TARGET 3001!

# TARGET 3001! V11 data sheet

● 32-bit-windows-power ● program and manual completely in English and German language ● toggle between schematic and PCB
● realtime data integration (forward-/back-annotation) ● realtime ground plane computation ● realtime air wire optimization ● autoplacer ● contour autorouter (shape based, gridless) ● drag & drop library browser ● drag & drop component placement ● clipboard as a documentation interface ● intuitive windows user interface ● discover version as free viewer software ● undo/redo (50 steps) ● interfaces: Gerber IN/OUT, XGerber, Excellon, Sieb & Meyer, DXF, PostScript, Bitmap In, ASCII In/Out („open data-interface"), XML in/out, generic HPGL out ● insulation milling as outline engraving (directly from the program!) ● maximum accuracy: vector graphics, resolution: 1nm = 1/1.000.000mm = 1/25.400.000 inch ● world coordinates: 1 inch on screen = 1 inch on PCB (when zoom = 100%) ● 1m x 1m PCB-/schematic area, 100 PCB layers, 100 schematic pages ● components in current project directly editable: change pad, modify package etc. ● flip-chips and ball-grid-arrays (BGAs) possible ● rotate by any angle e.g. for COB technology (chip on board) ● freely displacable tools

**In schematic mode:**
● permanent „forward-annotation" to PCB ● more than 5000 logic components (DIN/IEEE and old norm) ● automatic component numbering ● inserted symbols can still be edited or adapted (independently from library) ● library browser for easy library management ● Component Management System (CMS) ● import of Mentor and Orcad netlists if needed and output of individual component lists and netlists (also in Protel and Orcad format) ● BOM (bill of material) with item number, purchase prices, deliverer etc. and fields to be individually defined ● up to 100 schematic pages each up to 1m² ● electrical design check ERC

**In PCB mode:**
● permanent „back-annotation" to schematic ● component packages in the layout can be edited freely at any time ● air wires continually updated in display to aid placing ● up to 100 layers (copper, solder stop, solder paste, gold etc.) ● any solder pad shapes: circular, octagonal, oblong, etc. ● any width of track, circular track, bezièr-curves, spirals, teardrops and freely defineable padstacks ● configurable autoplacer ● interactive and automatic routing, gridless shape based contour auto router with t-connections (copper sharing) and maintaining of all design rules, single-, double- and multilayer routing, routing of SMDs, of components and of solder pads rotated at any angle ● floating ground planes calculated in real-time - without computation delay ● configurable design rule check (DRC) ● PCB generation and routing without schematic ● gerber-file import for further editing (from other design software programs). ● lots of PCB manufacturers directly accept *.T3001-files from TARGET

**System requirements:**
● 486DX-processor, recommended is an AMD K6 or an Intel Pentium II ● 32MB RAM, recommended is 64MB ● screen resolution: minimum 800x600 pixels and 256 colours ● CD-ROM- and 3.5" drive for installation neccessary ● Windows 95/98/ME/NT4/2000/XP

**Mixed mode simulation:**
● completely reviewed new graphical user interface ● no longer Spice blocks ● dialogues as assistants ● help in dialogues- with graphical explanations ● eased model edition ● modelbrowser enlarged (with Syntax-Error messages also) ● Subcircuit Generator enlarged ● Level ("Simulation-group") new defined, generailzed ● all basic elements of the simulator available in the library, virtual components too ● Component import (with model) in tree view technique possible ● PWL-Editor ● Graphical entry of PWL-curve ● interactive elements self definable: ● individual virtual instruments can be generated ● texts as value display ● filled objects as flashing elements ● rectangles as oscilloscopes ● MOSFETs: parameters L and W also implemented into the model card (PSPICE-Syntax), so MOSFet-Models from manufacturers (Internet, gen PSPICE-Syntax) can be imported ● SPICE3F5 and PSPICE compatible ● Graphical entry of the circuit – circuit simulation ● interactive circuit symbols ● mixed mode simulation (analog + digital) ● Analyses: DC sweep, AC sweep and transient ● automatical control of step widths in transient analysis ● event driven digitalsimulator ● delay, setup, hold, ... ● times for digital basic elements ● postprocessor (oscilloscope) for view and analysis ● graphical subcircuitgenerator

**EMV-analysis:**
● elektromagnetic compatibility (EMC) ● From the beginning of your design you should be sure of all EMC- options. In the EMC-Analysis you first have to define several parameters concerning the PCB and concerning the already placed tracks. Critical tracks are automatially highlighted and checked with respect to their emissions and self interference. **Don't fear EMC!** By using EMC-Analysis the PCB is checked against all EMC design rules whilst designing. The handling and evaluation of the results are realized by the Windows tree-view-technique. The following calculations are implemented: ● galvanic, inductive, capacitive and radiation coupling ● computation of the coupling factor, the wave resistance and maximum track length ● numerous tips for successful EMC-adequate PCB design.

**What´s new in V11**
● new Hybbrid Autorouter ● generating panels ● generating testing points for measurments ● selection assistant (logical selection of Elements for properties) ● Eagle Export ● frontpanel engraving tool ● automatic PCB enquiries and -orders ● automatic component enquiries and –orders ● listing the last 20 components used

Version 11 is steadily developing! Be sure having downloaded your latest release free of charge. See TARGET menu Help/Free Update. Within V11 an online DRC is planned as well as the generation of ground planes as grid.