TARGET 3001! QUICK REFERENCE GUIDE

MOUSE KEYS:

M1 = A short click of the left mouse button
M2 = A short click of the right mouse button (context menu)
M3 (M3H) = Scroll the mouse wheel (Zoom)
M11 = Double click the left mouse key (Edit element)
M22 = Double click the right mouse key (zoom out)
M12 = Click both right and left mousekeys (= ESC)
M1H = Move mouse with the left mouse key held (without object: Catch window)
M1H = Move mouse with the left mouse key held (with object: displace)
M2H = Move mouse with right mouse key held (left to right: enlarge)
M2H = Move mouse with right mouse key held (right to left: fit to screen)
[Shift]+[M1] = Add to selection
[Shift]+[M2H] = Move complete page (alternatively: M3H)

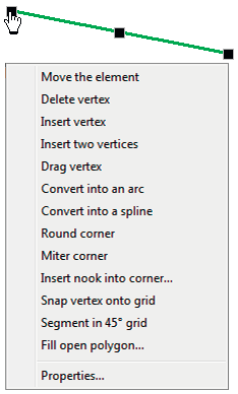
KEY COMBINATIONS:

[F1]	Help	[%]	Set scale
[Ctrl]+[F1]	Check project	[e], [@]	Edit selected elements
[F2]	Component database	[Del]	Delete selected elements
[F3]	Schematic ↔ PCB (Cross Probe)	[f]	Find and select component/signal
[Shift]+[F3]	Schem. ↔ PCB (without CP)	[g]	Drag (sharp bend, arc or curve)
[F4]	Toggle unit (mm, µm, nm, inch, mil)	[i], [Ins]	Insert symbol/package
[F5]	Grid visible/invisible	[m]	Mirror selected elem. horizontally
[F6]	Toggle snap on grid	[Shift]+[m]	Mirror selected elem. vertically
[F7]	Fit to screen	[n]	Refresh window (new)
[F8]	Toggle crosshair view	[o]	Edit options (in drawing mode)
[F9]	Simulation/Autorouter	[r]	Insert reference symbol to schem.
[F10]	Back to pointer mode	[Ctrl]+[r]	Insert rests symbols
[F11]	Create XGerber and Excellon	[s]	Select single element
[F12]	Calculate air wires new (PCB)	[Shift]+[s]	Select additional element
[1]	Place pin/pad	[t]	Rotate (turn) around cursor tip
[2]	Place signal wire/track	[Shift]+[t]	Set rotation angle and rotate
[Ctrl]+[2]	Draw line	[u]	Measure/place dimensioning
[3]	Draw triangle (solid)	[v]	Rename signals
[Ctrl]+[3]	Draw triangle (outline)	[w]	Draw air wire (PCB w/o schem.)
[4]	Draw rectangle (solid)	[x]	Export symbol/package
[Ctrl]+[4]	Draw rectangle (outline)	[y]	Selected elements to a symbol
[5]	Draw polygon (solid)	[„]	Enter text/variables
[6]	Draw signal polygon (ground plane)	[.]	(= full stop) node/via, change copper side while „place track“
[0] (= ZERO)	Draw circle (solid)		
[Ctrl]+[0]	Draw circle (outline)	[,]	Change side without via
[a]	Draw a torus (an arc)	[:] = (COLON)	Change copper layer without via
[b]	Place a bus / Place a wire bridge	[#]	Display only contour, X-Ray view
[c]	Catch window	[ ] (SPACE)	Toggle bending modes
[Shift]+[c]	Additional catch window	[+]	snap on coordinate/element
[d]	Move/Displace selected elements	[esc]	Exit mode (back to pointer mode)

SET GRID: Recommended: Schematic: 1/40“ = 25mil = 0.635mm PCB: 1/40“ or eg. 4mil
COORDINATES: Toggle absolute/relative using key [Home] at cursor position.
SWITCH BETWEEN SCHEMATIC AND LAYOUT: or [F3] cross probe
IMPORT A SCHEMATIC SYMBOL:
Drag and drop symbols from the sidebar M1 to one of these symbols opens the database in this particular component group. M2 on a sidebar symbol allows its presetting.

