

If the file “*Glue.dot*” is not present, the **Name** list is empty. The file will be updated after **Save**, **Save as...** or **Delete** was pressed.

In case more than one dot has to be placed, they can be placed either horizontally or vertically by using two special buttons, located on the right side of the group controls.

The button **New** clears the **Name** field and defines settings **Number** = 1, **Diameter** = 1 mm, **Distance** = 2 mm (these default settings appear when the *GMC* is opened).

The button **Delete** removes the actual name from the list, clears the **Name** field, defines settings **Number** = 1, **Diameter** = 1 mm, **Distance** = 2 mm and updates the file *Glue.dot*.

The button **Save as...** opens a dialogue box to enter a component name and updates the *Glue.dot* file.

The button **Save** saves actual settings for the **Number**, **Diameter** and **Distance** to the actual **Name**.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.0.13. Compare

The purpose of this section is to compare the originally placed objects in *_MARK* layer with objects placed in *_WORK* layer. When identical objects (according selected criteria in compare section) are detected, they are substituted by objects, modified in *_MARK* layer.

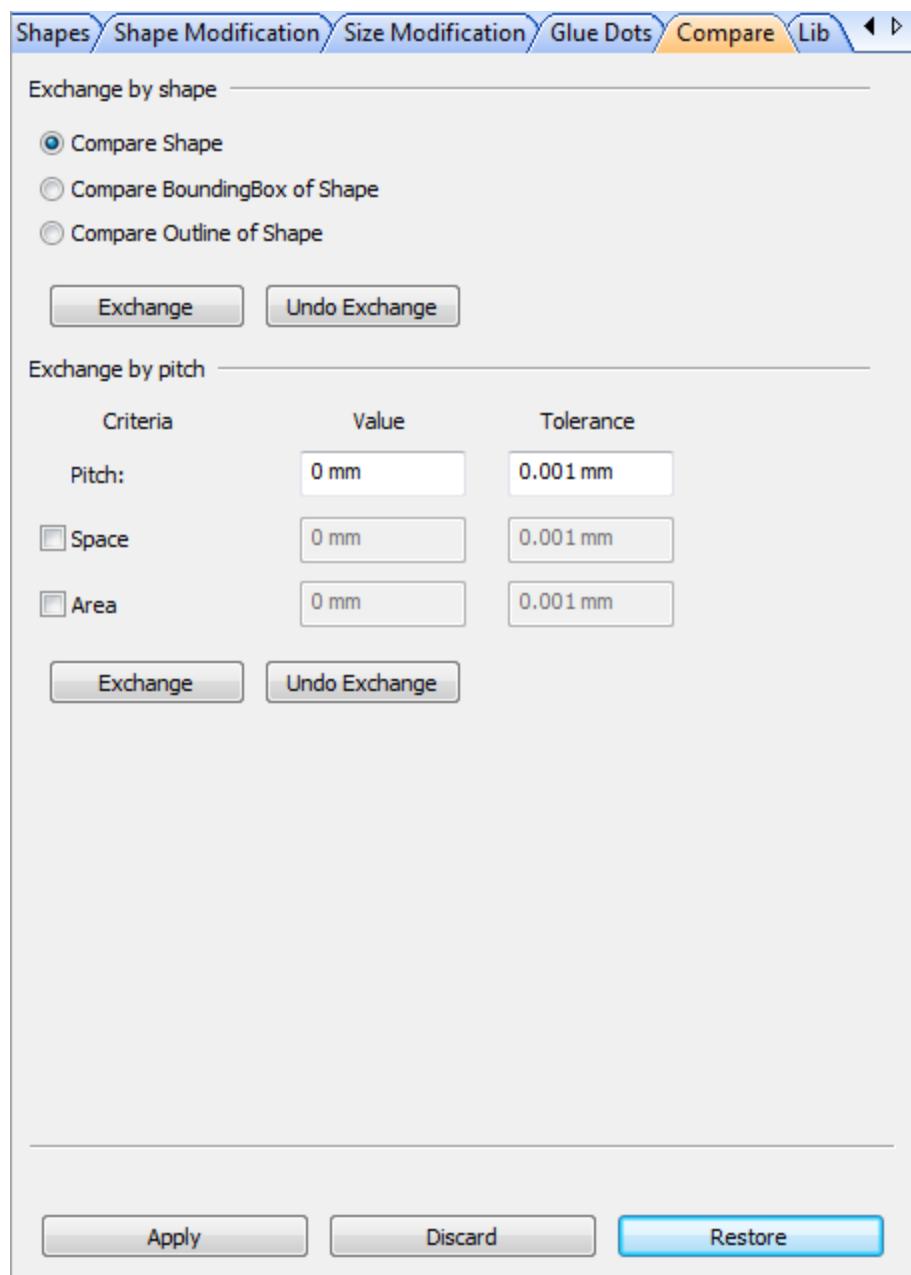


Figure 313. GMC, Compare panel

User can specify the criteria for the compare/exchange action:

- **Compare Shape** - compares all shapes with the original objects and replaces with the modified objects.
- **Compare Bounding Box of Shape** - compares a bounding-box of all shapes with a bounding-box of original objects, and replaces with modified objects.
- **Compare Outline of Shape** - compares an outline of all shapes with an outline of the selected flash, and converts matching outline.

For **Exchange by pitch** group fields description refer to [Pitch](#). The only one difference for pitch exchange part - detected elements are not only marked as selected (as it is single selection by pitch), but also changed by the modified ones.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.0.14. Library

The **Lib** panel allows managing of arbitrary shapes to be modified in a library. This includes viewing, creating, applying, and deleting of shapes and libraries of shapes. A shape in the library consists of the original geometry and the modified geometry, by which the original geometry is to be replaced.

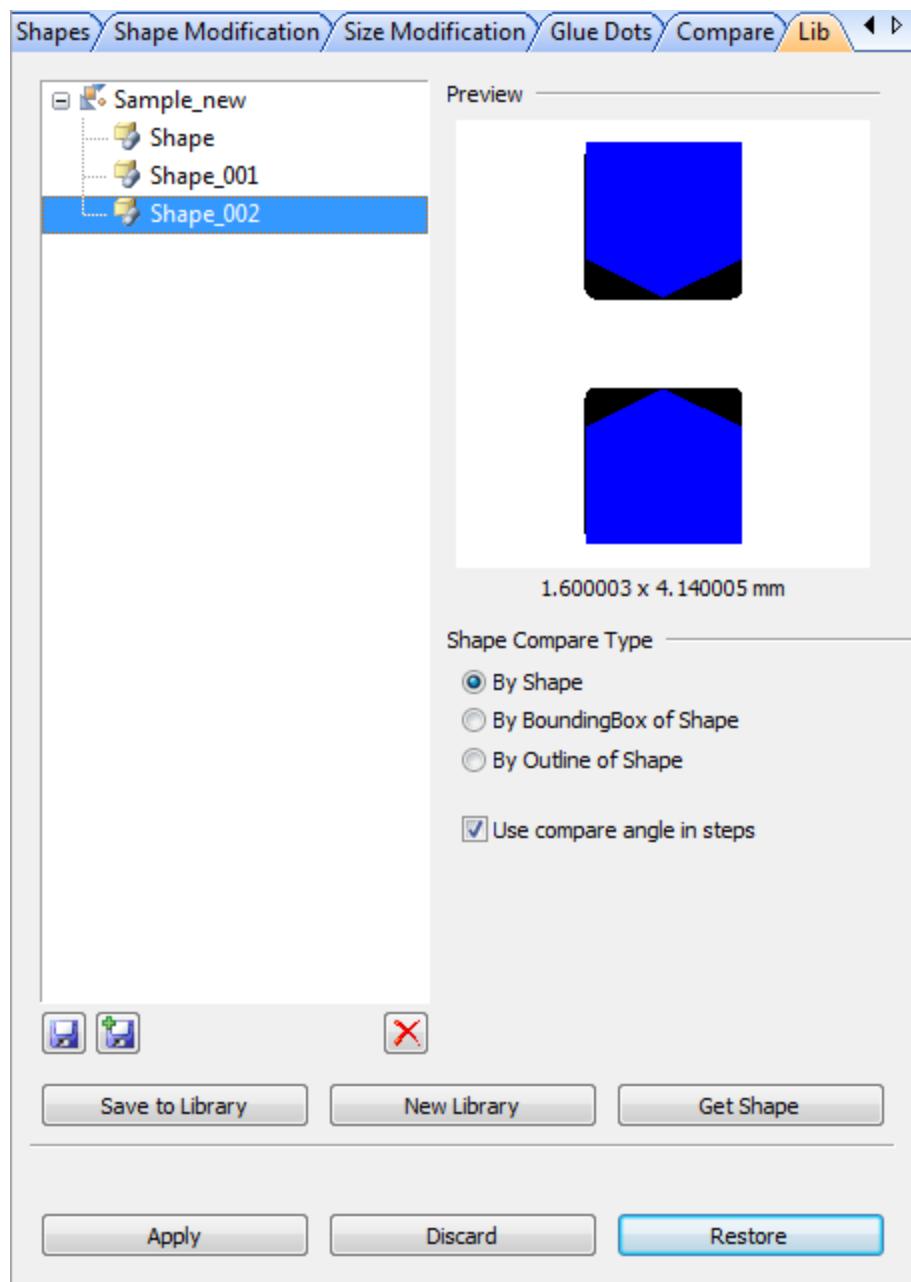


Figure 314. GMC, Lib panel

Lib list contains names of libraries, located in the directory (specified in **General Settings... - > Directories -> Library Directory**). Library files have extensions *.gmc and all these files are read and loaded by *CircuitCAM* before this dialogue box is opened. If the user changes extension of library file manually, *CircuitCAM* will not load this file.

The tree on the left side of the panel shows available GMC libraries and available shapes in each library.

Preview shows the graphical representation of a selected shape: black color indicates the original shape, and blue color the modified shape (see [Figure 1](#)).

Get Shape button loads the selected shape in the graphical window, where it can be modified manually.



Note: Only the modified shape can be edited, but not the original one.

Save to Library button saves the modified shape (together with original one) under a new shape in the current library.

New Library creates a new library.

  **Save** and **Save As...** icons are dedicated to save the selected library under the same or a new name.

 **Delete** button deletes the selected library or shape. If selected library is not empty, *CircuitCAM* will not allow a user to delete it. **Delete** button removes the selected shape in a library without any restrictions.

The **Shape Compare Type** group allows a user to specify a comparison process mode for each shape separately.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.0.15. Data Formats

There are several data formats supported by *CircuitCAM*:

- **CAM:** The native *CircuitCAM* format, *CAM* files are made to store the 2D graphic information of bare PCBs.
- **CAT:** The *CAT* files have the same internal format as *CAM* files, but have the file extension *CAT*. In this file users can store default layers, configurations, pads, tools, default jobs, etc. but usually no graphics. So this is an empty layout.
- **GERBER:** Is a format, in which *Gerber* data is stored. This format is actually a family of data formats which are subsets of EIA Standard RS-274D.
- **GERBERX:** Extended *Gerber* format, which is also called RS-274X, provides enhancements that handle polygon fill codes, positive/negative image composition, and custom apertures, and other features.
- **EXCELLON:** Excellon format usually means; the parts programming codes of the CNC-2, 4, 5, 6, or 7 Excellon machines.
- **HP-GL:** Hewlett Packard developed the *HP-GL* command language for its pen plotters. Later, it was taken on by other plotter manufacturers and has today become the standard for pen plotters.
- **SIEB & MEYER:** This data format is used by the milling and drilling machines from Sieb&Meyer.
- **BARCO-DPF:** This data format, developed by *Barco*, is designed to represent layer information of a PCB. This format describes the image of the layer such as pads, tracks, holes, power and ground planes, and also electrical netlist information, as well as additional product information represented with attributes.
- **PostScript:** Its a page description language, using vector and bitmap technology. For the purpose of the *CircuitCAM*, only vectors are used (lines and circular arcs). See *PostScript Language Reference* from *Adobe Systems, Inc.*
- **DXF:** DXF is a tagged data representation of all the information contained in an *AutoCAD* drawing file.
- **EESOF:** EESOF is an EDA System for High Frequency applications from *Agilent Technologies*.
- **G-Code:** Numerical control format *.NC which is commonly used by 3 to 5 axis NC machines according to the international standard-*ISO 6983* or a vendor specific variant.
- **ODB++:** Is a data format for PCB artwork from *Valor Computerized Systems Ltd.*

- **LMD (Laser Mill Drill):** This format is used to Export *CircuitCAM* data, and Import from *LMD*. The default file extension for the format is *.LMD
- **ISO 10303-210:** The international *STEP* standard *ISO 10303* is on *Product data representation and exchange*. Part 210 of *STEP* is the *Application Protocol: Electronic assembly, interconnect and packaging design*.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

Aperture Files and Templates (translation files)

The aperture file is an ASCII file, listing the apertures with their corresponding definitions, *e.g.: an aperture number, a type, and a size*. The ***Gerber*** data can be converted correctly only with the matching aperture list.

A translation file is used to convert the aperture file from any *CAD* system so that it can be imported into *CircuitCAM* without further modifications. This significantly speeds up the work with different aperture lists, as there is no need to define apertures manually. Furthermore, you can use the translation file to convert tool lists in *Excellon* or *Sieb&Meyer* format.

CircuitCAM has a number of prepared translation files for the most common CAD systems. However, it may be the case under certain circumstances that you need to customize a translation file individually. You can do it in the two ways:

1. Manually - by using any text editor;
2. Automatically – by using the graphical **Aperture Template Editor** (ATE) program.
The ATE can be used as stand-alone application. See [Aperture Template Editor](#).



Note: The translation files may be found in the *CircuitCAM "Ape_Templates"* sub folder, or in the folder you have specified in your *CircuitCAM General Settings* for the aperture and tool template files.

A translation file is an ASCII format file and can be edited in any text editor. The file is made up of three sections:

- Fixed parameters;
- Aperture definitions;
- Variable parameters.

Fixed Parameters

The following table shows a listing of the available fixed parameters with possible parameter values and the corresponding default setting.

Parameter name	Parameter value	Default setting for parameter value
APE_UNITS	INCH MIL MM	MIL
APE_SCALE	<real number>	1
COMMENT	<Character> <Character pair>	Empty set

	<Character> <Character pair>	
CONFIGURATION	ConfigExcelon ConfigGerber ConfigHpgl ConfigSiebMeyer	ConfigGerber
COOR_MN	<integer>.<integer>	2.3
COOR_MODE	ABSOLUTE INCREMENTAL	ABSOLUTE
COOR_UNITS	INCH MM	INCH
COOR_ZEROS	DECIMAL LEADING .TRAILINGING	.TRAILINGING
KEY_STRING	<character string>	
REQUIRED_STRING	<character string>	

Legend:
| Alternative (or)
[...] 0 or multiple repetition

- **APE_UNITS** - Defines the unit for the aperture size within the aperture list (**MIL**, **INCH** or **MM**).
- **APE_SCALE** - The aperture size within the aperture list can be scaled in addition by this parameter during import. *Example:* The circular aperture size is 10 mm and **APE_UNITS** is set to **MM**. Without **APE_SCALE** a circle with 10 mm diameter will be imported. If **APE_SCALE** is set to 0.1, the aperture with 1 mm diameter will be imported.
- **COMMENT** - Defines one or more characters, which precede a comment line. Multiple characters/character pairs are separated by a space. *Example:* **COMMENT @ # //** - means that any line beginning with '@', '#' or '//' is a comment line.
- **CONFIGURATION** - Specifies the format for which this translation file has been defined. (*Gerber, Excellon, HP-GL or Sieb&Meyer*).
- **COOR_MN** - Specifies the *m.n* code of the Gerber format. The appropriate values are specified here if the Gerber data are always created with the same number of integer and decimal places.
- **COOR_MODE** - Specifies the coordinate mode of the Gerber format. **ABSOLUTE** mode is selected for Gerber data with absolute values, **INCREMENTAL** mode - for relative values.
- **COOR_UNITS** - Specifies the unit for the Gerber data, imperial (**INCH**) or metric (**MM**).
- **COOR_ZEROS** - Specifies the preference for zeroes in the Gerber format. Set this parameter to **LEADING**, if the Gerber data are always created with leading zeroes, or to **.TRAILINGING**, if trailing zeroes are always used. Select either **.TRAILINGING** or **LEADING**, if both of leading and trailing zeroes are exported.
- **REQUIRED_STRING** (only for automatic detection) - Specifies the keyword that the aperture list must contain to narrow the search in automatic detection of the translation file. The scan process continues after finding the keyword to find the most suitable translation file. All translation files lacking this keyword will be ignored immediately

in the scanning process. This can be used to restrict the range of translation files to be considered.

- **KEY_STRING** (only for automatic detection) - It is similar to **REQUIRED_STRING** but prevents other templates to be checked at all. If a translation file with this keyword is found, the scan process on other template files will be ended immediately and this translation file will be used.



Note: If more than one file has the same KEY_STRING, checking process will be cancelled as soon as such files are located (user gets corresponding message).

Aperture Definition

Every aperture in the aperture list should be assigned precisely by one of the entries in the aperture definition of the translation file. A code, a mode, a form, and a size distinguish the apertures from each others. Each line in the aperture list is checked for a match with the entries in the translation file. The aperture information will be converted, if a match is found; otherwise the aperture will be ignored.

The following syntax is used for the aperture definitions:

1,<DRAW_FORMAT><FLASH_FORMAT>:<APE_FORMAT>"

<DRAW_FORMAT> parameter	Meaning
N	An aperture is only used as a flash aperture, never as draw aperture.
C or R	A line is drawn with round End Type and round Corner Type aperture.
Q	A line is drawn with extend End Type and extend Corner Type aperture.
T	A line is drawn with truncate End Type and truncate Corner Type aperture.
X	A line is drawn with extend End Type and round Corner Type aperture.
Y	A line is drawn with truncate End Type and round Corner Type aperture.
Z	A line is drawn with round End Type and extend Corner Type aperture.
U	A line is drawn with truncate End Type and extend Corner Type aperture.
V	A line is drawn with round End Type and truncate Corner Type aperture.
W	A line is drawn with extend End Type and truncate Corner Type aperture.

<FLASH_FORMAT> parameter	Meaning	Size parameter in <APE_FORMAT>
N	An aperture is only used as a draw aperture, never as flash aperture.	
C	An aperture is used as round or circular flash aperture.	%a (diameter)
Q	An aperture is used as square flash aperture.	%a (side length)
R	An aperture is used as rectangular flash aperture.	%a (side length X)%b (side length Y)
O	An aperture is used as an oval or finger-shaped flash aperture.	%a (side length X)%b (side length Y)
T	An aperture is used as an octagonal shaped flash aperture.	%a (diameter)
D	An aperture is used as rectangular flash aperture with rounded corners.	%a (side length X) %b (side length Y)%c (corner radius)
B	An aperture is used as rectangular flash aperture with cut corners.	%a (side length X) %b (side length Y)%c (cut length)
M	An aperture is used as a marker shaped flash aperture.	%a (diameter)
U	An aperture is used as an undefined shaped flash aperture.	
S	An aperture is used as special flash aperture. The actual form must be designed and implemented in <i>CircuitCAM</i> .	

<APE_FORMAT>parameter is shown in quotation marks ("") and defines the exact reproduction of each aperture (aperture line) in the aperture list. Both fixed and variable entries may be used here. The space is used as the default separator between the various parameters. Both a decimal point "." and a comma "," can be used in numbers.

Variable entries	Meaning
%%	is used for the aperture code
%s	ignores a string from space to space(may be used as often as necessary)

%a %b	are used for size parameters (diameter, side length)
Separator	
In the aperture file non-white-space characters are used for the variable entries separation, but in the translation file these separators must be provided in square brackets.	
Note: The separator can be used only between two variable entries (which have a format characters %).	
Fixed entries	
All strings appearing between the variable values in the aperture list are permitted as fixed entries. Multiple consecutive spaces in the aperture list are represented as a single space in the translation file.	
Tip: All entries after the last relevant parameter can be ignored. (There is no need to use the %s variable for this.)	

Variable Parameters

Some CAD systems are capable of writing more precise specifications of the *Gerber* format to the aperture list in addition to the aperture information. Variable parameters may be used for the evaluation of this information so, that it is not necessary to modify the translation file for each change in this supplementary information.

The aperture list will be searched during the conversion process for the strings defined for the variable parameters and assigned to the appropriate parameter where a match is found.

Variable parameter	Meaning
Coordinates unit	
100 : "<text>"	The string entered for <text> defines INCH as the coordinates unit.
101 : "<text>"	The string entered for <text> defines MM as the coordinates unit.
Coordinate mode	
110 : "<text>"	The string entered for <text> defines RELATIVE as the coordinates mode.
111 : "<text>"	The string entered for <text> defines ABSOLUTE as the coordinates mode.
<u>Integer and decimal places</u>	
120 : "<text>%d<text>"	The string entered for <text> defines the M code (number of integer places).& d stands for the digit that defines the M code in the aperture list.
121 : "<text>%d<text>"	The string entered for <text> defines the N code (number of decimal places).& d stands for the digit that defines the N code in the aperture list.
122 : "<text>%1d.%1d<text>"	The string entered for <text> defines the M.N code (number of integer and decimal places). The first variable &1d stands for the digit that defines the M code in the aperture list. The second variable &1d stands for the digit that defines the N code in the aperture list.
<u>Coordinate zeroes</u>	
130 : "<text>"	The string entered for <text> defines the coordinate zeroes as LEADING .(trailing suppressed.)
131 : "<text>"	The string entered for <text> defines the coor-

	dinate zeroes as TRAILING .(leading suppressed.)
132 : "<text>"	The string entered for <text> defines the coordinate zeroes as DECIMAL .

Example

Translation file:

APE_UNITS MM

APE_SCALE 0.01

CONFIGURATION ConfigGerber

1,CC : "Aperture D%% %a %s ROUND DRAW + FLASH"

1,CN : "Aperture D%% %a %s ROUND DRAW"

1,NC : "Aperture D%% %a %s ROUND FLASH"

1,QN : "Aperture D%% %a %s SQUARE DRAW"

1,NQ : "Aperture D%% %a %s SQUARE FLASH"

1,NR : "Aperture D%% %a %b RECTANGLE FLASH"

101 : "scale in mm"

111 : "ASCII data absolute"

122 : "%d.%d digits"

131 : "use trailing zero"

Associated aperture list (part of the file):

GERBER PRODUCTION INFO

ASCII Data absolute :

3,2 Digits m.n

Scale in mm

disable G - Codes

use trailing Zero

Blendenma e in 1/100 mm

APERTURE CODE WIDTH LENGTH

APERTURE D10 15 15 ROUND DRAW + FLASH

APERTURE D11 10 10 ROUND DRAW

APERTURE D12 100 100 ROUND DRAW

APERTURE D13 160 160 ROUND FLASH

APERTURE D14 150 150 ROUND FLASH

APERTURE D15 20 20 ROUND DRAW

APERTURE D16 90 90 ROUND FLASH

APERTURE D17 320 320 ROUND FLASH
 APERTURE D18 70 70 ROUND DRAW
 APERTURE D19 100 100 SQUARE DRAW
 APERTURE D20 10 10 ROUND FLASH
 APERTURE D21 40 40 ROUND DRAW
 APERTURE D22 232 232 SQUARE FLASH
 APERTURE D23 120 120 ROUND FLASH
 APERTURE D24 50 50 ROUND DRAW
 APERTURE D25 170 170 SQUARE DRAW

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

Aperture Template Editor

It was explained what a translation file is, and how to write it manually in the [Aperture files and templates \(translation files\)](#) section. But it is not so easy to write a template file for beginners. From the other hand, this process can't be absolutely automated because only a user can tell, which data are really needed from the aperture/tool file.

A graphical *Aperture Template Editor "ATE"* is a tool to generate template files from aperture files. The *ATE* can be used either as stand-alone application or called directly from the *CircuitCAM* when it is needed (the *Template Editor* button during standard aperture/tool list import or selection-button *Create a new translation file* in Wizard import ----> **TBD**).

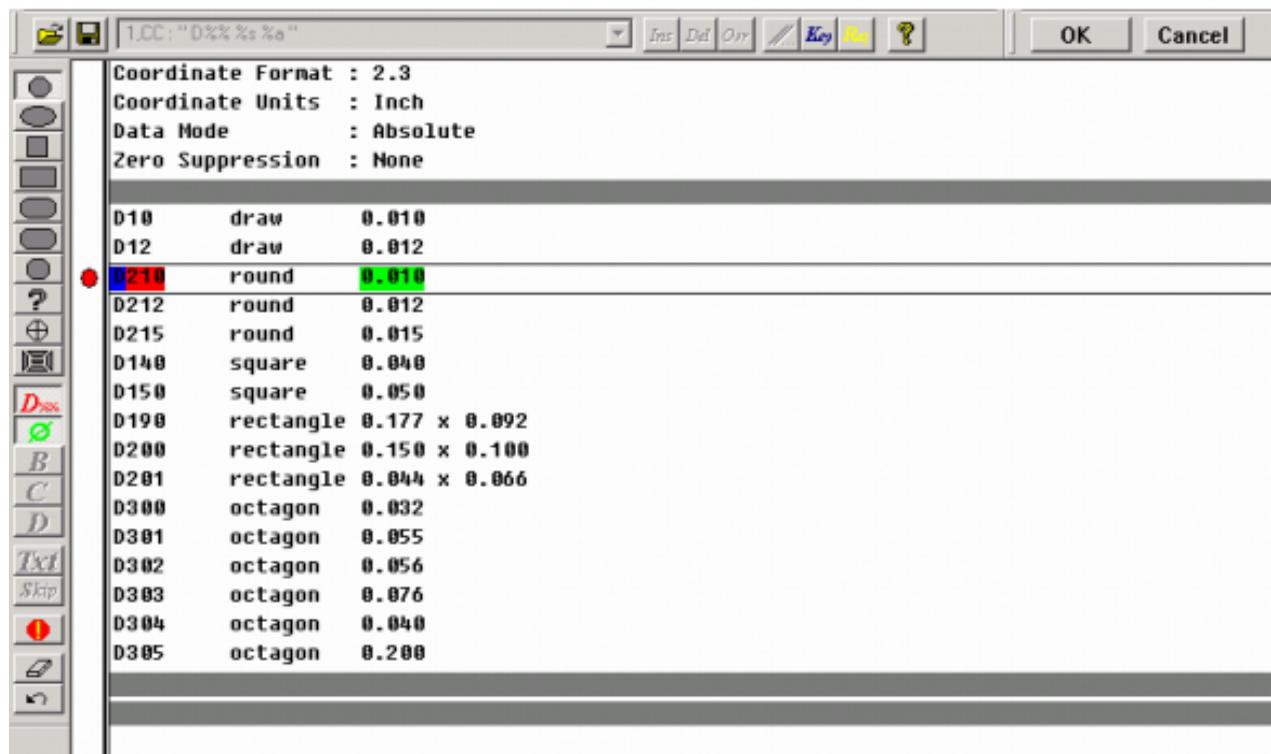


Figure 315. Main ATE window

There are some menu commands, and three toolbars around. The centre part shows an aperture/tool file, for which the translation file is to be created. The template writing scenario in outline is as follows:

- Select a needed tool-line (e.g.: line "D10 Draw Circle 20");
- Press a tool shape button (e.g.: Circle);
- Select tool numbers (e.g.: D10) and press D-code button;
- Select a tool dimension (e.g.: 20) and press Argument-A button;
- Press **Accept template line** button while it is already enable - then the selected line and all similar lines will be colored in grey. It means, that a template format for these tools is generated;
- Repeat same actions for other (not colored) lines in the aperture/tool file;
- Press **OK** button after the editing of a template file is finished (or **Cancel** to eject).

Shape & size toolbar

The toolbar contains buttons to code a tool.



Figure 316. ATE Shape&Size toolbar

10 first buttons correspond to all shapes of the tool, supported by *CircuitCAM*: circle, oval, square, rectangle, rounded rectangle, bevelled rectangle, octagon, undefined, marker and special.

D_{xx} - **D-code:** set tool number.

Next four buttons **Argument-A**, **Argument-B**, **Argument-C** and **Argument-D** define the tool size. Depending on what type was selected for the tool, these buttons have different look:

	For shapes with one parameter (\leq = diameter for circle and octagon);
	For shapes with one parameter (square, marker);
	For shapes with two parameters (oval, rectangle);
	For shapes with three parameters (rounded rectangle, bevel rectangle);
	For special apertures (up to four values).

Txt - **Embedded-Text:** Includes the selected text into template format. It allows a user to use more precise format. E.g.: for both lines "D10 Draw Circle 20" and "D13 Flash Circle 20" the template line "1,CC: "D%% %s %s %a"" is valid. But the text "Draw" by using **Embedded-Text** button is added for D10, the format line already will be "1,CC: "D%% Draw %s %a"" and do not fit for D13. To separate D10 and D21 needs to include text "Circle" additionally.

Skip - Skips the selected text by adding it into the square brackets [].

Erase - Cleans all selections (after the last pressing on only).

Undo - Undo button.

Accept template line: Enable state (!) indicates that the format line is already ready for usage. You still can update not specified fields (like texts with **Embedded-Text** button) or already now press this **Accept template line** button. Tools (lines), which description corresponds to this template, will be made in grey.

Edit toolbar

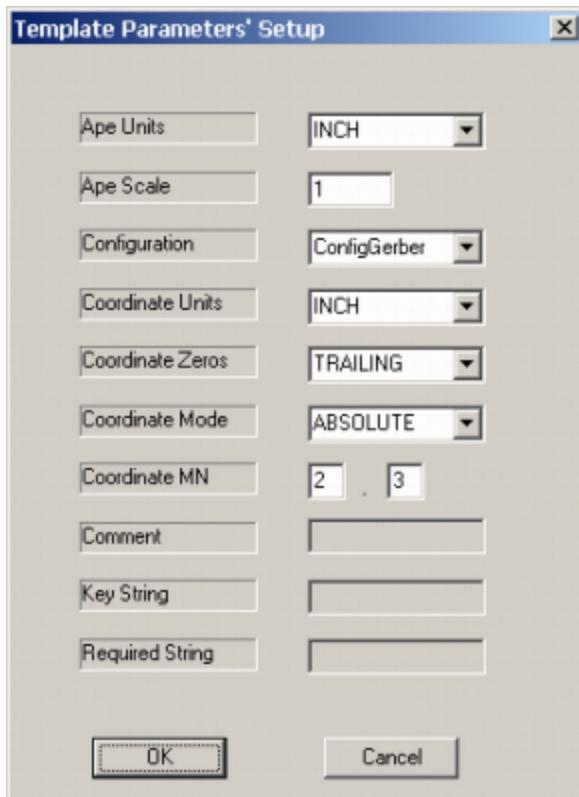


Figure 317. ATE Edit toolbar

During creation of a template file, all generated lines are kept into combo box, where user can edit them. It is allowed to add/write format lines manually as well. For this the button **Ins** inserts (duplicates) the selected template line, **Del** - deletes the selected template line and **Ovr** - overrides (Apply) the selected line after it is modified or written anew (buttons **Ins** **Del** **Ovr**). **Ovr** function automatically scans a tool file for this new template line and marks recognised lines in grey. Next group of buttons **/... Key Req** can be used to insert comment lines (**Comment-Characters** button) add special key strings (**Key-String** and **Required-String** buttons). For more information about comments and key strings see in [Aperture files and templates](#) clause.

Closing toolbar

To save the generated template into the specified file, press **OK** button in the **Edit** toolbar (right-top corner of ATE window). After selection *Yes* into question "Template file was modified. Do you want to save changes?", the final dialogue box of **Fixed parameters** appears.

**Figure 318. ATE, Template Parameters' Setup dialogue window**

Any needed changes for aperture/coordinate units, scaling, zeros mode and others, can be done here. For more information see in [Aperture files and templates](#) clause.

Pressing on **OK**. The standard **Save As...** dialogue comes up to specify the location and name of the template file. After this ATE window is closed and *CircuitCAM* returns into state where ATE was called from.



Note: While ATE window is opened all operations on *CircuitCAM* are blocked.

Barco-DPF

The current version of **Barco-DPF** command language is described in:
DPF Format Description, Version 5
November 1998

See also the BARCO-ETS web site

<http://www.barco.com>

<http://194.7.253.113/ets/>

Additional information can be found here:

<http://www.maniabarco.com/transdown.asp>

Units:

U=unit

Defines the DPF-unit. The possible units are:

MM : millimetre (metric)

INCH : inch (imperial)

MIL : mil (thousandth of an inch)

Head Information:

Xmin,max

Minimum and maximum of job in x.

Is ignored.

Ymin,max

Minimum and maximum of job in y.

Is ignored.

Informational Data:

;anything

Rest of the line are COMMENTS.

A line is terminated by carriage return (CR) or a line feed (LF).

Pn=x,y

POINT "n" at position x,y. Is ignored.

Plot Data:

Fx,y

FLASH on position x,y.

Mx,y

MOVE to position x,y.

Dx,y

DRAW to position x,y.

Cx,y,mx,my,[i]

DRAW ARC from the current position to x,y.

The centre of the arc is mx,my.

"i" specifies the direction of the arc:

i = CW : clockwise direction

i = CCW : counter clockwise direction (default)

These Commands are Not Supported:

Vx,y,"text",width:space,fontname,(vector text expansion)[,options]

VECTOR TEXT

"x" and "y" are the start position of the first letter in the text.

"text" can be any string, just like in the normal DPF text aperture.

"width" is the horizontal offset between the starting points of two consecutive characters, in this case it should be a mono-spaced font. If width is 0 then a "space" parameter defines the white space between two characters.

"fontname" is the name of the font (no pathnames allowed) and it is searched on UFNTDIR. The vector text expansion section defines the actual draws of the text, so that you don't need the font file to visualise the DPF file.

"options" are mirror, rotate and scale like for normal DPF apertures.

Nn[,name="value"]*

Netlist data:

"n" stands for net number.

"name" is a normal ASCII string, no special characters allowed.

"value" is an ASCII string.

If the value contains special characters (like ',', ','), the value should be declared between double quotes.

Remarks

- The net number is modal.
- The netlist data is generated automatically by the netlist generation program or the design rule check program.

Gn[,options]*

Group data:

"n" stands for group number.

"options" are the same as for netlist data.

Aperture Definitions:

Aa [=def [,options]]

Selects an aperture with a number "a". The number "a" is a positive number.

If "=def" is specified, the aperture is (re) defined. Options can only be specified when a definition is specified.

Possible definitions are:

- Circle aperture

C*IRCLE,d

"d" indicates the diameter of the circle.

■ **Square aperture**

S*QUARE,s

"s" indicates the size of the square.

■ **Rectangle aperture**

R*ECTANGLE,sx,sy

"sx" indicates the size of the rectangle x-direction.

"sy" indicates the size of the rectangle y-direction.

■ **Donut aperture**

D*ONUT,douter,dinner,kind

"douter" indicates the outer diameter of the donut.

"dinner" indicates the inner diameter of the donut.

"kind" indicates the type of the donut:

rr (round donut)

ss (square donut)

sr (square-round donut)

■ **Box aperture**

BO*X,sx,sy,corners=xcutoff[:ycutoff]

"sx" indicates the size of the box in x-direction.

"sy" indicates the size of the box in y-direction.

"corners" indicates the shape of the corners of the box: R*OUNDED, S*TRAIGHT,A*NTIQUE or C*UT.

"xcutoff" indicates the size of the corners in x-dimension.

"ycutoff" indicates the size of the corners in y- dimension (defaults to xcutoff if not specified).

■ **Octagon aperture**

OCTAGON,size

"size" indicates the size of the octagon.

■ **Thermal aperture**

THE*RMAL,outer,inner,gap,ngap,startangle,kind

"outer"outer diameter (or size)

"inner"inner diameter (or size)

"gap"size of gap

"ngap"number of gaps

"startangle"angle of first gap in degrees

"kind"rr,rs,ss,sr (r=round and s=square)

■ **Text aperture**

T*EXT,("text_string") [| height |] [:width |]

"text_string" indicates the string of text characters; note that the string must be enclosed by quotation marks ("").

1. if the string contains round brackets, enclose them within quotation marks ("").
2. if the string itself contains a quotation mark, specify it twice to differentiate it from the start and stop quotation marks.

"height" indicates the text width.

The width of the text defaults to 84.21% of the text height. That means - in case no text width is specified 84.21% of the text height is assumed.

■ Contour aperture

CO*NTOUR [,ST*ROKE=st]

A contour aperture is considered as a pen with size 0. This aperture is used to draw contour lines. After the selection of a contour aperture you can add an enumeration of contours described by means of move, draw and arc. If there is an overlap of two contours, the common zone is masked out (white).

You can add the option "stroke" which results in a contour with a solid line on the top of the contour. The stroke value is the width of this line.

■ Complex aperture

COM*PLEX,cdef

The definition of the complex aperture can be specified as:

1. Enumeration of contours: < cdef > == (< contour > [< contour > ...])< contour > == Mx0,y0Dx1,y1...Dxn,ynDx0,y0Contour lines are defined by normal DPF-commands: move, draw and arc. Each move defines the start point of a contour; draw and arc define the other points of the contour.
2. Keyword A followed by an aperture number of another complex aperture: < cdef > == Ac"c" is the aperture number of the complex aperture.

■ Block aperture

B*LOCK,blockdef

The definition of the block can be specified as:

1. Enumeration of DPF-commands enclosed between round brackets: < blockdef > == (DPF-commands)
2. Keyword A followed by the aperture number of another block: < blockdef > == Ab"b" is the aperture number of the complex block

Remark:

All apertures (except contours) can be flashed. Circle, donut, square and rectangle apertures can be used to draw lines and arcs.

Aperture Options

Options are separated by commas. Possible options are:

M*IRROR=m

m=X: to mirror along the x-axis.

m=Y: to mirror along the y-axis.

m=XY: to mirror along both x- and y-axes.

R*OTATION=r

"r" is an angle specified in degrees. A positive value indicates a counter clockwise rotation; a negative value indicates a clockwise rotation.

S*CALE=s

"s" is a scale factor.

RE*VERSE

Plots white on back.

Aperture Options Which are Not Supported:

P*ATTERN=type [:step [:width]]

type=V : vertical bars

type=H : horizontal bars

type=VH : both horizontal and vertical bars

type=L : diagonal bars with a top on the left

type=R : diagonal bars with a top on the right

type=LR : both left and right diagonal bars

type=C45 : 45 degrees circular dot pattern

type=C90 : 90 degrees circular dot pattern

type=B : 90 degrees square dot pattern

"step" indicates a distance between two units (the default step is 10 mil)

"width" indicates a width of the unit (the default width is 5 mil).

A*TTRIBUTES=(name="value"[,name="value"[..]])

N*AME="value"

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

CAM

The native *CircuitCAM* format, *CAM* files are made to store the 2D graphic information of bare PCBs. The following options are realized for *CircuitCAM*:

- Characteristics of plotters, milling, and drilling machines needed for PCB-production, together with their tools/apertures and converters to/from these formats: *Gerber*, *Excel-lon*, *HP-GL*, *Sieb&Meyer*, *PostScript*, *IEC1182 (IPC-D350D)*.
- Basis graphic objects: circle, rectangle, path, closed-path, polygon with line- and circular arc-segments.
- Specification of the corner and end-type round, truncate, extend and a width for path and closed-path.
- Complex graphic objects made from basis or other complex objects.
- Business information about the user, organization, time etc.
- Binary exchange format by default.
- Optional ASCII-exchange files with compression.
- Hierarchy structures of a main- and sub-layout.
- Differentiate between fabricated layers and logical levels.

- Support for the design-level and derived-level.
- Set operations union, difference, intersection and inverse on levels.
- Specification of the layer-structure and passages through them (via).
- Classification of electrical (DC) and dielectric layers.
- Support for physical nets and sub-nets.

The following options are **outside** the scope of the *CAM* format:

- Material characteristics of layers.
- External components soldered onto the PCB.
- Assembly of external components to the PCB.
- Hierarchy structures of mother- and daughter-board.
- Wiring.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

CAT

CircuitCAM template files have the same internal format as *CAM* files, but have the file extension ".cat". The purpose of *CAT* files is to provide default layers, configurations, pads, tools, default jobs, etc., but usually no graphics. So the main layout is empty.

For example, if you use the same aperture and tool list regularly, you can create a template file, which contains both lists. You will not need to import aperture and tool lists again for new circuit boards.

Each *CAM* file is based on a *CAT* file. The name of this *CAT* file is kept in the Info section of the *CAM* file. you can see this in the properties view of the *CAM* file

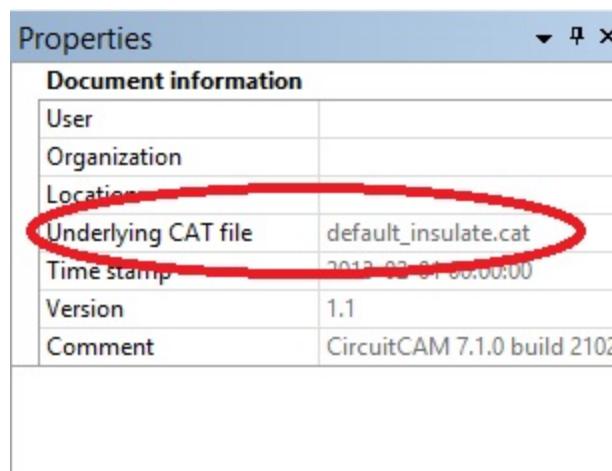


Figure 319. Properties of a CAM file, referring to the underlying CAT file.

In the case the underlying *CAT* file of a *CAM* file has changed, the *CAM* file can be updated for the changes in the *CAT* file with the [synchronize](#) command.

Available template files are listed in the **New** submenu of the **File** menu.

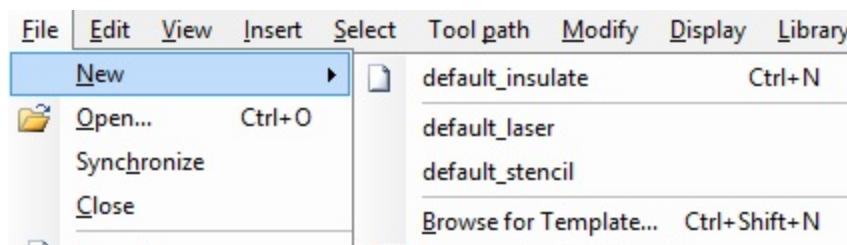


Figure 320. File -> New menu item and list of templates

The *default_insulate.cat* file is used by any unspecified new operation, e.g.: when *CircuitCAM* starts. The default *CAT* file is defined in the [General Settings dialogue](#), **Default Template Name** field of the **Directories** page.

New *CAT* files are stored in a special sub-directory indicated in the [General Settings dialogue](#) window, **New Template Directory** field of the **Directories** page.

To save the existing *CAT* file or to create a new one, select **Save As** from the **File** menu and choose the **Template (*.cat)** line from the **Save as type** field:

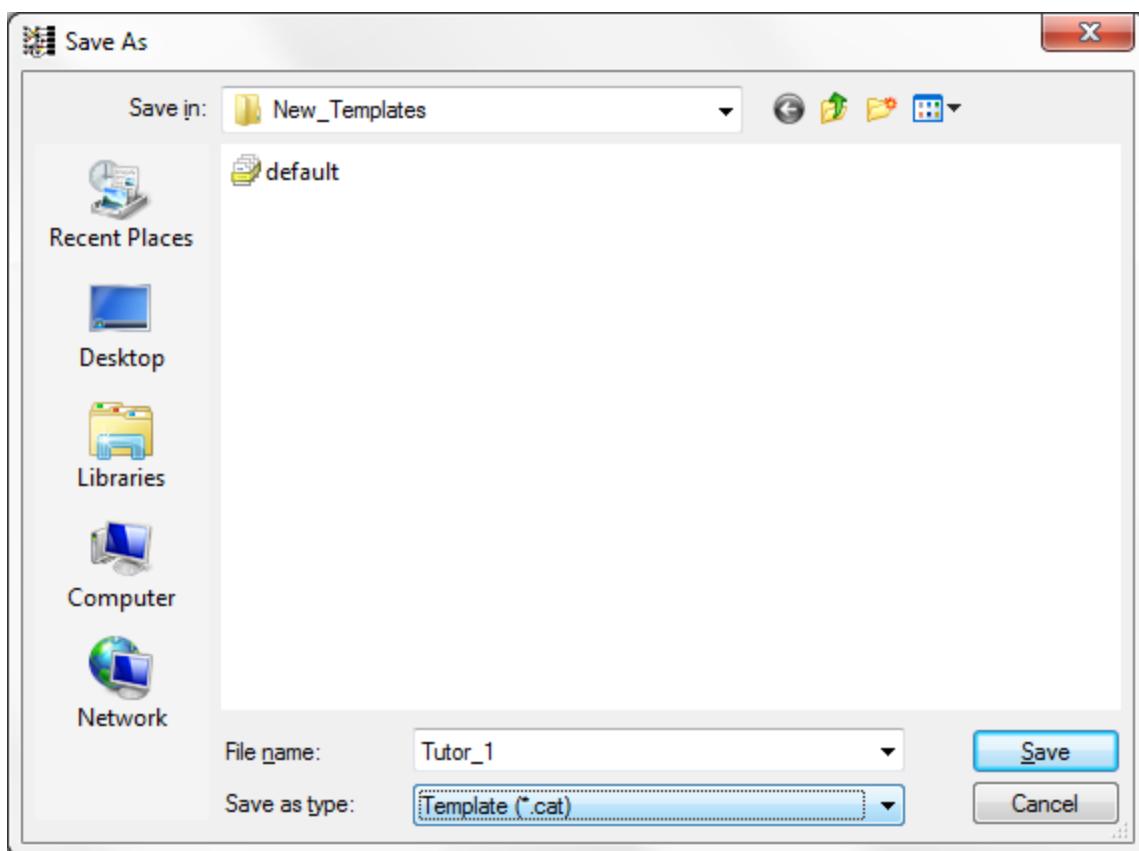


Figure 321. Save As dialogue window to save CAT files

During the **Save** of an actual document as a *CAT* file, the [General settings](#) are automatically stored in the *CAT* file.

Pressing on the **OK** in the **General Settings dialogue** writes the *General* and *Unit\Grid* information into *underlying CAT* file. The file name is taken from the active document. If there is no any active document, this information is saved into *default_insulate.cat* file.

During the **Open** of a *CAT* file, some [General settings](#) assignments are reset according to values of the *CAT* file. If *CAT* file does not have these data, then no changes are made. **User Information** and **Directories** pages of the **General settings** will not be changed because the *CAT* file does not keep such data.



Note: Any changes in **General settings** dialogues are automatically stored into the **Underlying CAT file**. If the current *CAM* file was created on other PC, the saving may be not possible because of different paths to the CAT file. In such case the warning message “**Invalid name of underlying CAT file. General Settings Assignments are not saved**” comes up.

When *CircuitCAM* is running and any document is closed or any *CAM* file(s) is opened, **General settings** and **Unit\Grid** information are neither changed, nor saved.



Note: When *CircuitCAM* starts (with or without any *CAM* file), **General settings** and **Unit\Grid** information are retrieved and set from the registry.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

DXF

See the DXF - AutoCAD™ documentation at <http://www.autodesk.com/techpubs/autocad/dxf/>.

Only the 2D geometry in DXF files is supported (not 3D).

Supported DXF versions are:

- AutoCAD 2000 DXF Reference - v.u15.0.02 ,
- AutoCAD Release 14 DXF Reference - v.u14.1.04.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

EESOF

EESof is an EDA System for High Frequency applications from Agilent Technologies, see <http://we.home.agilent.com/>.

The Mask Files (extension.msk) can be imported into *CircuitCAM*.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

Excelon

Speaking of the **Excelon** format usually mean the part programming codes of the CNC-2, 4, 5, 6, or 7 Excelon machines. The actual CNC-7 documentation is available at <http://www.excellon.com>.

There are two slightly different command structures (format 1 and format 2) for *Excelon* milling and drilling machines. Since *CircuitCAM* does not support all *Excelon* commands, there are no differences for these formats except the occasional switchover from mill to drill operation.

The length details in *Excelon* data (X, Y, A) are given without any decimal point in inches or in millimetres with various digits after the decimal point. Either leading or trailing zeros are also output. Length details can be absolute or incremental. The precise coordinate format is set with the *Gerber/Excelon* dialogue box, also see [excelon format properties dialogue](#).

Reading Production Data:

A...	Radius for arcs.
F...	Milling speed; is ignored
G00	Enable milling operation.
G01	Straight milling.
G02, G03	Mill arc clockwise, anti-clockwise.
G81	Switches the machine from mill to drill operation, format.
G05	Switches the machine from drill to mill operation, format 2..
G90	Absolute operating mode.
G91	Incremental operating mode.
G40, G41,	Compensation.
G42	
G93	Zero point.
G...	Other G commands produce a warning message.
M48...%	Program header.

The Following Commands are Supported in Program Header (Until %):

	Tool selection:
Tnn[Cr]	<ul style="list-style-type: none">• nn – aperture number• C – aperture circle• r – diameter
	Sample: T05C0.059055
	Circular tool, nn=05, diameter r = 0.059055
	Specify metric units for the plot data.
INCH[,{LZ,TZ}]	<ul style="list-style-type: none">• LZ – leading zeroes omitted• TZ – trailing zeroes omitted
	Specify metric units for the plot data.
METRIC[,{LZ,TZ}]	<ul style="list-style-type: none">• LZ – leading zeroes omitted• TZ – trailing zeroes omitted
	Format statement.
FM{L,T},n,m	<ul style="list-style-type: none">• L – leading zeroes omitted• T – trailing zeroes omitted• n – N format• m – M format
M71, M72	Imperial or metric length details.
M25	Start of step and repeat.
M01	End of Step and Repeat macro.
	Macro offset.
	<ul style="list-style-type: none">• X,Y – offset values
M02XY[M70,M80,M90]	<ul style="list-style-type: none">• M70 – swapping of X and Y axes: X become value Y and vice versa• M80 – mirror image of X value: +X becomes -X• M90 - mirror image of Y value: +Y becomes -Y
M08	End of Step and Repeat.
M0, M6, M9, M15, M16,	Is ignored.

M17, M30, M47

M...	Other M commands generate a warning message.
S	Spindle speed (rpm), is ignored.
R#	Repeat hole (both forms R#X#Y# and X#Y#R# are supported).
X...Y...	Coordinate details.
/, %	Read past block to line end.
Line end	End of a command unit.

Writing Production Data:

	%
Start:	M70 or M71
	G90 or G91
Tool selection and compensation index:	T#(#)
Start of a path:	G01X...Y...M15
Line segment:	G01X...Y... 1. Arc Radius mode A#: G02 or G03 and then X#Y#A# 2. Arc Centre Offset mode I#J#: G02 or G03 and then X#Y#I#J#
Arc segment:	M17
End of a path:	G00 perhaps, G05 to switch back from mill operation to drill operation; perhaps, T... to select another tool X...Y...
Flashing a tool:	
End:	M30

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

G-code

Common used numerical control (NC) programming language that is used in automation, it is part of the computer aided engineering used for the 3-6 axes CNC machines. It is a basic language which is defined by set of instruction (see below) according to the standard ISO-6983 that tells the machine where to move, which path and with what speed.

Example:

See [G-Code Format Properties Dialogue](#) for the example setup

This example shows a sample of the generated text for the G-Code file.

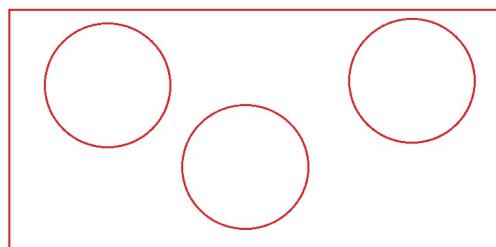


Figure 322. Example of drawn PCB.

This is what the ".NC" file look like according to standard ISO-6983 :

```
( G-Code file generated by CircuitCAM 7.1.0.2103 )
G21 programing in mm
G90 Fixed cycle, simple cycle, for roughing (Z-axis emphasis)
G64 profiling mode
G40 Tool radius compensation off
G17 XY plane selection
G00 Z0 Rapid positioning on the Z-axis
( T01 : 0 ) Comments are enclosed in parenthesis
M6 T01 Automatic tool change
G00 X17.1617 Y-17.4642 Rapid positioning on xy axis
G01 Z-5 Linear interpolation, z-axis down
G03 X-32.8383 Y-17.4642 I-25 J0 Circular interpolation
G03 X17.1617 Y-17.4642 I25 J0 anti-clockwise
G00 Z0 Rapid positioning, Z- axis up
G00 X-88.9714 Y55.6931
G01 Z-5
G01 X77.6953 Y55.6931
G01 X77.6953 Y-52.9428
G01 X-89.2464 Y-52.6678
G01 X-88.9714 Y55.6931
G00 Z0
G00 X66.6667 Y20.7646
G01 Z-5
G03 X16.6667 Y20.7646 I-25 J0
G03 X66.6667 Y20.7646 I25 J0
G00 Z0
G00 X-32.3432 Y20.4895
G01 Z-5
G03 X-82.3432 Y20.4895 I-25 J0
G03 X-32.3432 Y20.4895 I25 J0
G00 Z0
M05 spindle stop
M30 End of program, with return to program top
```

Figure 323. G-Code example.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

Gerber X

The **Extended Gerber** format also called **Gerber X** carries addition information, which is missing in the simple *Gerber* format. The main important information is a complete definition of apertures (see AD and AM commands below). In addition it may contain information about layer name and the polarity of layers.

For further information see http://en.wikipedia.org/wiki/Gerber_format.

AD and AM commands:

% ADDnnMacro[P1XP2XP3...]* %	Mass format macro aperture definition: nn – aperture number
% AM Macro*type, on/off, \$1,\$2....* %	Macro – the name of macro that must be previously described by an AM command. P1, P2, P3... – macro parameters. Mass format macro definition: Macro – the name of macro type – one of the seven primitives listed in Table C – 2 on/off – 1=ON (laser on, film exposed) 0=OFF (laser off, no exposure)
	\$1,\$2,\$3 – are position indicators and should be numbered sequentially in the order that the macro expects the values to appear in the AD command. Parameters are separated by X in the AD command.
% ASA{X,Y}B{X,Y}* %	Mass format axis select: A {X,Y} – assign plot data coordinates to the plotter A axis B {X,Y} – assign plot data coordinates to the plotter B axis. default: AXBY also supported: AYBX.
% IN Name* %	Mass format for image name: Name – the name of image. The command will generate message into the report window.
% IOA±mB±n* %	Mass format image offset: A ±m – X offset value ±m B ±n – Y offset value ±n.
% IR rotation* %	Mass format image rotation: rotation – 0, 90, 180 or 270.
% LN Name* %	Mass format Layer Name: Name – the name to be assigned to the layer. If one LN command is used, the name from Import Dialogue Box is taken. If few LN commands are used, a few layers are created with the names supported by LN commands.
% MIA{0,1}B{0,1}* %	Mirror Image: A {0,1} – mirror feature for data on the plotter A axis (0 – disables, 1 – enables). B {0,1} – mirror feature for data on the plotter B axis

%MO{IN,MM}* %

(0 – disables, 1 – enables).

Mass format specifies the units of the plot data:

IN – English units

MM – metric units.

Mass format Scale Factor:

An – scale factor for the plotter A-axis

Bn – scale factor for the plotter B-axis

% SFAnBn* %

Command SF enables you to specify, independently, scale factors for

the plotter A and B axes.

Mass format Step and Repeat:

X count – number of times to repeat the image in the X axis

Y count – number of times to repeat the image in the Y axis

% SR[X count][Y count][I increment][J increment]

I increment – distance in inches or millimetres between the lower left corner of original/source object and the lower left corner of newly created (repeated) object. Distance is measured parallel to X axis.

J increment – distance in inches or millimetres between the lower left corner of original/source object and the lower left corner of newly created (repeated) object. Distance is measured parallel to Y axis.

The image is created as one flash.

This flash is plotted according to SR parameter.

RULES for SR commands with LN and LPD/LPC: (2001-11-27):

1. A new SR block is started only with SR, which either number of repetition on X or on Y is greater than 1. If there are a few SR in the sequence, only the last SR starts the next repetition block. All SR in between are ignored.
2. The SR block is ended with:
 - a. SR command SR*%;
 - b. any new SR command:
 - if a new SR has at least one number of repetition > 1;
 - if starts a new SR block (it goes from the rule (1));
 - c. M02 command;
 - d. end-of-file.
3. LN (Layer Name) does not end SR, but all next objects will be repeated on other layer.

% LPD*%

Layer Polarity Dark (default value) applies to all data following the LP parameter until another LP parameter is encountered.

% LPC*%	Layer Polarity clear (opposite parameter to LPD) applies to all data following the LP parameter until another LP parameter is encountered.
%OFA6nB6n*%	Offset the final image along X and Y axis. Default: %OFA0B0+%

The Following Commands are Not Supported:

AA	(Aperture Assignment)
AF	(Auto Forms)
AP	(Aperture Offset)
AR	(Aperture Record)
AV	(Aperture Velocity)
BD	(Block Delete)
DL	(Dash Line Specification)
IC	(Input Code)
ID	(Input Display)
IF	(Include File)
IJ	(Image Justify)
IPNEG	(Image Polarity Negative)
KO	(Knockout)
LS	(Load Symbol)
NS, NF	(Sequence Numbers)
OP	(Optimal Stop)
PE	(Perspective)
PF	(Film Type)
PK	(Park)
PO	(Pen Offset)
RC	(Rotate Symbol)
RO	(Rotate Position Data)
SC	(Single Step Mode)
SM	(Symbol Mirroring)
SS	(Symbol Scaling)
TR	(Translation)
VL	(Velocity Limit)
WI	(Window Specification)

At Gerber(X) File Output, The Following Commands are Used:

Format statement; leading zeroes suppressed, absolute Coordinate values:

Configuration:	%FS{L, T, D}{A, I}Xn,m*%
Units:	%MO{IN,MM}* %
Layer name:	% LN Name* %
	%ADDnn{C, R, O, P} value1[X value2]*%
Apertures definition:	% AMMacro*type, on/off, \$1,\$2...* %
	% ADDnnMacro [,P1XP2XP3...]* %
Tool selection:	G54D..* or just D..* if the G-Code is disabled.
Start of a path:	X...Y...D02*
Line segment:	X...Y...D01*
Arc segment:	G75*

	G02* or G03*
End of a path:	X...Y...I...J...D01*
Tool flash:	G74*
	G01*
not applicable	
Filling of a Polygon Area:	X...Y...D03*
	G36*
	X...Y...D02*
	X...Y...D01*
	...
	G37*

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

Gerber

The precise syntax of the **Gerber** format is given in the
Gerber FORMATPlot Data Format Reference Book

Tel.: (203) 644-1551; FAX: (203) 649-7157

The actual documentation can be found by the link:

<http://www.maniabarco.com/transdown.asp>

Gerber files can come in many forms, in theory. In practice, however, only a few forms are possible for PCBs which are also supported by *CircuitCAM*.

In this way, only the word address format with ASCII coding can be edited. The other codes (EIA RS-244-B, BCD, EDBC DIC, ISO and ASCII) and the binary format are not supported. Also some commands of the *Extended Gerber Format* (Mass Parameter, Extended RS-274-D) are supported. A typical *Gerber* file could be as follows:

G54D20*	Tool 20 is selected.
G01X01Y00D01*	A straight line is drawn to position x=1, y=0.
G54D10*	Tool 10 is selected
D03*	With this tool, there is a flash to the current position (flash).
G01X-1Y00D02*	The current tool is moved to position x=0, y=0. In this case X-Y data is incremental values
D03*	The tool is flashed at the current position

Gerber files are divided into data blocks, which ends with "*" and in some cases with "\$" too. A simple line end is not considered the end of a data block; it may even be that a line end interrupts coordinate data.

A data block can consist of commands N, G, X, Y, Z, I, J, K, W, C, T, U, V, D, M, in each case followed by a number.

The Following Commands are Supported:

N nn	Continuous numbering of data blocks is ignored.
G01, G10, G11, G12, G60	Line interpolation.
G02, G20, G21	Arc interpolation in clockwise direction.
G03, G30, G31	Arc interpolation in counter-clockwise direction.
G04	Commentary; is ignored to line end.
G36, G37	Filling of a Polygon Area.

G54	Tool selection. Is ignored.
G70	Coordinates are in inches.
G71	Coordinates are in millimetres
G74, G75	360 degree circle interpolation enabled and disabled. If G74 is disabled, arcs to a maximum of 90 degrees only can be described.
G90	Coordinates are given in absolute values
G91	Coordinates are given in relative (incremental) values
G55	Is ignored.
G-other codes	There is an error message.
Xm.n, Ym.n	Coordinates in M.N format.
Im.n, Jm.n	Centre point coordinates of an arc in M.N format.
D0	Is ignored.
D01	Enable tool.
D02	Disable tool.
D03	Flash tool.
D10-unlimited	Select tool 10 to unlimited
D-other codes	There is an error message
Z,K,W,U,V	Not supported. There is an error message.
M0, M01, M03, M30	Are ignored.
M02	End of the file. Is ignored.
M-other codes	There is an error message.
%	Commentary; is ignored to line end Mass format command to set: L - leading zeroes suppressed T - trailing zeroes suppressed D - explicit decimal point
%FS{L, T, D}{A, I}{X,Y}n,	A - absolute coordinate values
m*%	R - relative coordinate values X,Y – leading/trailing zeroes format (one for both X and Y) n -format N m – format M Mass format aperture definition: nn - aperture number
%ADDnn{C, R, O, P}	C - Circle, diameter
value1[X value2]*%	R - Rectangle, width, height O - Oval, width, height P - Polygon, diameter, size

Also see [Gerber format properties dialogue](#)

Hewlett Packard developed the *HP-GL* command language for its pen plotters. Later, it was taken on by other plotter manufacturers and has today become the standard for pen plotters. The current version of this command language is described in:

The HP-GL / 2 Reference Guide
a handbook for program developers
Hewlett-Packard Company, 1990
ISBN 0-201-56308-8
Addison-Wesley Publishing Company, Inc

The number of available commands has constantly increased over time and today fills a whole book. Almost all systems have restricted themselves to a few commands, which can also be imported by *CircuitCAM*. This is done due to the reason that available plotters may support only minimal set of commands.

SP n	Select Pen number n; valid values for n are 1 to 9 and 0. Value 0 means put down one pen without taking a new one.
PD	Pen Down - head down on paper.
PU	Pen Up - head up, away from paper.
PA { x, y}	Plot Absolute - move to the given absolute positions x,y.
PR { x, y}	Plot Relative - move to the given relative positions x,y.
AA x, y, a (, c)	Arc Absolute - draw an arc out from the current position with Centre point x,y and angle a. A value of 360 forms a full circle. With negative a values, the circle is drawn in a clockwise direction and with positive values, in a counter-clockwise direction. A chord angle C can also be entered, which reduces a copy of the arc in line segments. <i>CircuitCAM</i> ignores this value.
AR x, y, a (,c)	Arc Relative - similar to AA but the Centre point is given relative to the current position.
AT xBulge, yBulge, xEnd, yEnd, (c)	Absolute arc through 3 points: Current position, bulge point and end point. A chord angle c can also be entered, which produces a copy of the arc in line segments. <i>CircuitCAM</i> ignores this value
RT xBulge, yBulge, xEnd, yEnd, (c)	Relative arc through 3 points: Current position, bulge point and end point. A chord angle c can also be entered, which produces a copy of the arc in line segments. <i>CircuitCAM</i> ignores this value
CI r (,c)	Circle - a complete circle is drawn around the current position, which forms the Centre point. The value r defines the radius.
EA x, y	Frame of an absolute rectangle. It is defined by two diagonals opposite corner points: the current position and the given absolute x,y position.
ER x, y	Frame of a relative rectangle. It is defined by two diagonals opposite corner points: the current position and the given relative x,y position.
RA x, y	Filled absolute rectangle. It is defined by two diagonals opposite corner points: the current position and the given absolute x,y position.
RR x, y	Filled relative rectangle. It is defined by two diagonals

FT fill_type	opposite corner points: the current position and the given relative x,y position.
LT	Filling type for areas (rectangles). Only the solid types 1 and 2 are supported, otherwise an error message is performed.
DF	Line Type. No parameter allowed for the continuous line, otherwise an error message is given.
IN	Default Values
PW	Initialize
PW width	In this case the pen width is set to 0.35 mm.
PW width, pen	The <i>Path width</i> is set to the provided width. The value of <i>width</i> is an integer number from 1 to 10.
BL, CA, CJ, CT, DJ, DT, FS, IW, LT, NP, PT, PC, PG, PS, SA, SL, VS and the ESC symbol	The <i>Path width</i> is set to the provided width. The pen information is ignored.
All other commands	All this data is ignored.
	An error message is generated.

If your CAD system should use a command, which is not listed, then please ascertain the meaning of the command. It could be an initialisation, a page selection or such like which is irrelevant to *CircuitCAM*. However, if important information lies behind the command, with the result that the data input is incomplete, then please try to generate simpler HP-GL data with your system.

The Following Commands are Used for HP-GL File Output:

Tool selection	SP ...;
Path width	PW ...;
Start a path	PA ..., ...; PD;
Line segment	PR ..., ...;
Arc segment	AR ..., ..., ...;
End of a path	PA ..., ...;
Tool flash	PU; PA ..., ...; PD; PU;

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

ISO 10303-210

ISO 10303 is the International Standard for *Product data representation and exchange*. AP210 of STEP is the Application Protocol for *Electronic assembly, interconnect and packaging design*. *CircuitCAM* data can be converted to STEP-AP210 data, using IDA-STEP.

For more information and links see <http://www.ida-step.net/components/import/circuitcam>.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

Laser Mill Drill (LMD¹)

¹Laser, Milling, Drilling

LMD is a proprietary format in binary form. Its only purpose is to have an optimal connection from *CircuitCAM* to *BoardMaster* to drive the mill/drill plotter. This format is used to Export *CircuitCAM* data and Import from LMD to *CircuitCAM* format. The default file extension for the format is "***.LMD**".

An LMD-file is split into several **Phase**. A phase contains information for one manufacturing step of one or more *CircuitCAM*-layers. By default the following phases are supported:

- **DrillingPlated:** Drilling information for plated through holes.
- **DrillingUnPlated:** Drilling information for unplated holes.
- **MillingCompSide:** Milling information for the component side.
- **MillingSoldSide:** Milling information for the solder side.
- **RuboutCompSide:** Further milling information for the component side.
- **RuboutSoldSide:** Further milling information for the solder side.
- **Cutting:** Milling or cutting the board frame.

Format Specification

LMD files have a binary format, it means that you can't look to them with a usual text-editor. Possible tools to "see" the contents of an *LMD* file are *BoardMaster* and *CircuitCAM* (re-importing). In case you intend to write your own application to process *LMD* files, the binary format specification is given here for the C programming language.

 **Note:** that the specification may change for future versions of *CircuitCAM* and *BoardMaster*.

```
//----- Basis Type Definition -----  
#define LMD_BLOCK_END 0  
  
#define LMD_BYTE      1 // + unsigned 8-bit value  
#define LMD_SHORT    2 // + signed 16bit integer  
#define LMD_INT      3 // + signed 32 bit integer  
#define LMD_LONG     4 // + signed 64bit Integer  
#define LMD_FLOAT    5 // + 32bit real  
#define LMD_DOUBLE   6 // + 64bit real  
#define LMD_SSTRING  7 // + 0-terminated single-byte string  
#define LMD_DSTRING  8 // + 0-terminated double-byte string  
(unicode)  
#define LMD_BINARY1  9 // + 1byte(=n) + (n/8)bytes;  (n==no.of  
bits,round up)  
#define LMD_BINARY4 10 // + 4bytes(=n) + (n/8)bytes;  
(n==no.of bits,round up)  
  
//----- Extended Type Definition -----  
#define LMD_TYPE_DEF 11 // + byte(=new-type) + int (=parameter_  
size)  
  
//----- Extended Types -----
```

```
//- All commands from 12 to 127 are reserved for extended types
//- All extended types have a fixed length (command + parameter_size)

//----- Predefine Extended Types -----
#define LMD_MOVE          12 // + float(=x) + float(=y)
#define LMD_DRAW           13 // + float(=x) + float(=y)
#define LMD_CIRCLE          14 // + float(=x) + float(=y) + float
                           (=r; r>0 CCW)
                           // not used for V3.0 but maybe in
                           future
#define LMD_DRILL          15 // + float(=x) + float(=y)
#define LMD_TOOL_SELECT    16 // + int(=iTool)
#define LMD_ARC            17 // + float(cx)+float(cy)+float(ex)
                           +float(ey)+float(r)
                           // draw arc 0..+-355 degree.

// If start and end points are the same,
// draw a 360 degree circle
// ***** Scanner head extended types *****
// scanner head has its window (box), defined by points
// (-xmax,-ymax) and (xmax,ymax).
// Origin of the head is at (0,0)
// Position of the head is the point at which the origin (0,0)
is placed
#define LMD_HEAD_BOX       18 // + float(=xmax) + float(=ymax)
#define LMD_HEAD_POS        19 // + float(=x) + float(=y)
#define LMD_HEAD_NONE       20 // head position undefined, do not
use head
// for subsequent data
/** 2001-03-10
The scanner box sizes in LMD are now defined as full size; the
real scanner
window min and max corners are defined as follows :

lower left point (xpos - xbox/2, ypos-ybox/2)
upper right point (xpos+xbox/2, ypos+ybox/2)

where
xpos,ypos  is defined by LMD_SCANNER_POS
xbox,ybox  is defined by LMD_SCANNER_BOX
*/
// extension for 3D movements (2D+Height = 2.5D      2002-09-19
#define LMD_MOVE3D         21 // + float(=x) + float(=y) + float
                           (=z)
```

```
#define LMD_DRAW3D      22 // + float(=x) + float(=y) + float  
(=z)  
#define LMD_CIRCLE3D    23 // + float(=x) + float(=y) + float  
(=z) + float(=r; r>0 CCW)  
#define LMD_ARC3D       24 // + float(cx)+float(cy)+float(z)  
+float(ex)+float(ey)+ float(ez)+float(r)  
                                // draw arc 0..+-355 degree.  
#define LMD_DRILL3D     25 // + float(=x) + float(=y) + float  
(=z)  
#define LMD_MACRO_DEF   26 // + int(=macro-number-def, start-  
ing with 1, 2 ...n )  
                                // + path + LMD_MACRO_END  
#define LMD_MACRO_END   27  
#define LMD_MACRO_CALL  28 // + int(=macro-number-ref, start-  
ing with 1, 2 ...n )  
                                // + float(=x) + float(=y), off-  
set values  
  
/**  
----- Block Types -----  
- All commands from 128 to 255 are reserved for block types  
- A block is terminated by LMD_BLOCK_END  
- Blocks can be nested and have variable length  
- Elements in the block are identified by the position in  
the block. Order of elements in the block can not be  
changed. New elements can be added just before end of  
the block. Elements, which are not used any more, must  
still  
be present.  
*/  
// ----- Predefined Block Types -----  
#define LMD_TOOL      128  
// + LMD_INT iTool for use in LMD_TOOL_SELECT  
// + LMD_BYTE for mode: 0:draw, 1=flash 2=flash+draw  
// + LMD_FLOAT: width  
// + LMD_SSTRING: tool-name  
#define LMD_PHASE_HEADER 129  
// + LMD_SSTRING: phase-name  
// + LMD_SSTRING: layer-name  
#define LMD_PHASE      130  
// + LMD_PHASE_HEADER()  
// + LMD_TOOL { LMD_TOOL }  
// + { LMD_TOOL_SELECT() { LMD_DRILL() | path }  
// path ::= LMD_MOVE() { [LMD_CIRCLE()] LMD_DRAW() };
```

```
//Please note that this structure may change in future ver-
sions.

/***
----- File-Structure -----
file ::= header body;
header ::= "LMD_Mill_Drill Version 3." subversion "\n" {
string "\n"}\032";
subversion ::= "0"; // for the here predefined extended and
block-types
string ::= sequence of ASCII >= " "; user defined strings.

E.g.:
"Generated by: CircuitCAM V3.0 (36)"
"Source-file: c:\...\Tutor.cam"
"OutputJob: MySpecialJob"
"User: Muster Meyer"
"Organization: My Company"
"SerialNumber: 979999:99"
"Timestamp: 1997 7 18 12 0 0"

Bmaster may display these strings somehow (Property
DlgBox?)

body ::= { LMD_TYPE_DEF() } { basic_type | block };
basic_type ::= 1..9
block ::= (128..255) { basic_type | block } LMD_BLOCK_END;
----- Recommendations -----
The order in which things to be written to the file:
0) header
1) all extended type definitions (including LMD_MOVE,
   LMD_DRAW, LMD_CIRCLE, LMD_DRILL and LMD_TOOL_SELECT
   according to the subversion "0").
2) all tool definitions (it is also possible that the tool
   is defined somewhere in the file - but before it is
   referenced).
   2.1) the scanner head box definition LMD_HEAD_BOX (if used
        later).
3) The LMD_PHASE
   3.1) The LMD_PHASE_HEADER
   3.2) Sequence of LMD_TOOL_SELECT, LMD_HEAD_POS,           LMD_HEAD_NONE and
drill, move, draw, circle

How BMASTER will interprets the blocks:
a) After the header and some LMD_TYPE_DEF there are on the
primary
   level only LMD_TOOL and LMD_PHASE.
   All other types (block, extended or basic) are skipped.
```

b) Inside the LMD_PHASE BMaster will find the phase-attributes in

LMD_PHASE_HEADER followed by the geometry.

c) Always identify the Type:

If it is an unknown block-type, skip elements until block end.

If it is an unknown extended-type, skip the number of bytes defined by LMD_TYPE_DEF.

If it is a basic-type which role is unclear at this position skip it.

Interpretation of scanner head data:

The LMD_HEAD_BOX must be defined before first LMD_HEAD_POS appears.

By default at the beginning the LMD_HEAD_NONE is active.
After the

LMD_HEAD_POS appears the drill, draw, arc and circle data are interpreted

as follows:

- if they fit into scanner head box, they are done on scanner head

- if they not fit into box they are skipped

For data that appears after the LMD_HEAD_NONE the scanner head is not used.

They are performed by the usual head.

<End of LMD file description>-----
*/

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

ODB++

ODB++ is a data format for PCB artwork from Valor Computerized Systems Ltd. The electronic documentation is available at <http://www.valor.com/>.

The actual supported version is 7.1.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

Sieb & Meyer

We refer below to the documentation:

S&M commands

Format 5000

Issue: May 1991

from:

Sieb & Meyer Elektronik GmbH
Auf dem Schmaarkamp 21
21339 Lüneburg, Germany.

Milling and drilling machines from *Sieb&Meyer* work in the *S&M* internal format or in the *Excelon* 1 or 2 formats. We look below at the specific features of *S&M* format.

The length details in *Sieb&Meyer* data (X, Y, R) are always given with decimal point in inches or millimetres, thus e.g.:

X40. Y50. G2 R15. F1.2

If the decimal point should however be missing at input, then - as with *Gerber* data - there is a switch to M.N format with leading or trailing zeros. For this reason, the *Gerber/Excelon dialogue box* is also used for the format specification.

At input and output, no milling radius compensation is supported.

Reading Production Data:

CircuitCAM processes the following commands at input:

F...	Feed speed, is ignored.
G1	Straight milling.
G2, G3	Arc milling.
G90, G91	Coordinate absolute, relative
M48....%	Commentary (not original Sieb & Meyer))
M...	Other M commands are ignored.
R...	Radius for arc.
T...	Tool selection.
X..., Y...	Coordinates
(...)	Commentary
%	Jump to line end.
Line end	End of a command block

Writing Production Data:

The following commands are used at Sieb & Meyer file output:

Tool selection	T...
Path start	X...Y...T...T0
Line segment	X...Y...G1
	X...Y...G2R...
Arc segment	X...Y...G3R...
Path end:	X...Y...T0
Tool flash:	X...Y...

No milling speed (F...) is given since *CircuitCAM* does not have any information ready about this

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.1. Tool Paths

4.1.1. Laser Scanner

The scanners are generated according to the settings in the **Laser Scanner** dialogue window (to open this dialogue select the **Laser Scanner** line from the **Tool path** menu):

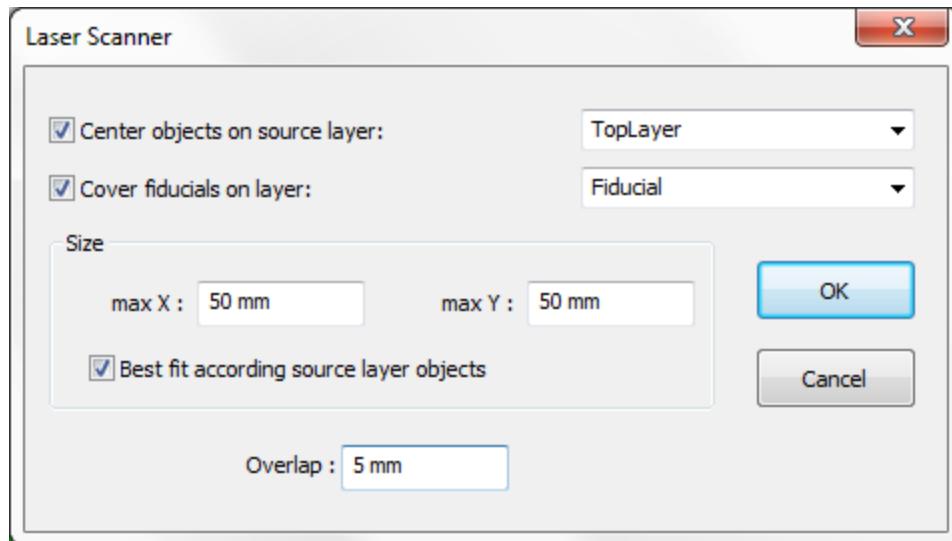


Figure 324. Laser Scanner dialogue window

If the flag **Centre on source layer** is set the net of scanners is Centred on layer's objects.

The scanners' generator creates one configItem of type Rectangle, where the **maxX** and **maxY** fields define the configItem size. The item has the fixed **RectangularScanner** name in the **LaserScanner** configuration. Then the generator places a set of flashes of this type on the fixed **LaserScanner** layer. The scanners are generated in according to horizontal serpentine law: in a line with odd number the scanners go from left to right, in an even line – vice versa. If **Overlap** > 0, neighbouring scanners have overlap (common area).

If the flag **Cover fiducials on layer** is set *On*, the fiducials, which are present on the defined layer (selectable from the combo box), are covered by the **Rectangular Scanner** configItem. The flag **Best fit according source layer objects** can be set *On* or **maxX** and **maxY** size parameters should be applied for the configItem size.

Of course, the user can modify the set of scanners manually – delete, move, and create some new scanners and so on.

Laser Scanner is used in LMD and HP-GL export, when special export flag **Scanner support** in the export task is set (see [Export Task](#)).



Note: Mentioned actions can destroy the generated arrangement of scanners (serpentine). But again, the user has possibility easy to restore/change the arrangement of scanners with the **Swap** function.

4.1.2. Tool Compensation

Tool Compensation is typically needed for contour routing of a board outline or cutout. Typically the higher level [Contour routing](#) functionality is used for board outlines, but for specific usages the underlying tool compensation functionality is available as well.

To open the **Tool Compensation** dialogue window, select in the main menu **Tool Path -> Tool Compensation**.

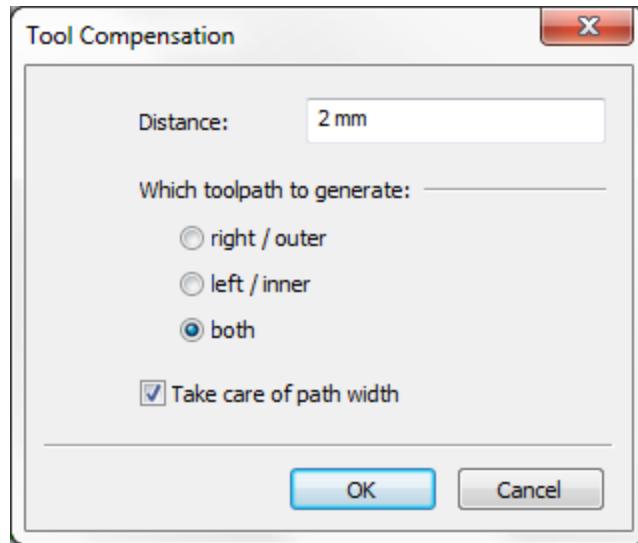


Figure 325. Tool Compensation dialogue window

- **Distance:** Specifies an offset of the toolpath from the selected path or outline.
- **Which toolpath to generate:** Specifies on which side of the selected path the toolpath should be generated. Available selections:
 - **right / outer** - Creates a toolpath on the right side (for open paths) / outer side (for closed paths).
 - **left / inner** - Creates a toolpath on the left side (for open paths) / inner side (for close paths);
 - **both** - Generates a toolpath for both sides of the selected path.
- **Take care of path width:** If *On*, the offset is calculated from the centre-line of the selected path. If *Off*, the offset is calculated from the edge of the selected path.
- **OK:** Saves the values in registry and generates the toolpath.



Note: The results of the tool compensation operation is located on a new layer, which name is a combined string of a layer name (the selected path is located on) and a letter *L* (left/inner) / *R* (right/outer).

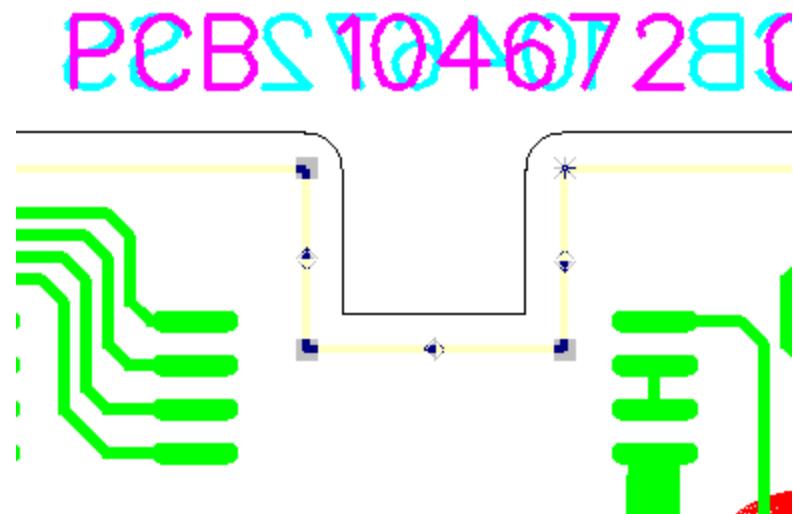


Figure 326. Selected outline (light yellow), tool compensation result (on outer side (black line), Take care of path width *On*)

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.1.3. Contour Routing

Contour routing of a PCB is a process to cut out the final PCB from the surrounding material. This is done by milling along the outline of the PCB, taking into account the width of the milling tool. To avoid uncontrolled movements of the PCB, some small breakout tabs should remain, so that the final PCB could later be broken out from the remaining material by hand. The **Contour Routing** dialogue has various ways to control the number, size and location of these breakout tabs. The result of the Contour Routing is tool paths for the selected milling tool. As an additional functionality the **Contour Routing** dialogue can also be used for milling of cutOuts in a PCB.

The **Contour Routing** dialogue is started from the **Tool path -> Contour Routing** menu. It operates on the currently active layout, typically the main layout.

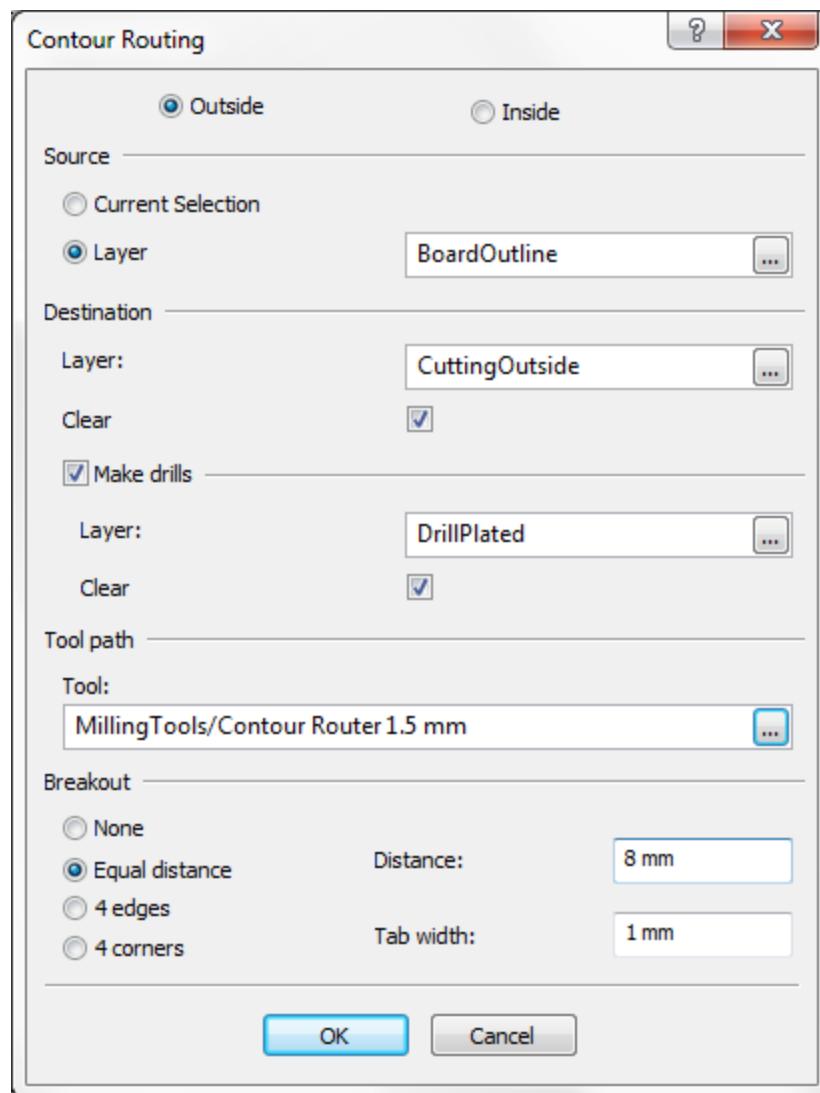


Figure 327. Contour Routing dialogue window

- **Outside/Inside:** Specifies the milling area. In case of a closed path (or when the start-point and the end-point of the open path are contiguous) an inner area for the milling contour calculation has sense only.
- **Source:** An object for the milling contour routing can be selected before the dialogue window is called (**Current Selection**). In case of **Layer**, a milling contour will be calculated for all objects of the selected layer. To select a layer, hit the ... button next to the text field and select the layer in the **Layers** dialogue window.

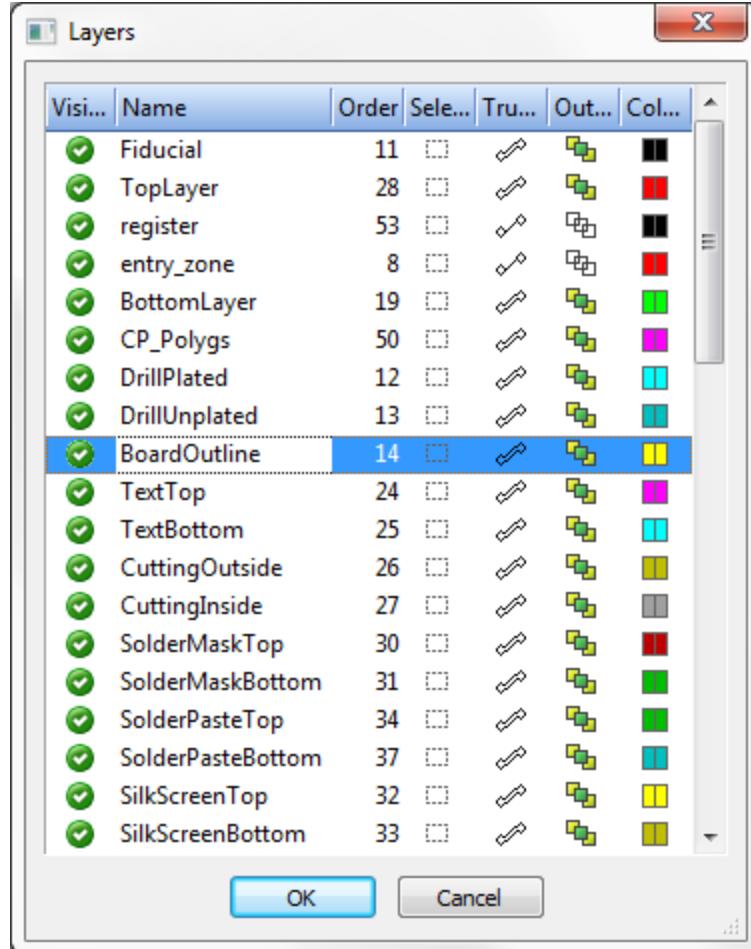


Figure 328. Layer dialogue window

- **Destination Layer:** Defines the resulting layer. If the **Clear** option is *On*, all components on the selected destination layer are removed.
 - **Make drill:** If *On*, inserts a hole at the beginning of each cutting line (see blue circles in the picture below). The hole has the same diameter as the cutting line. These holes are located on the specified layer, which is defined in the **Layer** field. To select a layer, hit the ... button next to the text field and select the layer in the **Layers** dialogue window.
- If the **Clear** option is *On*, all components on the selected destination layer are removed.
- **Tool path:** Specifies a type and a width of a tool. New values can be selected from the **Tools** dialogue window, which appears after the button ... is hit.

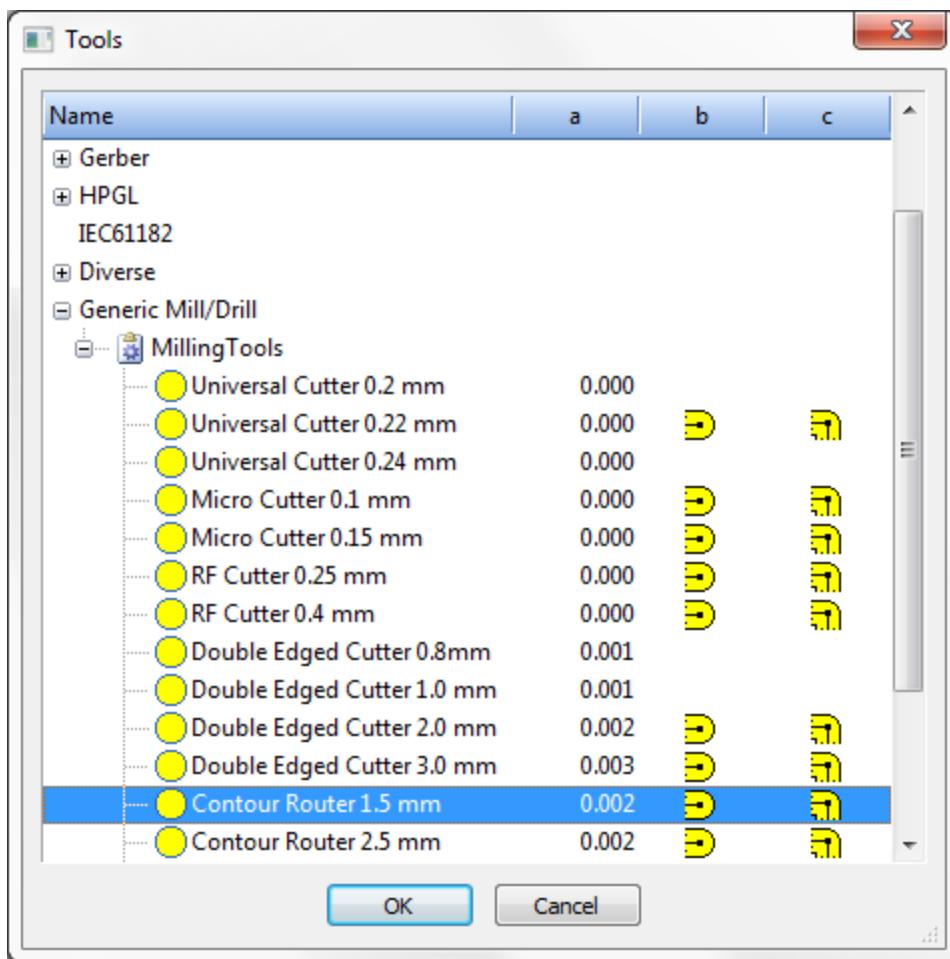


Figure 329. Tools dialogue window

- **Breakout options:** The group generates gaps around all contours. There are four modes to do this:

- **None:** No gaps in contour paths.
- **Equal distance:** Gap width and distance between two gaps are set in fields **Tab width** and **Distance**.
- **4 edges:** Only four gaps with **Tab width** size are generated around each contour - on left, bottom, right and top sides. The gaps coordinates are calculated as the middle points of corresponding edges of surrounding rectangle of the contour.
- **4 corners:** Only four gaps with **Tab width** size are generated around each contour - on left-top, left-bottom, right-bottom and right-top corners. The gaps coordinates are calculated according to intersection points of diagonals of the surrounding rectangle with the contour.

The results of *breakout variations* are shown below (*Dark Yellow outline*).

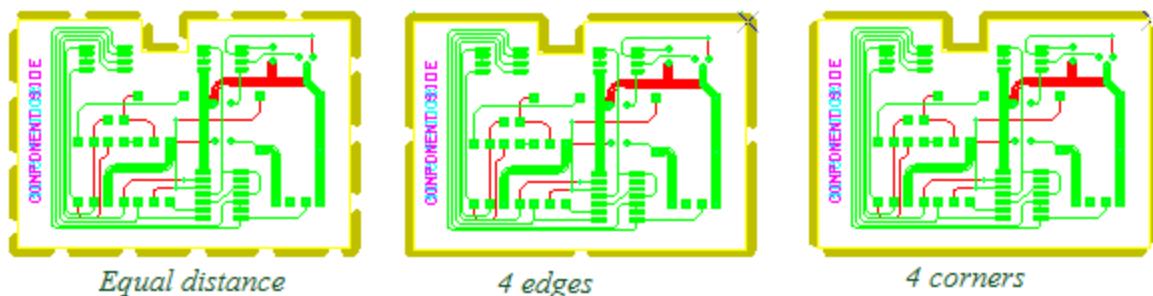


Figure 330. Contour routing result

Sequence

This operation analyses the sequence of objects. It changes the selection of objects, works on objects inside the same layer and between the objects of the same type (e.g., between paths). A type and a layer define the selected object. If there are no initial selections, the *Select next* and *Select previous* operations are disabled. If a layer is not set, all operations are disabled. The order of objects can be changed with the **Swap** operations.

Properties of the selected object (type, current position) are shown in the *Status Bar*.

This function has four operations:

1. Select the next object in the group **Ctrl + Shift + Right**;
2. Select the previous object in the group **Ctrl + Shift + Left**;
3. Select the first object in the group **Ctrl + Shift + Up**;
4. Select the last object in the group **Ctrl + Shift + Down**.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.1.4. Insulate Milling

CircuitCAM provides functionality to generate insulation paths with up to five different tools: Primary, Big, Bigger, Small and Smaller.

The Primary tool is mandatory because it is used in the primary insulation channel around all copper areas, such as tracks and pads on the specified layer. Additional insulation channels can be generated with either the primary, big or bigger tool, dependent on the availability. When primary and bigger tools are used for insulation with multiple channels, the first insulation channel will be for the primary tool, the second insulation channel will be for the big tool, and all other concentric insulation channels will be for the biggest tool if available. *CircuitCAM* tries to maximize the usage of the big and bigger tools because of their enhanced life time.

The small and smaller tool are used for narrow places between the primary insulation channel and the copper areas, that are too narrow for the primary tool. Because of limited life time of the small tools, their usage is minimized.

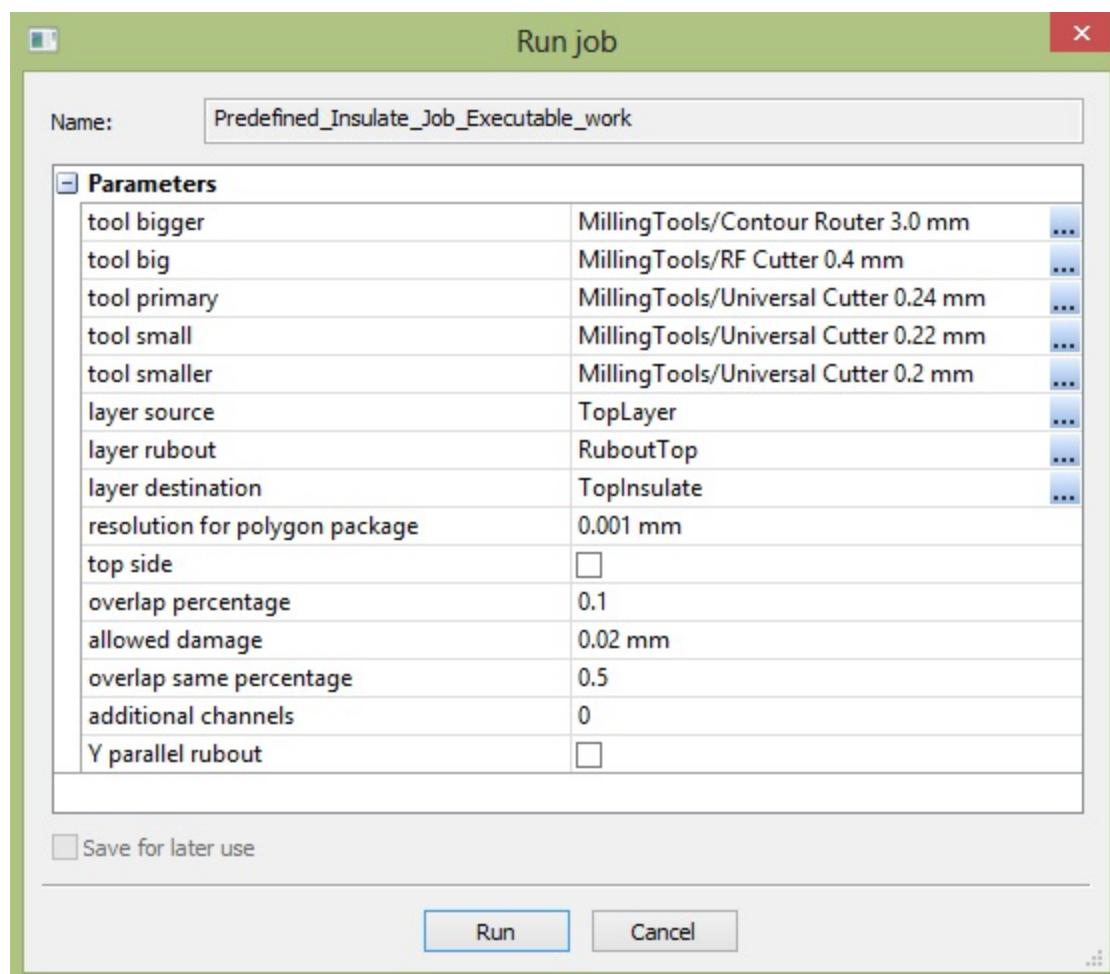
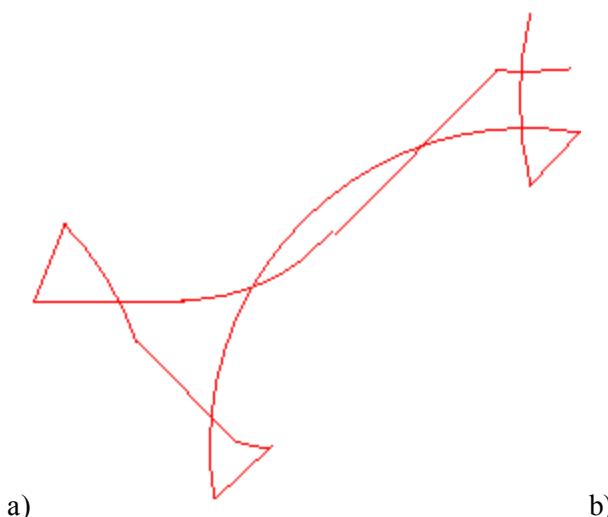


Figure 331. Predefined insulate job

Besides the primary channels generation, the removal of smaller or larger areas is supported. The user has a possibility to define a rubout area (see [Create rubout area](#)).

Old concentric rubout channels with triangles (see below) were updated to use needles and this caused the reduction of the vectors number in concentric paths, and improvement of the insulation procedure.



a)

b)

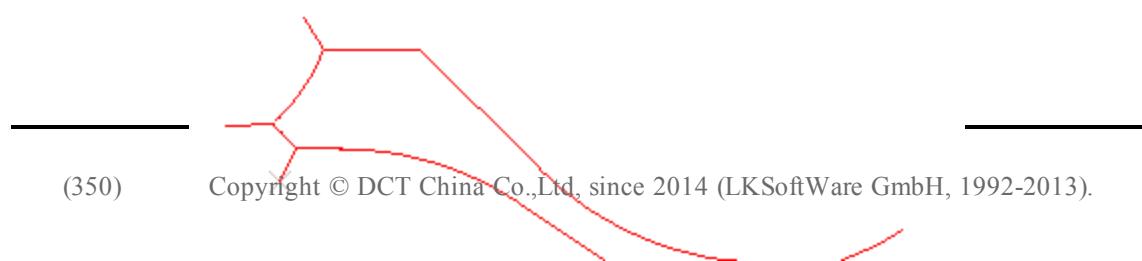


Figure 332. a) Old concentric paths with triangles; b) new concentric paths with needles (right side)

The set of predefined insulate Scripts are provided with the *CircuitCAM* installation. Predefined insulate jobs can be accessed and executed via the [Jobs view](#). These jobs are located beneath the **Batch(scripts)** node and have a category **Customized**. The **Customized** category name is used not only for insulate jobs.

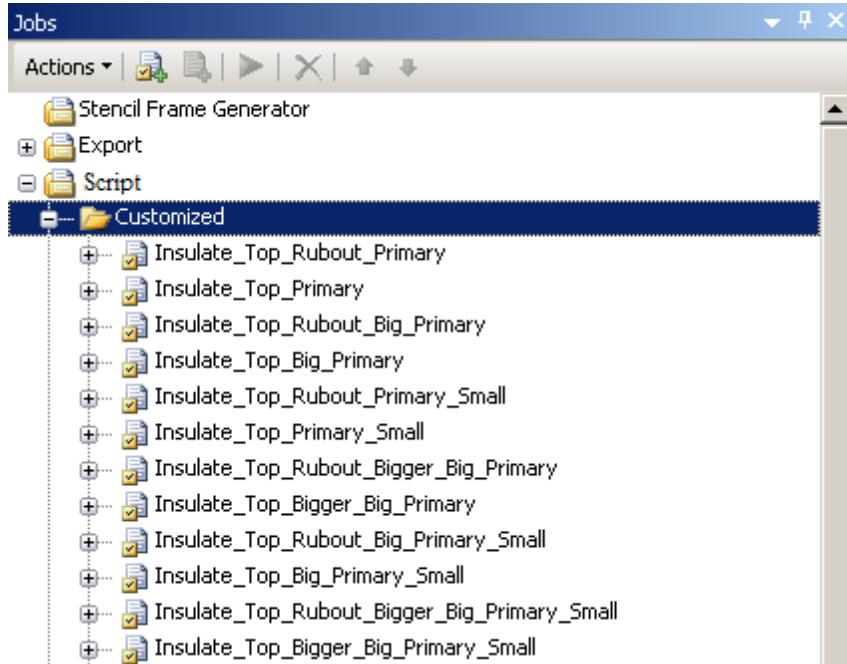


Figure 333. List of Scripts

When running one of the insulate scripts, the area to work on is defined by the source layer is subtracted from the rubout area. In the case of no remaining area Warring message will be shown.

Insulation Examples

Insulation tracks using 4 tools with defined rubout area.

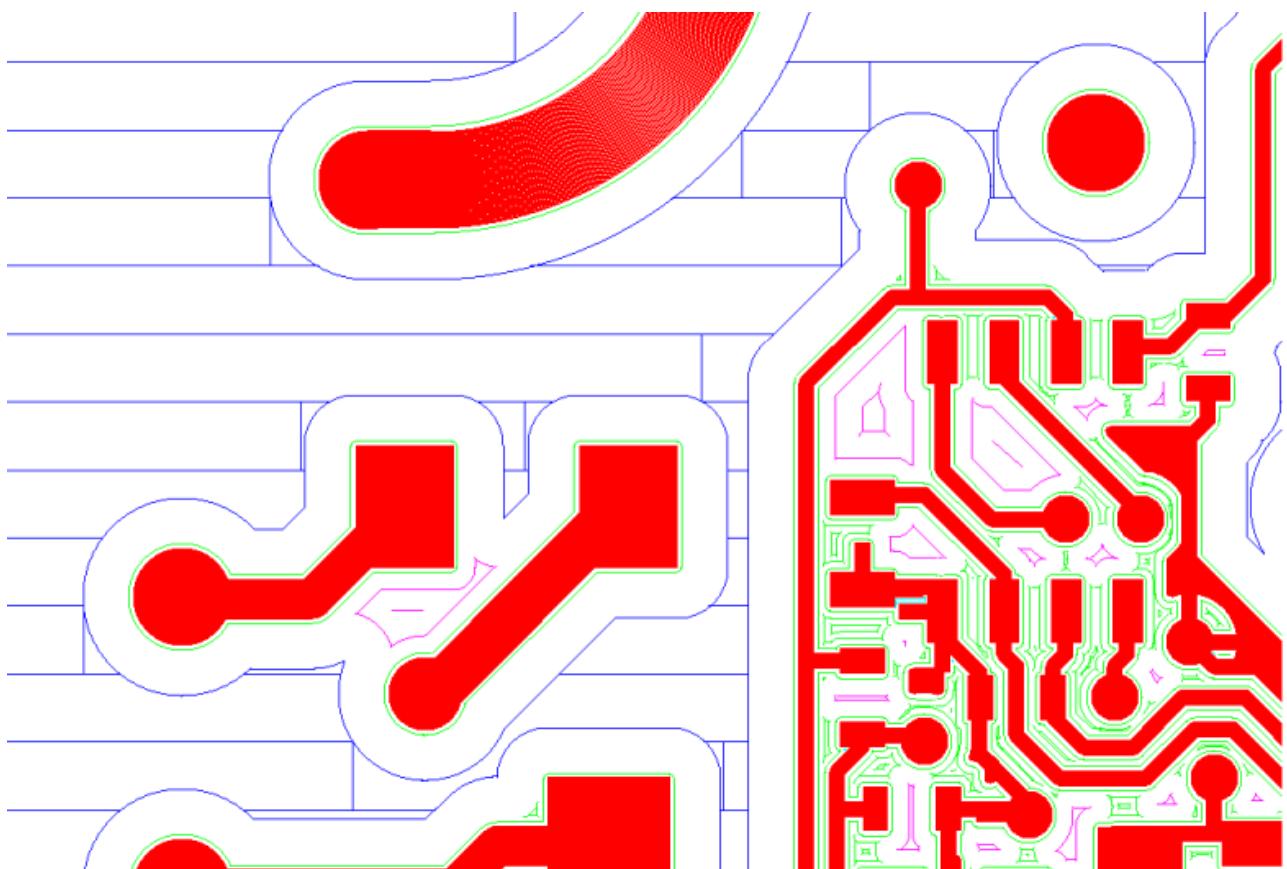


Figure 334. Insulation tracks (4 tools, defined rubout area)

Insulation tracks using 3 tools with defined rubout area.

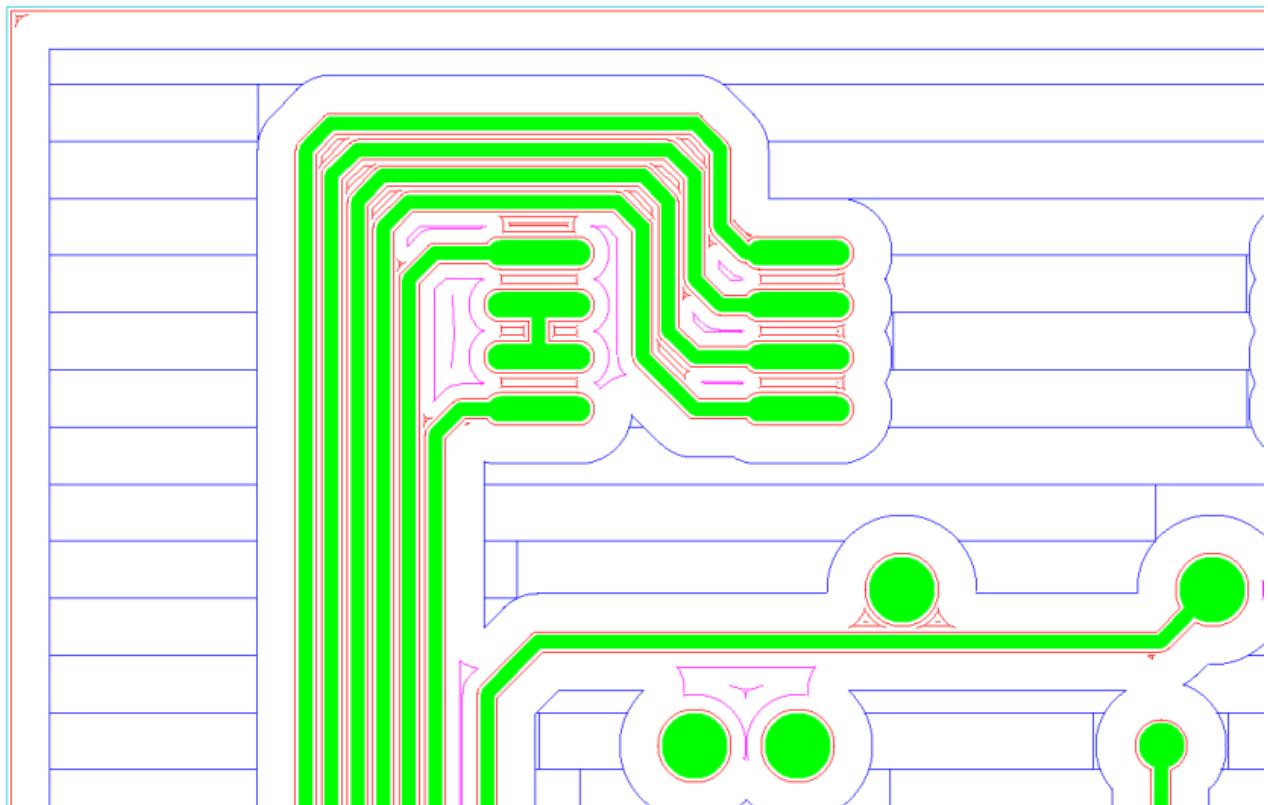


Figure 335. Insulation tracks (3 tools, defined rubout area)

Insulation tracks using 3 tools with a defined rubout area and with additional channels around the primary channel.

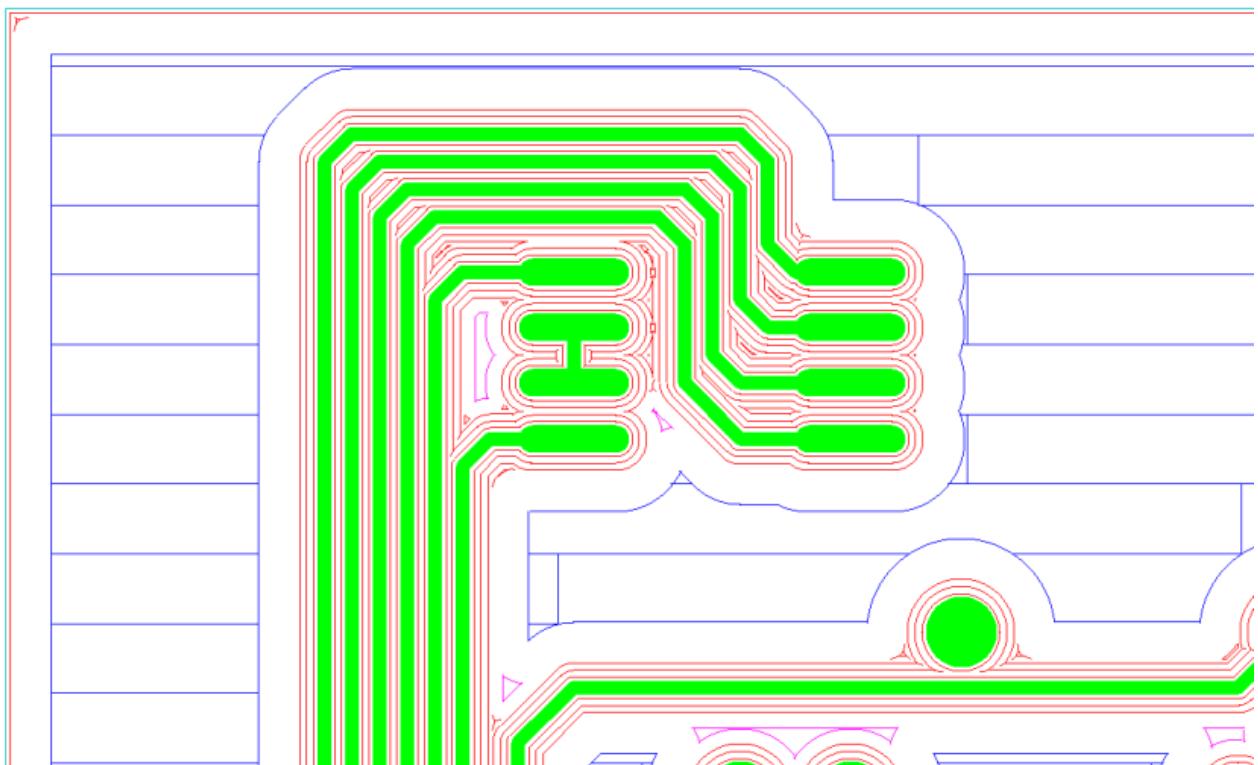


Figure 336. Insulation tracks (3 tools, defined rubout area, additional channels around primary channel)

Insulation tracks using 2 tools with a defined rubout area.

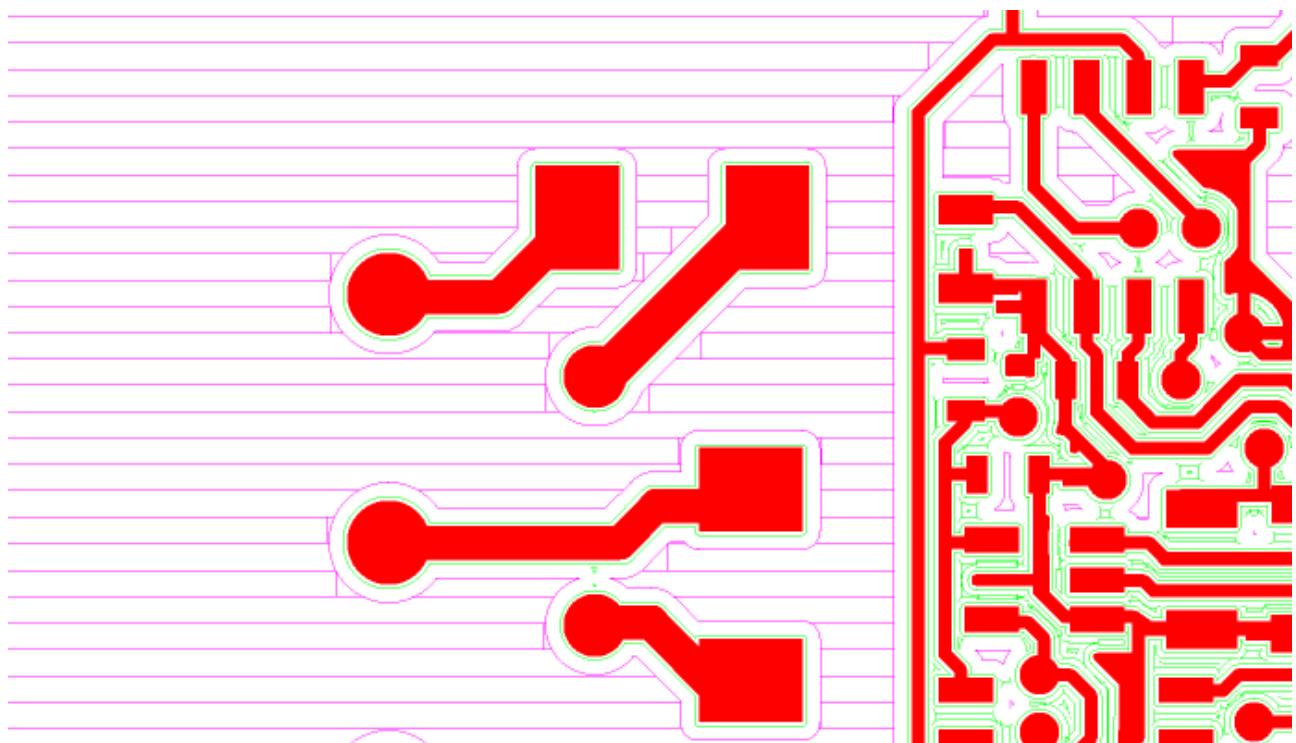


Figure 337. Insulation tracks (2 tools, defined rubout area)

Insulation tracks using 2 tools with a defined rubout area and with additional channels around the primary channel.

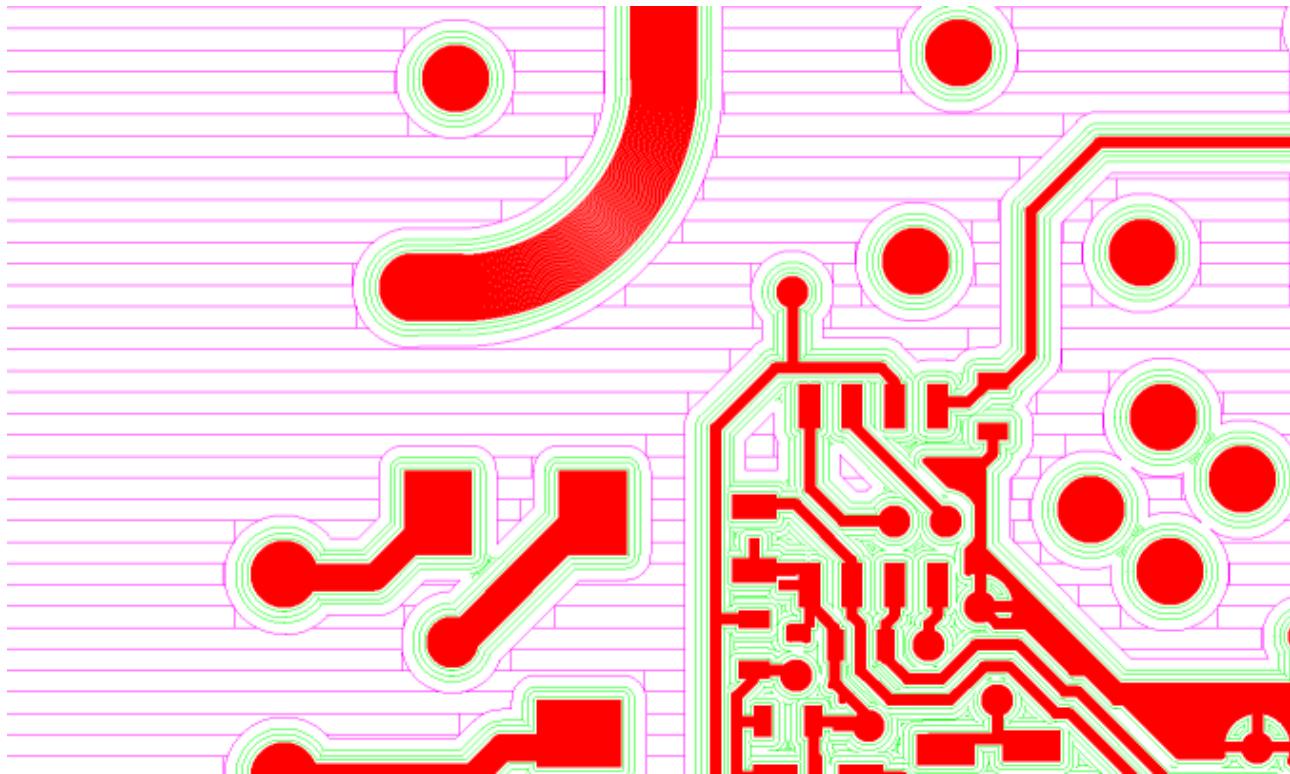


Figure 338. Insulation tracks (2 tools, defined rubout area, additional channels around primary channel)

Insulation tracks using 1 tool with additional channels around the primary channel.

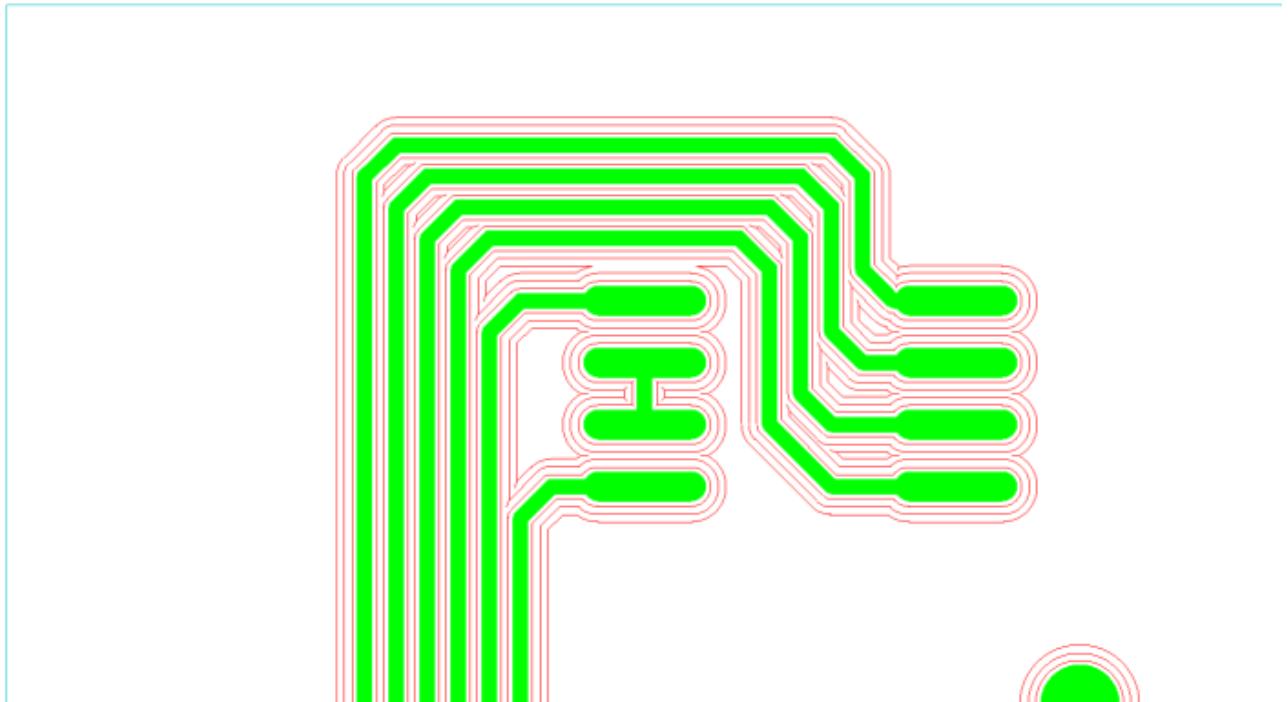


Figure 339. Insulation tracks (1 tool, additional channels around primary channel)

4.1.5. Striping & Stripping

Striping and Stripping is a technology to remove copper or equivalent material that is laminated on a base substrate, e.g. a PCB.

For this process a Laser NC machine must be able to operate in two modes; cutting and heating.

In a first phase, the copper layer is cut into stripes using the laser; in a second phase, the stripes are heated by the laser so they can be stripped away by an additional airflow.

Striping and Stripping is only available for the **Laser Pro variant** of *CircuitCAM* that comes with additional documentation and a "***default_striping_and_stripping.cat***" file that you will have to copy into the template directories.

To learn how to copy template files, please refer to [Synchronize](#).



Note: For additional and detailed information, please refer to the separate "***Striping and Stripping Documentation***" or contact **DCT**.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.1.6. NC Mode

The purpose of the NC mode is to directly control a connected NC machine. The NC mode allows to control an NC machine manually, or via automated NC data such as; *tool paths and drill positions*. The automatic execution of data is applied layer by layer, depending on which side of the PCB is targeted, the data is either used as it is, or is mirrored.

Before starting the NC mode, we usually need to establish a connection with an NC machine. Before any data can be executed, the connected NC machine, has to be configured *i.e.:* "*defining absolute positions for x, y, and z, figuring the orientation of the base material, setting up tools and more*".

NC Mode is accessible via the **NC/Tool Path** menu;

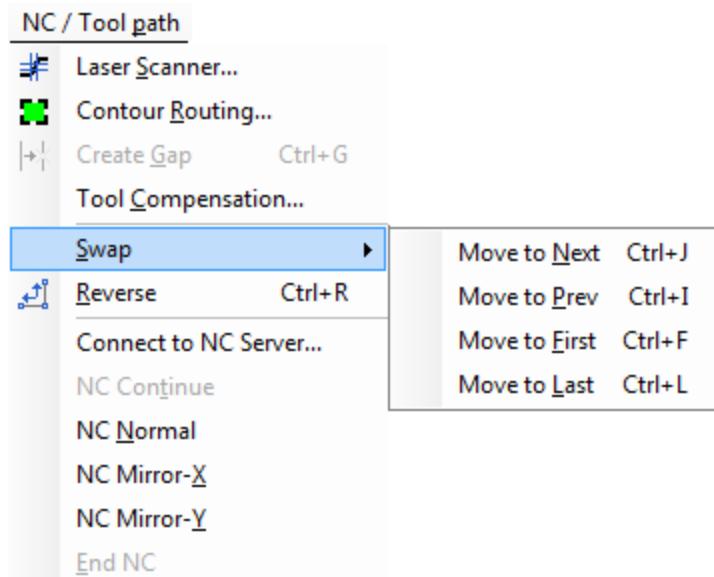


Figure 340. NC/Tool Path menu

To start working on NC Mode, the following steps are applied:

Connecting to the NC Server

Before starting the NC mode, *CircuitCAM* needs to establish a connection with a so called I++NC server, this server might be connected; locally to the same computer terminal, or remotely on another computer terminal, or it may even exist directly built-in an NC machine. When this connection is established, *CircuitCAM* receives data about the type of the NC machine, its working area, and other miscellaneous information.

To connect to an NC server, navigate to the **NC/Tool Path** menu, select **Connect to NC Server**. The connection window appears; in this window the address of the machine is entered in the following format; "**IP address: port number**";

if connecting to a remote server, an **IP address number** and **Port number** are entered,

if connecting to a machine attached to the same computer terminal, **Default localhost** is selected.

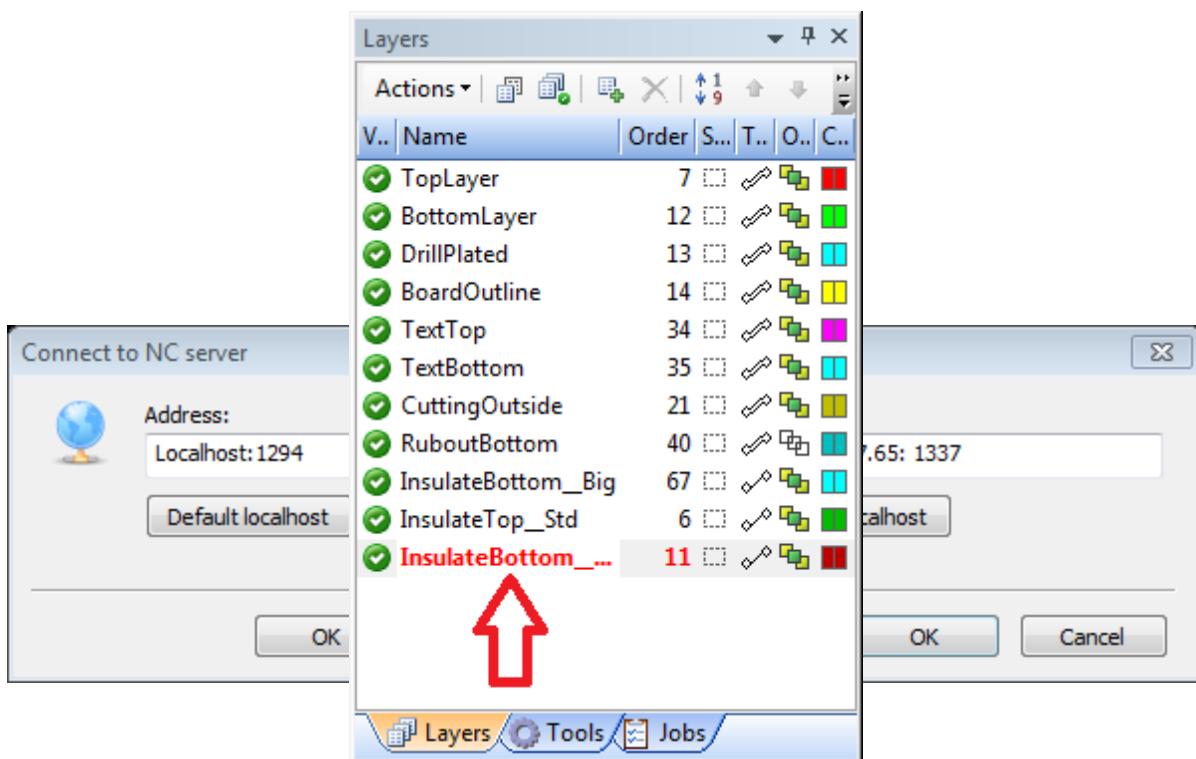
Figure 341. Connecting to an NC sever; local machine (left), or remote machine (right)

Note: When connecting to a remote NC server, network or internet access must be available, depending on the location of the server.

Selecting a layer to process

While in NC Mode, *CircuitCAM* can only process one layer at a time

The layer to process is selected, and is set as the active layer. See [Setting Active Layer](#).

**Figure 342. Setting the active layer for NC mode**

Switching to NC Mode

To activate the NC layout, one of the following modes must be selected from the **NC/Tool Path** menu:

- NC Normal: Opens the NC layout with normal orientation.
- NC Mirror-X: Opens the NC layout with all objects mirrored on the X-axis.
- NC Mirror-Y: Opens the NC layout with all objects mirrored on the Y-axis.

NC Mode Layout

The NC mode has its special layout; this special layout shows the graphical data of the layer selected to be processed.

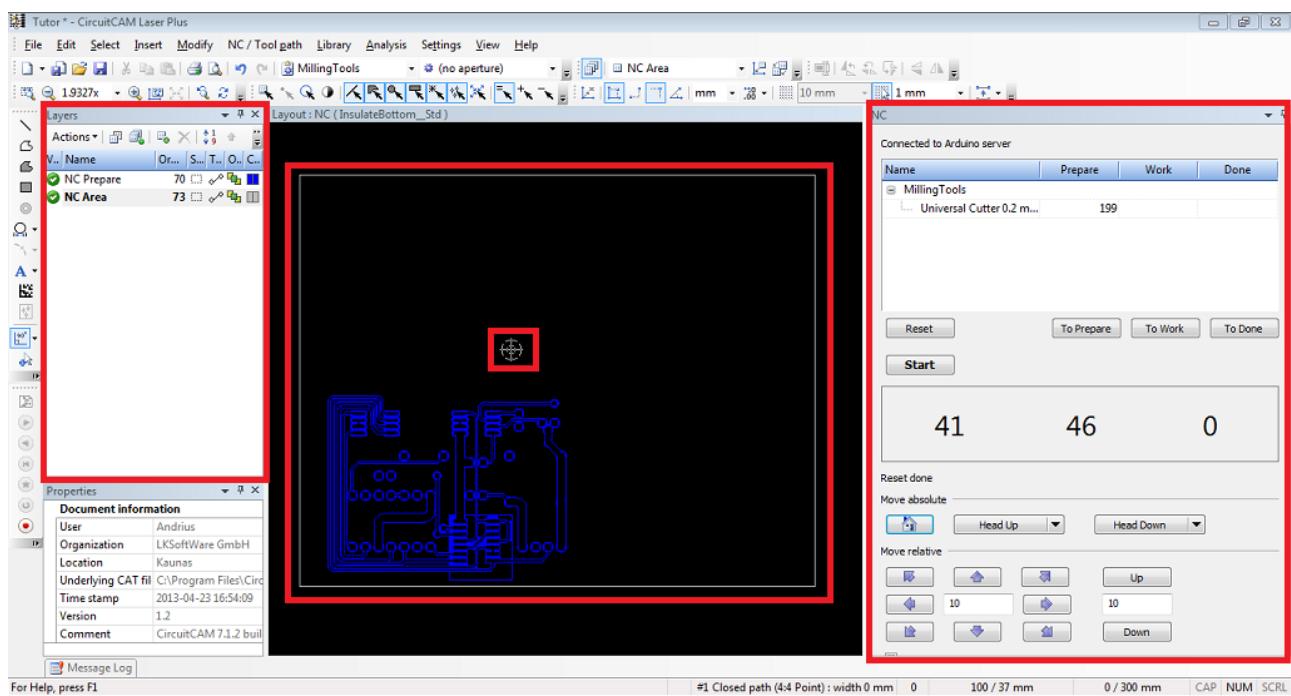


Figure 343. NC Mode layout

While the NC mode layout is active, and while the NC server is connected, four differences are noticed as in the figure above, listed from left to right;

- NC Special Layers.
- NC Machine Area.
- Machine head position.
- NC Controls.

NC Layers

The NC Mode has its special layers; these layers contain the graphical data of various stages;

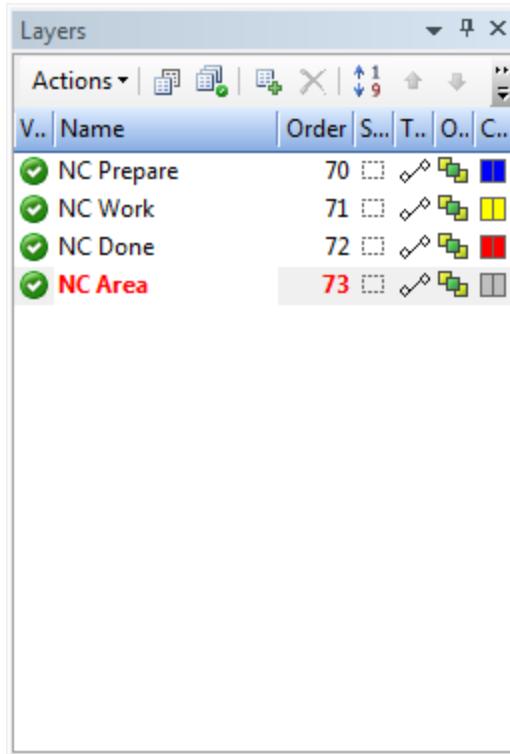


Figure 344. NC Mode layers

- NC Prepare: This layer represents the first phase while working with NC Mode; this layer contains the items which still need more editing and preparation.
- NC Work: This layer represents the second phase in NC Mode; items are moved to this layer when they are ready to be processed, this layer acts like a *ready flag* before the NC machine starts.
- NC Done: The third and final phase in NC Mode; items are displayed here after they are fully processed by the machine.
- NC Area: This layer only displays an outline, this outline represents the working area of the machine.

Machine Area/Head Position

An NC machine needs to be configured prior to connection; one of the configurations is to determine the coordinates of the working area, *i.e.*: $xMin$, $xMax$, $yMin$, $yMax$, $zMin$, and $zMax$.

After connection, the size of the working area is automatically read from the machine. The graphical data displayed in the layer NC Area, represent the boundaries of the machine working plane (*a rectangle in the example figure*). Since *CircuitCAM* works only in a 2D mode, the boundaries of this rectangle are calculated using the $xMin$, $xMax$, $yMin$, and $yMax$. The values of $zMin$ and $zMax$ are variables of the NC machine, used to determine the depth of the drilling and milling.

The current **CAM** document is then displayed in the NC mode, the absolute coordinates of the document are applied to the working area of the machine, *i.e.*: the origin point of the document $(0, 0)$ is matched to the origin point of the NC machine.



This special marker indicates the position of the machine head. The initial position "**home**" of the head is read automatically from the machine. The place of this cursor changes place accordingly, as the machine is in process

NC Controls

The **NC Control View** only appears while in NC Mode, this window displays various controls for the NC Mode, each explained in details below.

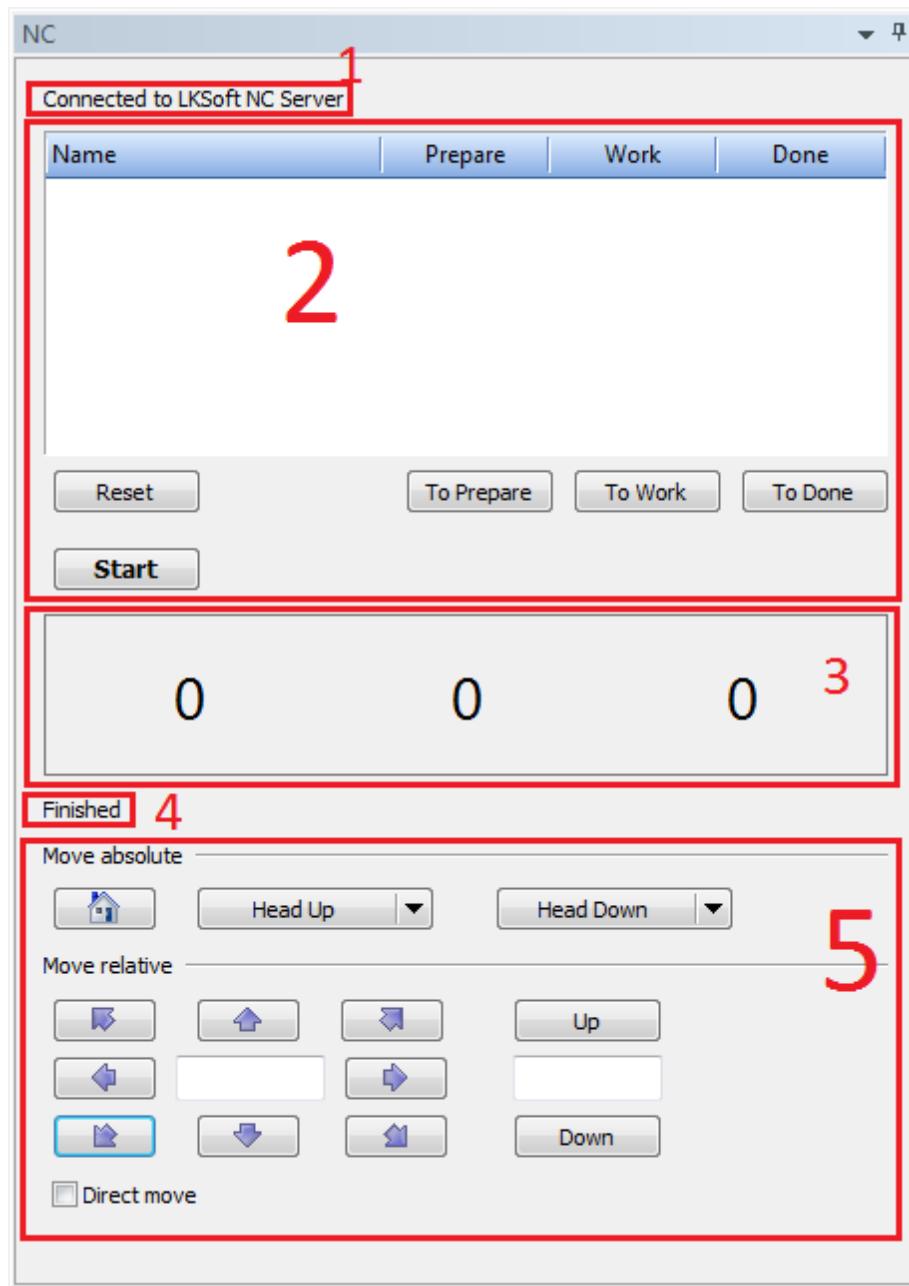


Figure 345. NC Control View

1. Connection Status:

This section indicates whether an I++NC server is currently connected or not.

2. Tools and Progress:

This table displays the different tools that are present in the selected layer. Beneath the table, the layer and order controls are displayed.

3. Head Position Coordinates:

This section displays the current coordinates of the machine head.

4. Machine Status:

This section displays the machine status whether; *running*, *stopped*, or *finished*,

5. Machine Head Controls:

The movement group is divided into two parts; *Move Absolute* and *Move Relative*.

Connection Status

After connecting to the I++NC server, and after switching to the NC Mode, this section displays the current connection status.

When a connection is successfully established, *CircuitCAM* displays "*Connected to name_of_the_server*".

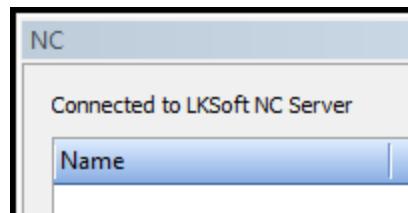


Figure 346. The connections status while connected to an I++NC server

If for any reason (*power failure, network/internet error*) the connection with the I++NC server is interrupted, the following error message appears, and the connection status is then changed to "*Not connected*".

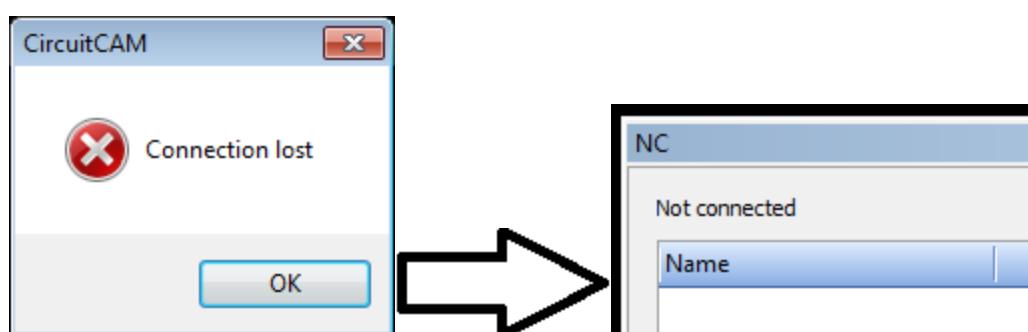


Figure 347. NC Connection error

 **Note:** Even it is recommended to first establish a connection and then switch to NC Mode, however, it is possible to open NC Mode without connection, in such case the status is displayed as "*Not connected*".

Tools and Progress

The "**Tools and Progress Group**" displays the various tools present in the layer, the number of tool paths present under each tool type, it also displays the progress control buttons. By standard, NC mode starts with all the tool paths in the prepare layer, as seen in the figure below.

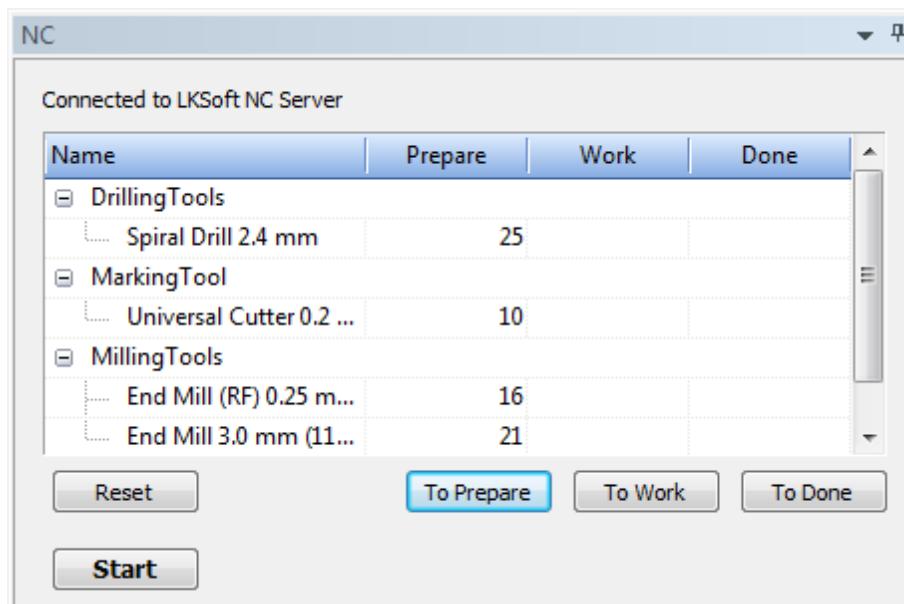


Figure 348. NC control standard view, displaying all tool paths in the "Prepare" layer

Listed under the **Name** column, tools are grouped into separate nodes, these nodes represent the tool category; (milling, drilling,etc). Under each category, the shape and the size of the tool is displayed.

Next to the name column are the three progress layers; **Prepare**, **Work**, and **Done**, the number in each field indicates the number of tool paths associated with each layer, *e.g.: In the image above, the **Drilling Tool**; **Spiral Drill 2.4 mm** is used in 25 tool paths, those 25 tool paths are all in the prepare layer.*

After reviewing the prepare layer, and after all the objects/tool paths do not need anymore editing, comes the time for the second phase; **Work Layer**.

The work layer enables ordering and organization of tool paths; *i.e.:* the tool paths are processed in the same order they are added to the work layer, regardless of the tool used. The work layer is considered as a *ready flag* before sending the data to the NC server.

To move one or more tool paths to the work layer, they are selected from the graphic layout, then a left click is made on the "**To Work**" button.

To select all of the tool paths created by the same tool, just double click on the number displayed in any of the three columns, *e.g.: In the figure above, if a double click is applied on the number "10" next to the **Marking Tool**; **Universal Cutter 0.2 mm**, all of the 10 tool paths, which are present in the prepare layer will be selected.*

The final stage is to transfer the data to the NC machine and start processing it, to do this, a left click is applied on the **Start** button.

Clicking the **Start** button will transfer the tool paths from the **NC Work** layer, to the NC server (*machine*), after each tool path is processed, it is then moved to the layer **NC Done**. While the machine is running, the head position cursor moves in accordance, identifying in which place the machine is operating.

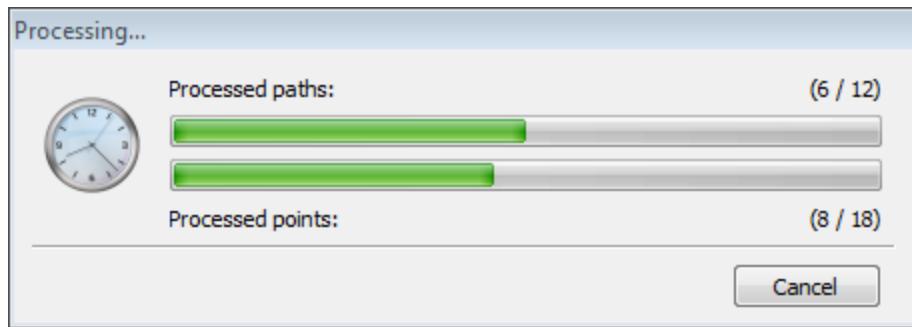


Figure 349. NC Machine progress window

While the NC machine is processing the data, the progress window is visible (shown in the figure above). The top progress bar indicates the number of tool paths finished in respect to the whole number of tool paths. The lower progress bar indicates the number of points complete in each single tool path.

After the data is finished processing on the NC machine, the progress window closes automatically, after that, either more work is done, or the NC mode is ended.

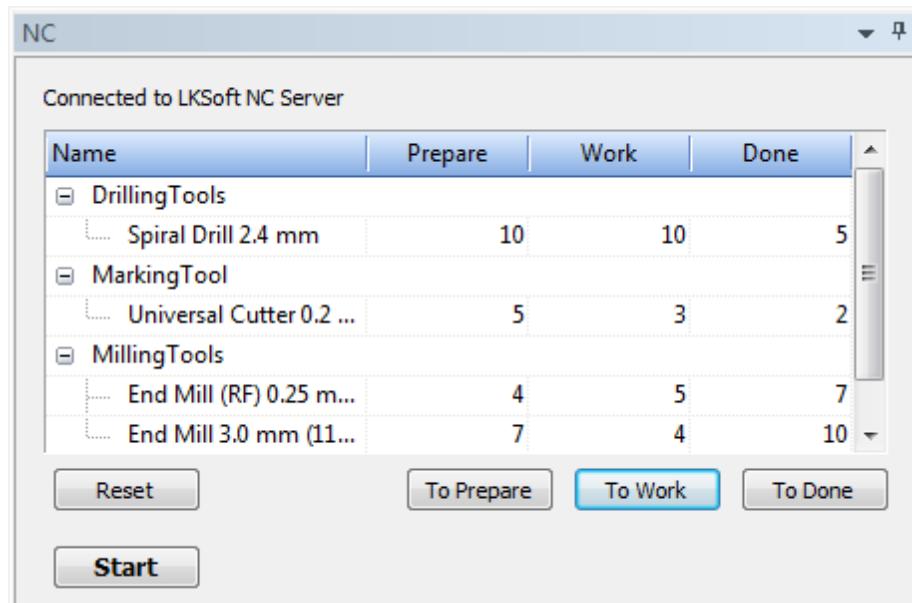


Figure 350. NC control, displaying the tool paths distributed among layers, according to their progress

After some or all of the tool paths are processed, the NC control displays the numbers of tool paths that are processed, as seen in the figure above, *i.e.: some tool paths have already been processed, and another set of paths have been moved to the NC_Work layer.*

The **Reset** button, reverts all the changes done in NC mode, *i.e.: all tool paths are moved back to the NC Prepare layer, any manipulations or modifications are reverted.*

Head Position Coordinates

This section displays the current coordinates of the NC machine head, the coordinates are displayed in the sequence; x, y, and z.

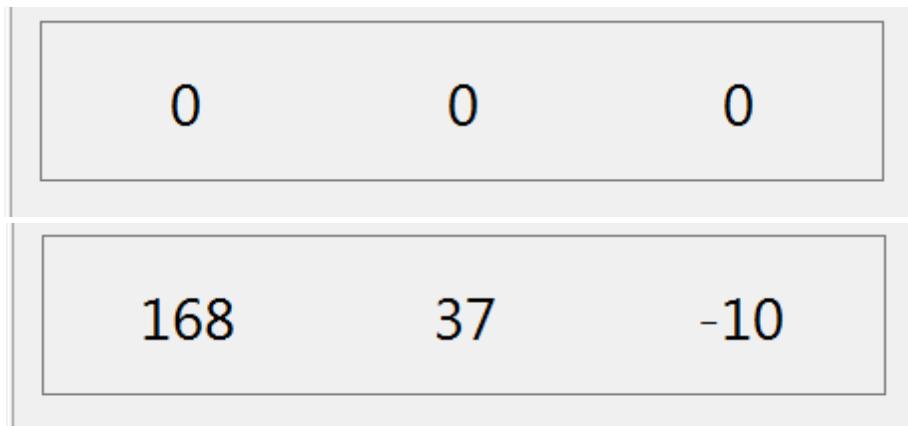


Figure 351. NC control displaying the current coordinates for machine head position

As explained before, *CircuitCAM* only works on 2D designs, the values of the z coordinate are only for drilling, the value "*zero or 0*", indicates that the head is up, and a negative value indicates that the head is drilling down.

Machine Head Controls

This section in the NC control, controls the machine head movements,

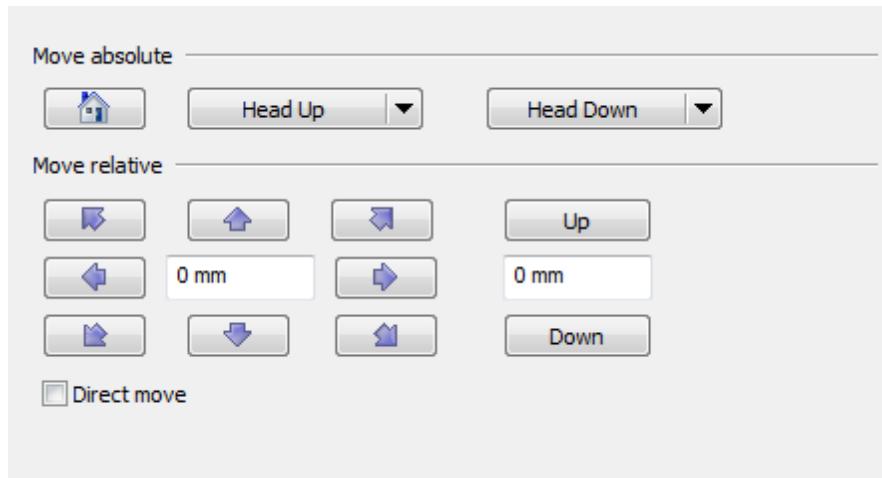


Figure 352. Machine head controls in NC mode

we have two modes to control the position of the machine head;

a. **Move absolute:**

In this section there are three control buttons;

Home Button: Moves the machine head to its home position; the initial head position is automatically read from the machine settings.

Head Up: Retracts (*moves up*) the machine head, up on the Z axis.

Head Down: Moves the machine head down the Z axis (active drill position).

b. **Move relative:**

In this section, it is possible to move the machine head to any direction by a desired value.

A value is entered in the blank field, then the desired direction arrow is clicked, the head moves by the entered value.

When the "**Direct move**" flag is checked, the head place is chosen with a mouse click on the desired place. See figure below.

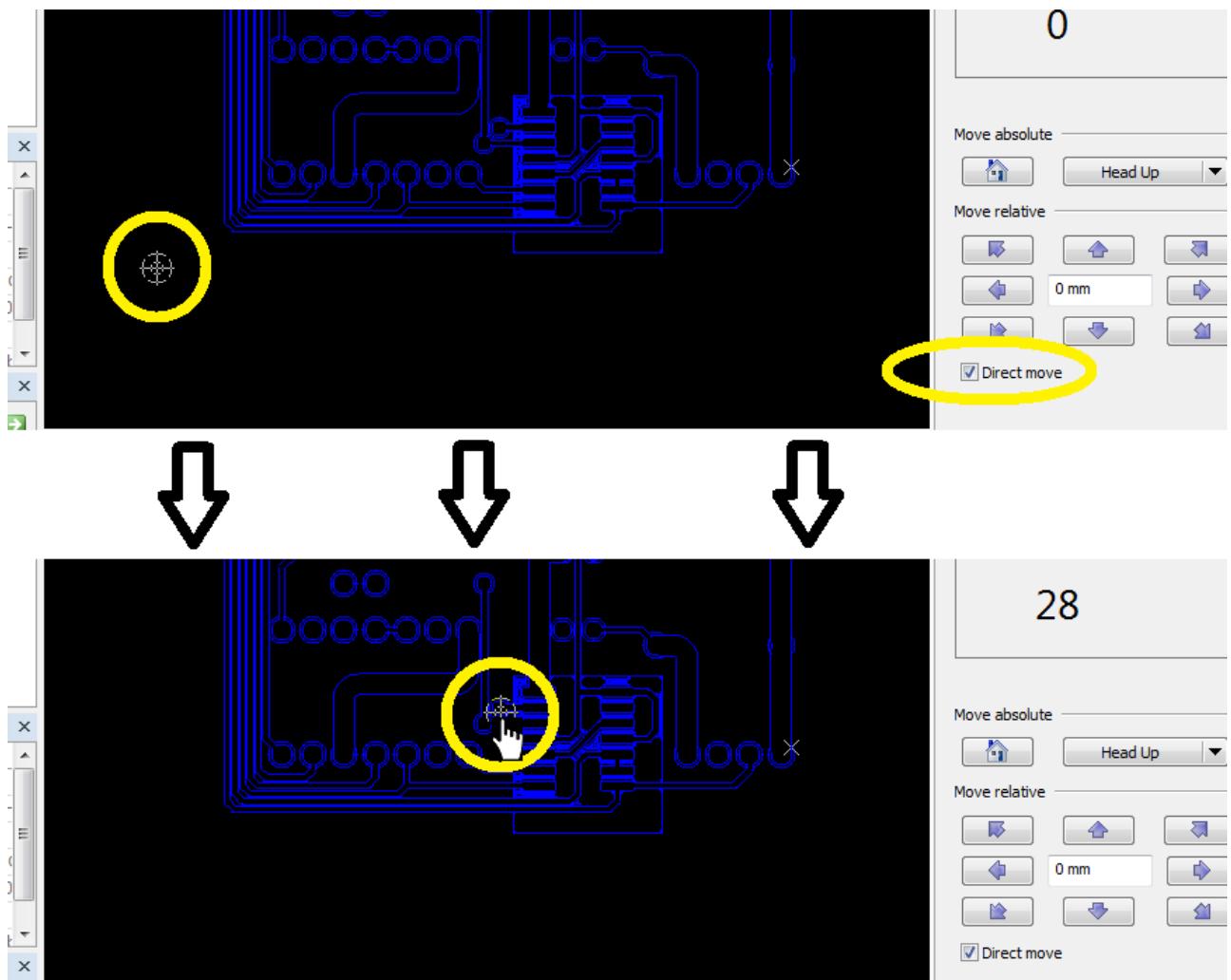


Figure 353. Changing the machine head position using "Direct move"

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.2. Operations and Checks

4.2.1. Base Material with Fixture Hole

The **Base Material** dialogue creates a boundary box around the PCB design and places fixture holes and markers to align for the manufacturing processes.

Inserting Base material procedure:

- Select in the main menu **Insert -> Base material / fixture holes** feature.

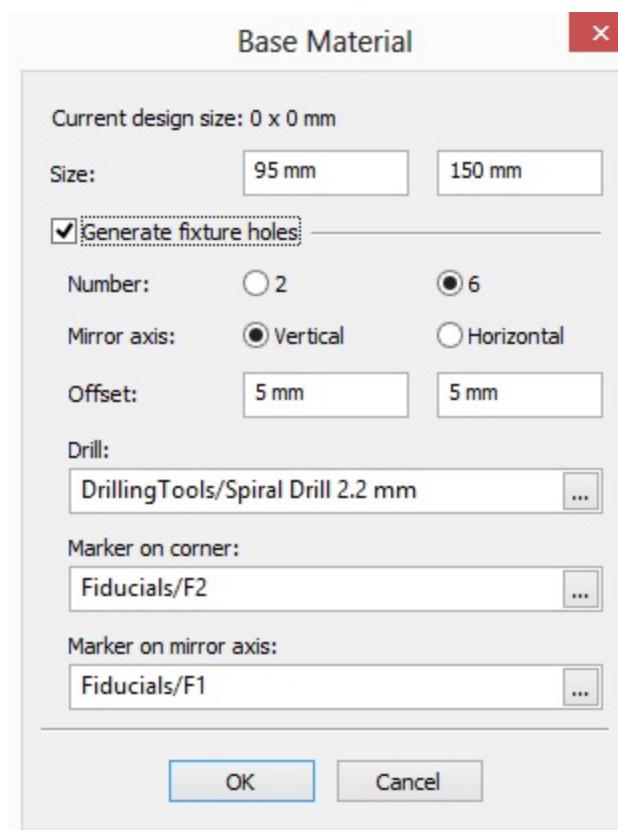


Figure 354. Base Material dialogue

- **Current design size:** Shows the dimension of the current X-Y plane of the PCB.
- **Size:** Sets the desired size of the Base material boundary box e.g.: 100 x 60 mm, 200 x 320 mm, 300 x 400 mm default size is 95 x 150.
- **Generate fixture holes:** If the flag is set *ON* fixture holes will be generated to be used for aligning PCB with the layout for manufacturing.
- **Number:** Sets the number of points to be added to the layout e.g.: 2 or 6
- **Mirror axis:** Sets the orientation of the mirror that will be used in Markers.
- **Offset:** Sets the distance that the hole will be created from the vertical corner and the horizontal corner .
- **Drill:** Sets the drilling tool and so the diameter for the fixture holes.

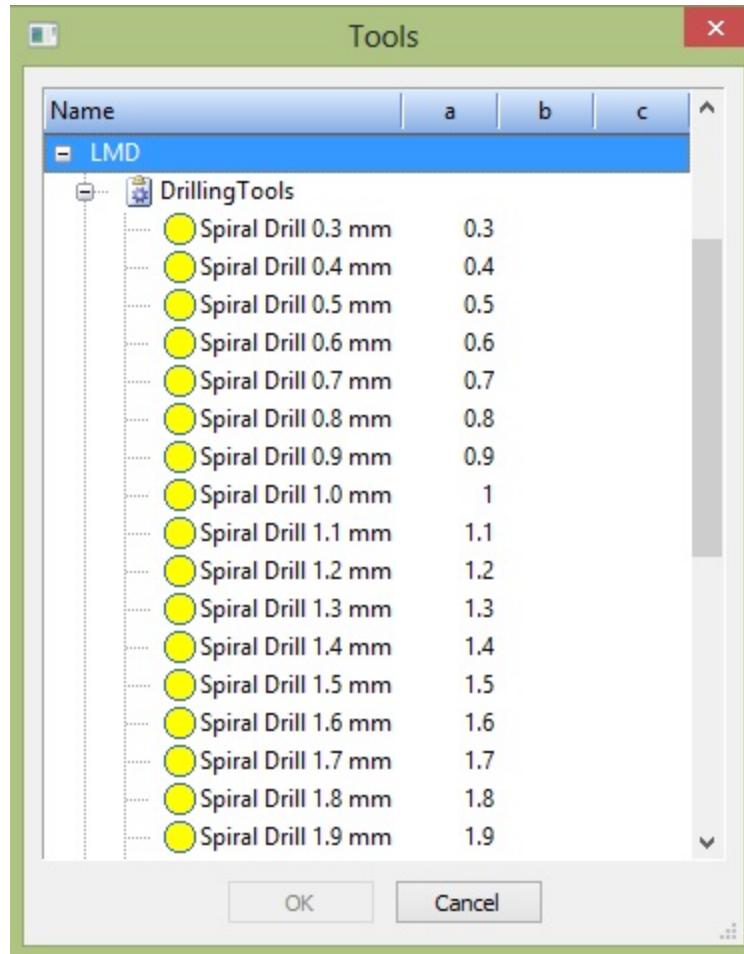


Figure 355. Drilling Tools List

- **Marker on corner:** Indicates the markers to be used for the four corners. The default is F1 from the Fiducials tool-list.
- **Marker on mirror axis:** Indicates the markers to be used for the mirror axis, horizontal or vertical. The default value is F2 from the Fiducials tool-list.

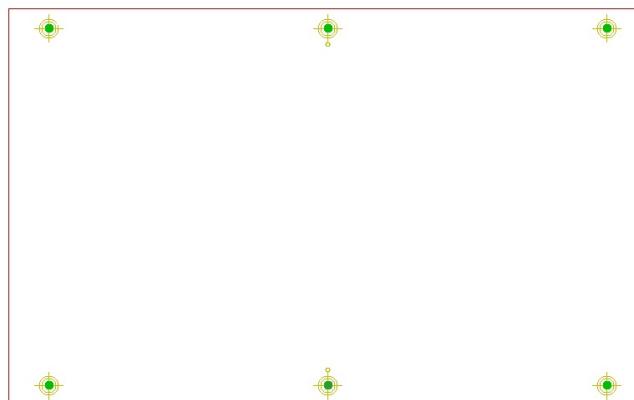


Figure 356. Example of the final Base Material

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.2.2. Copper Pouring

The function is rather similar to function for the creating of a [polygon with cutOut](#).

- Select the items (for example pads, tracks), which should be cut out of the ground area, in the **Graphic** view.
- Select in the main menu **Insert -> Copper Pouring** menu item. The *Select copper pouring ground* mode is turned *On* (see message line in the status bar). The special mouse cursor  appears on the screen.

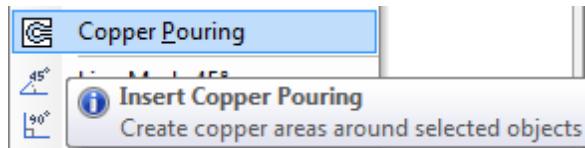


Figure 357. Insert -> Copper Pouring menu item

- Select an area, which should become a ground shield in the **Graphic** view. The **Copper pouring** dialogue window for entering a space between *Pads/Tracks* and the ground appears on the screen.

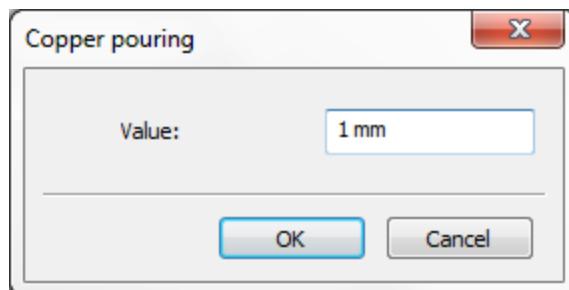


Figure 358. Copper pouring dialogue window

The copper pouring can be performed several times in a sequence for different objects with different widths.

Let us take a sample with two pads (flash-finger and flash-circle) and two tracks (paths). Create a rectangular copper area around these objects.



Figure 359. Flash-finger and flash-circle with paths and rectangular copper area

Now use the *Copper pouring* function with a different spacing around pads and tracks: 0.4 mm for finger, 0.3 mm for flash-circle and 0.2 mm for paths.



Figure 360. Result of copper pouring



Note: This function is only available in *CircuitCAM* variants **STANDARD** and **PRO**.

4.2.3. DataMatrix

The **Data Matrix** barcode is a very high-density two dimensional symbology, that can encode text, numbers, files and actual data bytes. It is a very efficient two dimensional (2D) barcode symbology that uses a small area with a unique perimeter pattern, which helps the barcode scanner to determine the cell locations.

The cells are mainly made up of square modules. Because it can encode letters, numbers, text and actual bytes of data, it can encode just about anything including text characters, Unicode characters and photos. The encoding and decoding process of Data Matrix is very complex and several methods have been used for error correction in the past.

Data Matrix is one of the smallest and most dependable barcode symbologies. Data Matrix is approximately 30 times smaller than a Code 39 barcode representing the same data. Data Matrix is the best choice to use when sending barcodes over faxed documents.

Data Matrix is covered today by an ISO standard, ISO/IEC 16022—International Symbology Specification, Data Matrix is in the public domain, which means it can be used free of any licensing or royalties.

Follow the routine to insert a new text:

- Activate the **DataMatrix** function in the **Insert** menu or click on the **DataMatrix** in the **Insert** toolbar and fill fields of text dialogue window:

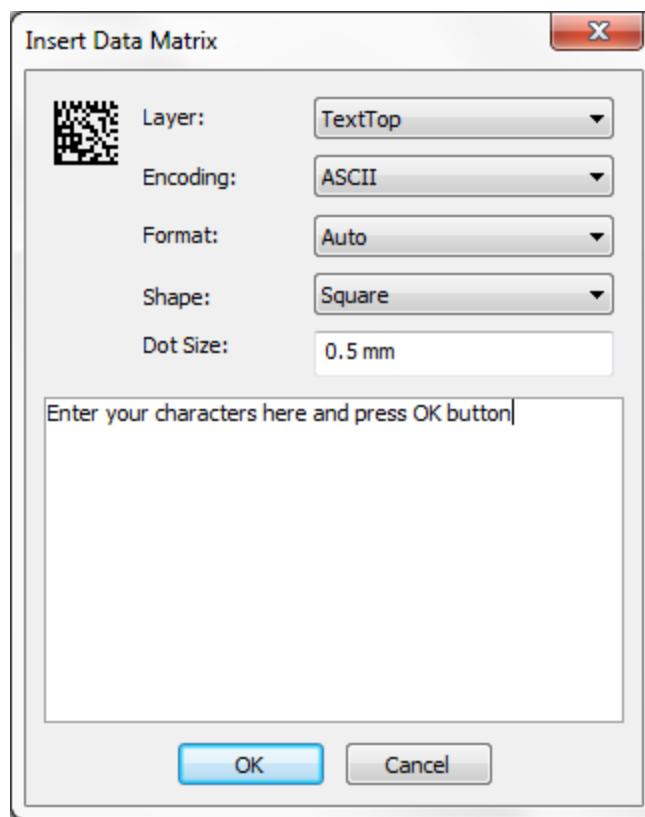


Figure 361. Insert Data Matrix dialogue window

- Set a destination layer in the **Layer** field
- Set an encoding type in the **Encoding** field.
- Set a Data Matrix size in the field **Format**. It varies from 8×8 to 144×144 cells. The size can be chosen automatically by using the **Auto** value.
- Set the shape of cells in the field **Shape**.

- Set the size of one cell in the field **Dot size**.
- Press **OK**.

Set the point for left-bottom corner of the Data Matrix in the graphic window with the mouse and do left-click to fix the D.

Here is an example of a Data Matrix code, encoding the text "Always a cut ahead". Used parameters are shown in the dialogue.



Figure 362. Example of Data Matrix code, encoding text "Always a cut ahead"

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.2.4. Selecting Operations

Pitch

The *Pitch* function searches and selects the graphical objects which have the same pitch, the same spacing and area. To specify a pitch, spacing, and area, the **Selection by Pitch** dialogue window is introduced (edit fields for the value and check boxes to switch *On/Off* space and area). The dialogue window enables the user to define the layer to search on, or search on all layers. If two selected pads are on the same layer, the layer combo box shows this layer, otherwise - it is empty. Three tolerances are specified as well (pitch, spacing, and area).

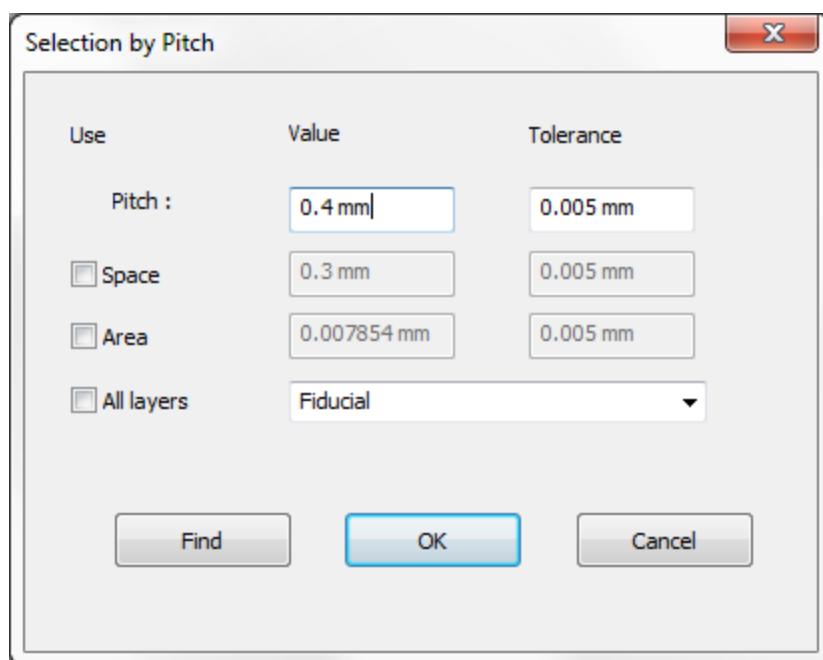


Figure 363. Selection by Pitch dialogue window

This dialogue window can be accessed by one of the following means:

- Via menu **Select -> Pitch**;
- By right mouse button menu function **Pitch of two pads** and pressing on the button **Select** (see [Pitch of two Pads](#));
- From GMC “Exchange by Pitch”.

The **Find** button deselects all previous selections and starts new calculations. All settings of the dialogue are kept for one session.

The **OK** button stores previous settings.

The **current status** in all 3 modes (via mouse menu, **Select -> Pitch** and GMC **Exchange by Pitch**) is:

1. A pad (its area) can consist either from one object (flash, path, rectangle,...), or can consist from a few connected (overlapped) objects on the same level;
2. An area, which consists only of undefined-flash or zero-diameter circle, cannot participate in any pitch operation (selection or exchange), because its polygon size is zero. It means that any such selection prohibits the pitch operation (a message comes up).
3. Areas can belong to different levels.
4. If number of selected areas is either more or less of 2, a message comes up.

You can see more selecting operations in the [Select](#) menu.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.2.5. PCB Design Rule Check (DRC)

CircuitCAM allows the user to check for two design rules on the Printed Circuit Boards (PCB):

- **Inter Min Spacing**, which checks for the minimum distance of networks between two layers.
- **Intra Min Spacing**, which checks for the minimum distance of networks within one layer.

To open the *PCB-DRC* dialogue, select **Design Rule Check -> Printed Circuit Board** from the **Edit** menu.

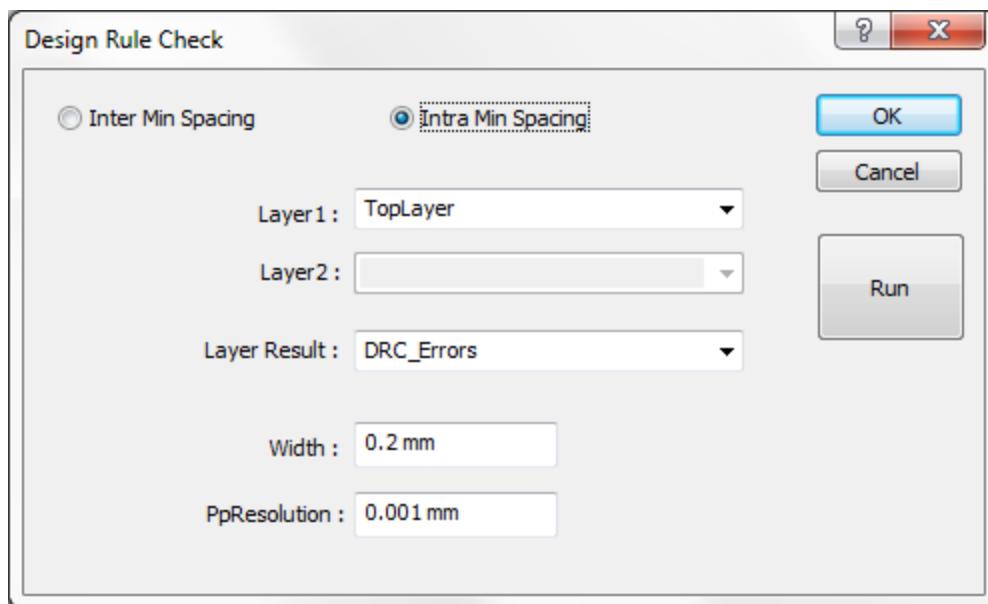


Figure 364. Design Rule Check dialogue window

- **Layer 1:** Specifies the layer to be checked.
- **Layer 2:** Specifies the second layer to be checked for **Inter Min Spacing**. In case of **Intra Min Spacing** mode the combo-box is disabled.
- **Layer Result:** After the rule evaluation, all detected violations are placed as graphic objects on this layer. The layer can be selected from the combo-box or entered as a new one.
- **Width:** Specifies a minimum spacing distance between objects.
- **PpResolution:** Specifies the Resolution of Polygon Package.
- **Run:** Starts the design rule check.

Example: After starting the above settings on the tutor example a few errors are reported.

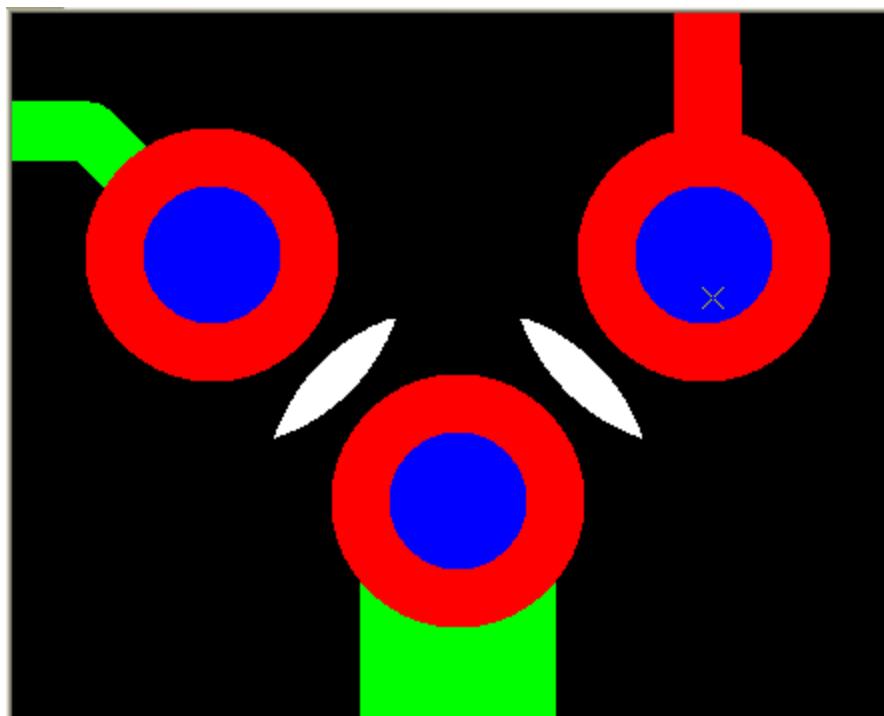


Figure 365. Graphic view with errors (white areas)

The white area indicates where the minimum distance of 0.33 mm is violated.



Note: For a measuring, the anchor-point can be set at arbitrary positions, e.g.: at the outline of a track, with a *ctrl-shift-click*. After the coordinates were switched to relative or polar coordinates, the distances can directly be read out in the status bar. The picture shows that there is a distance of only 0.3 mm.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

4.2.6. Stencil Design Rule Check

IPC-7527, "Solder Paste Printing Acceptance Guide" helps to evaluate and optimize the solder paste printing process.

To check the stencil patterns according to the IPC-7527 guidelines, select **Design Rule Check -> Stencil** line from the **Edit** menu.

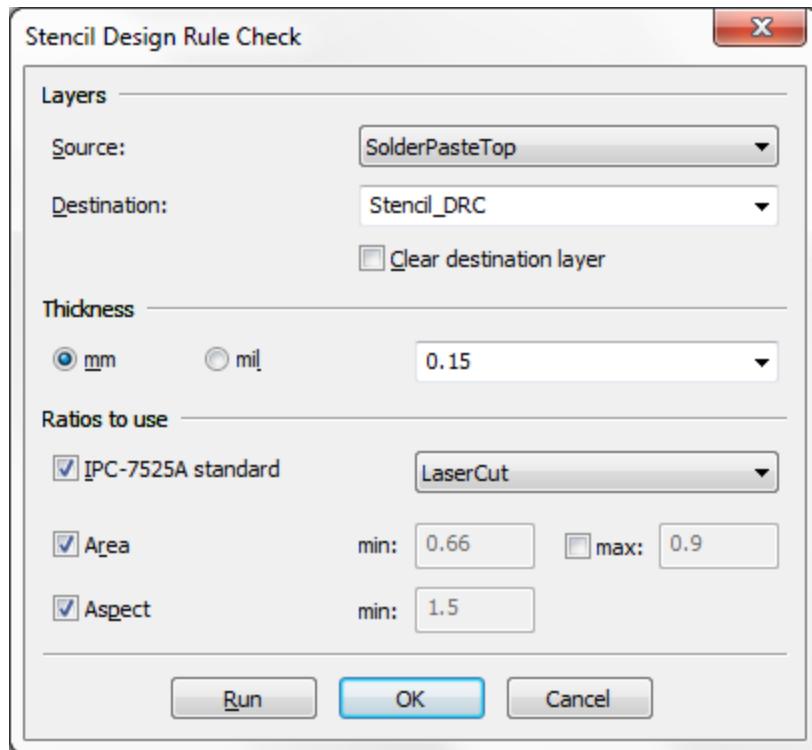


Figure 366. Stencil Design Rule Check dialogue window

Layers group

Contains information about used layers:

- **Source:** Specifies the layer to be checked.
- **Destination:** Objects out of the limits are copied in this layer. Default value is "Stencil_DRC". A user can select the layer name from list or enter some string xxx. The resulting layer will have an extension of selected **Thickness** value, e.g., "xxx_0.1mm" or "xxxX_2mil".
- **Clear Destination Layer:** Deletes all objects from all previous **Destination** layers.

Thickness: Is applied to all objects on the base layer. The **Thickness** is the height of the stencil foil. There are pre-defined values 2, 3, ... 10 mil (if **mil** selection button is chosen) or 0.05, 0.075, 0.1, ... 0.25 mm (if **mm** selection button is chosen).

Ratios to use:

Specifies the parameters for area-ratio or aspect-ratio limits calculation. User can choose to check only *Area-ratio*, or *Aspect-ratio* only, or both (check boxes **Area** and **Aspect**):

- **IPC-7572A standard:** Allows the user to specify one of the application area from the combo box: "LaserCut", "Electroformed", "Chem-Etch". Each type has own predefined *Area-min*, *Area-max* and *Aspect-min* values:

	LaserCut	ElectroFormed	ChemEtch
<i>Area-min</i>	0.66	0.5	0.9
<i>Area-max</i>	0.9	0.66	0.9
<i>Aspect-min</i>	1.5	1.5	1.5

- **Area:** If *On*, checks *Area ratio*. When the **IPC-7572A standard** flag = *Off*, a user can set **min** and **max** values manually.
- **Aspect:** If *On*, check *Aspect ratio*. When the **IPC-7572A standard** flag = *Off*, a user can set **min** value manually.

Run: Starts the stencil design rule check.

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

5. Hotkeys

The table below presents the list of predefined *Hotkeys* for *CircuitCAM* commands. The user can modify, assign new *hotkeys* in [General Settings](#) dialogue window, **Keyboard** panel.

Category	Function	Hotkey	Affects/Activates
General	Help about current contents	F1	
	Toggle between CircuitCAM views	Ctrl + Tab or Alt + F7	
	Switch between open views	Alt + F6	
	Zoom in	Page Up or Mouse Wheel Up	Cursor position
	Zoom out	Page Down or Mouse Wheel Down	Cursor position
	Remove item(s)	Delete	Selected item(s)
	Cancel/interrupt current operation	Esc	
	Pan the graphic view; up, down, left, and right	Cursor keys; up, down, left, and right	
	Set Anchor Point	Ctrl + Shift + Left Click	
	Close	Alt + F4	Active window
File menu	New document, based on default template	Ctrl + N	
	Open	Ctrl + O	
	Save	Ctrl + S	Active document
	Print	Ctrl + P	Active document
Edit menu	Undo	Ctrl + Z	
	Redo	Ctrl + Y	
	Cut	Ctrl + X	
	Copy	Ctrl + C	
	Paste	Ctrl + V	
	Remove	Delete	
	Move Selected	Space	
	Duplicate Selected	Ctrl + Space	
	Set Cursor	Ctrl + Shift + C	
	Set Cursor to Zero	Ctrl + Shift + 0	
	Set Anchor Point	Ctrl + Shift + A	
	Anchor to Centre Point	Ctrl + Shift + Z	
	Anchor to Mass Point	Ctrl + Shift + M	
	Set Zero Point	Ctrl + Shift + N	
	Move Layer to Zero	Ctrl + Shift + L	

Category	Function	Hotkey	Affects/Activates
Select menu	Select All	Ctrl + A	
	Select next object according to sequence	Ctrl + Shift + Right	
	Select previous object according to sequence	Ctrl + Shift + Left	
	Select first object according to sequence	Ctrl + Shift + Up	
	Select last object according to sequence	Ctrl + Shift + Down	
Modify menu	Add Point/Segment	Ctrl + T	
	Cut Point/Segment	Ctrl + U	
	Delete Point/Segment	Ctrl + D	
	Move Point/Segment	Ctrl + M	
	Create or Move Arc	Ctrl + E	
	Combine	Ctrl + B	
	Close	Ctrl + Shift + Q	
	Path to Polygon	Ctrl + Shift + P	
	Combine to Polygon	Ctrl + K	
	Convert to Polygon	Ctrl + Shift + Y	
	Convert to Closed Outline Path	Ctrl + Shift + W	
	Denest all Cutouts	Ctrl + Shift + I	
	Denest single Cutout	Shift + J	
	Convert to Circle	Ctrl + Shift + K	
	Draw to Flash	Ctrl + Shift + D	
	Convert to One Flash	Ctrl + Shift + H	
	Compare Shape	Ctrl + Shift + S	
	Compare Bounding Box	Ctrl + Shift + B	
	Compare Outline	Ctrl + Shift + T	
	Denest Flashes/Instances	Ctrl + Shift + E	
	Size Modification	Ctrl + Shift + U	
	Step and Repeat	Ctrl + Shift + R	
	Create Bounding Shape	Ctrl + Shift + G	
	Create Outline	Ctrl + Shift + O	
	Perfect Shape	Ctrl + Shift + V	
NC/Tool path menu	Create gap	Ctrl + G	
	Reverse	Ctrl + R	
	Move to next	Ctrl + J	
	Move to previous	Ctrl + I	
	Move to first	Ctrl + F	
	Move to last	Ctrl + L	
Analysis menu	Measure	Ctrl + Shift+ X	
View menu	Zoom Window	Alt + draw box with	

Category	Function	Hotkey	Affects/Activates
		mouse	
	Zoom In	Page Up <i>or</i> Mouse Wheel Up	
	Zoom Out	Page Down <i>or</i> Mouse Wheel Down	
	Overview	Home	
	Previous	Insert	
	Refresh	F5	
	Full Screen	Ctrl + Shift + Enter	
	Properties	Ctrl + Q	

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

6. Epilogue

At the end, the whole team behind *CircuitCAM* wishes that this was a helpful tour around the program.

We really wish that *CircuitCAM* will comply with all your designing/production needs.

Any suggestions are more than welcome;

Please forward your notes to: info@lksoft.com

You are welcome to visit our website for more information.

<http://www.circuitcam.com/>

<http://www.lksoft.com>

www.dct-china.cn

Copyright © DCT China Co.,Ltd, since 2014 (LKSoftWare GmbH, 1992-2013). All Rights Reserved.

Glossary

A

ATE

Aperture Template Editor

B

BGA

Ball Grid Array

C

CAD

Computer Aided Design

CAM

Computer Aided Manufacturing

CSG

Constructive Solid Geometry

D

DIL / DIP

Dual in-line package

DRC

Design Rule Check

DXF

AutoCAD text format

E

EMF

Enhanced Metafile for Windows

G

GMC

Geometry Manipulation Centre

L

LMD

Laser, Milling, Drilling

N

NC

Numerical Controlled (machine)

P

PCB

Printed Circuit Board

PLCC

Plastic Leaded Chip Carrier

PNG

A bitmap image file format that supports lossless data compression

R

RF

Radio-frequency

S

SMD

Surface-mount devices

SMT

Surface-mount technology

SOT

Small-outline transistor

STEP

STandard for the Exchange of Product model data, ISO 10303

W

WMF

Windows Metafile

Index

A

About 31
Activation; Offline 25
Activation; Online 21
Activation; Software Registration 20
Analysis; Menu 107
Anchor Point; Set Anchor Point 205
Aperture List 39
Aperture List; Examples 153
Arc 207
Area of Selected Objects 161
Area; Selection 200
ATE; Aperture Template Editor 317

B

Barco; Tools Properties 232
Barcode 215
Base Material 71, 365
Batch and Background Processing 165
Boolean Operations 184
 Difference 185
 Intersection 186
 Paths off Area 190
 Paths on Area 189
 Union 185
 XOR 188
Breakout Tabs 69

C

CAM & CAT Documents 140, 324-325
Circle 173
 Insert 207
 Properties 196
Close; Exit Program 85
Configurable Geometry Engine 268
Context Menus
 Graphic View 201
 Jobs View 240
 Layers View 221
 Layouts View 286
 Libraries 289
 Tools View 227
Contour Routing 346
Convert to Circle 178
Copper Pouring 367
Create Gap 202
Create Package 292
Cursor; Set Cursor 205
Cutout 210

D

DataMatrix 369
Denest; Polygon Cutout 177
Denest; Polygon Cutout Hole 178
Directories; Installation 19
Directories; Settings 123
Diverse 231
DRC; Design Rule Check
 PCB 371
 Stencil 373
DXF; Tools Properties 232

E

Edit; Menu 94
Epilogue 378
Esc Key 139
Excelon; Import Assignments 46
Excelon; Tools Properties 229
Export 150
 Export Jobs 244

Export Layout Instance 151
Export Strategy 130
Export Tasks 247
Export Tasks Properties 248
Production Data 84

F

File Formats 310
Barco-DPF 320
CAM 324
CAT 325
DXF 327
EESOF 327
Excelon 327
G-code 329
Gerber 334
Gerber X 330
HP-GL 335
ISO 10303-210 337
LMD; Laser Mill Drill 337
ODB++ 342
Sieb & Meyer 342
File; Menu 93
Fixture Holes 365
Flash 173
Properties 197
Flash Attributes 234
Flash Rotations Toolbar 117
Fonts 215
Footprints 295
Footprints; Create 295
Full Screen Toolbar 117

G

G-Code; Tools Properties 232
General Settings 122
Gerber; Import Assignments 40
Gerber; Import Panel 147
Gerber; Tools Properties 334
GMC
Compare 307
Glue Dots 305
GMC Toolbar 117

Library 308
Predefined Shapes 300
Shape Modification 302
Size Modification 304
GMC; Geometric Manipulation Centre 297
Graphic Manipulation
 Add Point/Segment 174
 Area of Selected Objects 161
 Combine 175
 Combine to Polygon 176
 Compare Bounding Box 180
 Compare Outline 181
 Compare Shape 180
 Convert to Closed Outline Path 177
 Convert to Instance 180
 Convert to One Flash 179
 Convert to Polygon 176
 Create Bounding Shape 182
 Create or Move Arc 175
 Create Outline 182
 Cut Point/Segment 174
 Delete Point/Segment 174
 Denest Polygon Cutout 177
 Denest Polygon Cutout Hole 178
 Draw to Flash 178
 Exchange with Flash 180
 Move Point/Segment 175
 Path to Polygon 175
 Perfect Shape 183
 Polygon to Path 184
 Remove Holes 184
 Step and Repeat 181
Graphic Objects 167
Graphic View 166
Grid/Unit Toolbar 115
Grid; Display Grid 168
Grid; Snap to Grid 168
GUI
 Graphic View 166
 Main Menu 93
 Analysis 107
 Edit 94

File 93
Help 110
Insert 99
Library 105
Modify 101
NC/Tool Path 104
Select 97
Settings 107
View 109
Main User Interface 91
Message Log 120
Preview/World View 138
Properties Window 194
Status Bar 118
Toolbars 111
 Flash Rotations Toolbar 117
 Full Screen Toolbar 117
 GMC Toolbar 117
 Grid/Unit Toolbar 115
 Insert Toolbar 114
 Layer Toolbar 115
 Relative Rotations Toolbar 116
 Select Toolbar 113
 Standard Toolbar 112
 Zoom Toolbar 113

H

Help 137
Help; Menu 110
Holes; Remove Holes 184
Hotkeys 375
HP-GL; Tools Properties 231

I

IEC61182; Tools Properties 229
Import 144
Import; Assignments 87, 131
Import; Excellon Settings 127
Import; Extended Gerber 50
Import; Gerber Settings 126
Import; Production Data 38, 144
Import; Simple Gerber 38

Insert

Arc 207
Circle 207
Closed Path 209
Open Path 209
Pad 209
Polygon 211
Polygon Cutout 210
Rectangle 212
Rubout Area 212
Text 213
Insert Toolbar 114
Insert; Menu 99
Inserting Operations 206
Installation 19
Insulate Milling 349
Introduction 16

J

Jobs View 238
Context Menu 240
Toolbar 241

K

Keyboard Shortcuts 375
Keyboard Shortcuts; Customize 135

L

Laser Fill 77
Laser Insulate 79
Laser Scanner 344
Layer Properties 223
Layer Toolbar 115
Layers View 218
Layers View; Context Menu 221
Layers View; Toolbar 222
Layers; Active Layer 219
Layers; Move 55
Layers; Order 54
Layers; Rotate 58
Layouts View 285
Layouts View; Context Menu 286
Layouts View; Toolbar 287

Libraries View 288
Libraries; Context Menu 289
Libraries; Library Components 289
Libraries; States of Libraries 289
Libraries; Toolbar 290
Libraries; Types of Libraries 289
Library; Menu 105
Line Drawing Modes 208
List Views 216
LMD 231

M

Main User Interface 91
Manipulation Operations 212
Markers 172
Measure 160
Message Log 120
Milling Contour 67
Milling Direction & Sequence 71
Modify; Menu 101
Mouse Precision 160

N

NC
I++NC Server 356
Machine Head 359
Machine Head Controls 364
NC Controls 360
NC Coordinates 363
NC Layers 358
NC Mode 355
NC Mode Layout 357
NC/Tool Path; Menu 104
New; Documents 143
New; Templates 143

O

Open; Documents 141

P

Package Recognition 294
Packages & Footprints 291
Pad 209
Panning 170

Path
Properties 195
Perfect Shape 183
Pitch of Two Pads 160
Polygon

Insert 211
Properties 197
Polygon Cutout Hole; Denest 178
Polygon Cutout; Denest 177
Polygon; Polygon Operations 193
Postscript 230
Postscript Properties 230
Predefined Jobs; Default Template 34
Predefined Layers; Default Template 33
Predefined Libraries; Default Template 35
Predefined Tools; Default Template 34
Preview / World View 138
Print 154
Printer Calibration 157

Properties 194
Circle 196
Flash 197
Path 195
Polygon 197
Rectangle 196
Selection 198

Q

Quick Access 238

R

Rectangle 173
Insert 212
Properties 196
Relative Rotations Toolbar 116
Requirements; System Requirements 19
Reset; Layout 120
Reset; Settings 133
Reverse 202
Rotation & Mirroring 203
Rubout Area 71
Insert 212

S

Save; Documents 149

Scaling 192

Scripts 258

 Add Script 259

 Make Derived 264

 Properties 260

 Run Script 263

 Script Commands 269

 Script Tasks 265, 269

Select; Menu 97

Selection

 Operations 370

 Properties 198

 Selection of Graphic Objects 199

 Sequence 200

 Single Object 199

 Toolbar 113

Settings

 Customize 133

 General Settings 122

 Menu 107

 User Settings 30

Sieb & Meyer; Tools Properties 229

Size Modification 191

Size Modification Dialogue 191

Slection

 Area 200

 Standard Toolbar 112

 Statistics 162

 Status Bar 118

 Stencil Frame Generator 241

 Striping & Stripping 355

 Swap Order 206

 Synchronize 142

T

Template; Create Template File 85

Template; Synchronize Template File 142

Text 213

Tool Compensation 344

Tool Edit Mode 237

Tool List; Translation File 44

Tool Path; Sequence 349

Toolbars 111

Toolbars; Customize 134

Tools 225

Tools List 229

Tools List; Properties 229

Tools Properties 233

Tools View 225

Context Menu 227

Toolbar 228

Transform; Export Transforms 252

Transform; Tools 235

Translation File 311

Translation File; Tool List 44

Tutor Example 35

U

Usage Guide 33

User; Information Panel 124

V

Variants; Program Variants & Releases 16

View; Menu 109

View; View Arrangement 119

Z

Zoom 168

In & Out 169

Preview/World View 138, 170

Previous Views 170

Selected 169

Toolbar 113

Zoom Window 169