Also use [i] or [ins] or toolbar icon
SEARCH COMPONENT: Enter e.g. „LM7805“ to the search line. But e.g. „10K smd“ search by parameters using
POWER SUPPLIES insert separately as a „Rest“ component.
REFERENCE SYMBOLS/INSERT GROUND separately as well.
NEW SCHEMATIC PAGE: See sidebar right.
SEARCH AN ELEMENT IN PROJECT: respective Elements will be highlighted.
DRAWING FRAME: Load like a component.
WIRE A CONNECTION PIN: Icon (left) or [2]
SNAP ON GRID: Wire snaps to connection. Size of the snapping square: 1.5 x grid size, max. 80mil (2mm).
TOGGLE BENDING MODE: [SPACE BAR] during placement or from the sidebar right.
PLACE A NODE (schematic) or **PLACE A VIA** (layout): keyboard key [.] The layer changes.
DISPLAY NAME OF SIGNAL WIRE: [n] Place the mouse middle to the signal, press [n]. Also see: Settings/Settings/Options
GIVE A SIGNAL A NAME: M11 on a signal. Set highlighting range.
OPTIONS DURING PLACEMENT: keyboardkey [o]
SELECT, HIGHLIGHT ELEMENT: by M1 on handle or M1H and mouse movement (catch window)
REACH A HIDDEN ELEMENT: Place mouse over the element in question and press [s] as often till it flashes. Now press key [e] for edit or [del].
EDIT: [M11] on handle or [M1]+[e]

DISPLACE ELEMENT BY GRID: Select, press [d] and displace by arrow keys.
IMPORT A PACKAGE PROPOSAL TO THE LAYOUT:
Drag & drop from the proposal list or use [Ins] or [i]
PLACE TRACK: or [2]
BUTTON EXTENSION: M1H on will save track options on this button.
PLACE A VIA: While routing press fullstop [.], layer changes. Add [Ctrl] avoids layer change. In routing mode press [o] to set track options.
LAYER HISTORY: Left: Current drawing layer. Right: The three recently used layers. It distinguishes copper/drawing layers.
M2 on layer number: Activate drawing layer. Grayed = non-Cu layer. Cu coloured = Cu layer. Box ticked: Layer is visible. Set layer functions by the layer tool. M11 upon a layer line allows to set it's function. „Assignment“ refers to the copper side a layer function is meant for. Single, double sided layout: See layer tool to the right side of dialog.



EDITION OF LINES, SIGNALS AND TRACKS: Highlight a segment and see black end boxes. M2 on a box opens a contextmenu (left)

DRAW A COMPONENT: Schematic menu: Components/Create a new component select, generate or draw a package first so that the symbol later can refer to it. Begin drawing with the magic wand in the

Component sidebar on the left, if an assignment or a generation isn't possible. Subsequently go through the steps. Set grid 25mil (0.635mm) or 4mil (SMD). Click to define position of handle cross. Save package to the data base by the sidebar icons or by keyboard key [x]. Now continue with the symbol.

3D-VIEW:
In layout view press 3D button.

MODIFY 3D MODEL/DRAW IT NEW: Press [F2], component browser opens. Now click M2 on the 3D image of the part for modification or creation if there is no model yet. Save the 3D model within the package to the data base.

STEP EXPORT: File/Input-Output formats/File transfer/Documentation/STEP export (3D).

START SIMULATION IN SCHEMATIC:
[F9] or and there:

In the dialog Assistant note the

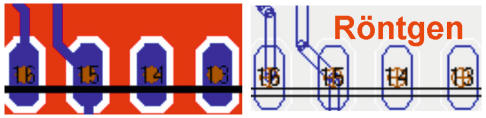
= everything OK.
= there are hints
= there are warnings
= there are errors. See Help.

Start simulation

CREATE A PCB OUTLINE: Menu Actions/Automatisms & Assistants/PCB Outline Wizard. Or draw it manually on layer 23, PCB outline. Cutting edge is middle of the line drawn.

GENERATE GROUND PLANE: Menu: Actions/Ground planes Draw ground polygon [6]. Assign the signal, e.g. GND. These ground polygons know two conditions: „filled", means rendered and „unfilled" means not yet rendered. [<] hides all polygons. [>] recalculates all polygons.

TOGGLE BETWEEN SOLID AND X-RAY VIEW:
Use the hash key [#]:



DELETE AN AIR WIRE: Delete the signal name from pin (schematic) and pad after M11.

START AUTOROUTER: In layout press [F9] or use menu Actions/Automatisms & Assistants/Autorouter.

CAN'T HIGHLIGHT A SINGLE DRAWING ELEMENT but always highlight the complete part instead? Release the button (left). If depressed, a M1 hit always highlights the entire part. Alternatively: M2 in empty space and unmark first item in context menu.

EXCHANGE A SYMBOL: Delete it from the schematic and insert a different one from the database. Signal wire ends are kept.

CHANGE VALUE OR TYPE: M11 on handle, then dialog-button: [CHANGE VALUE OR TYPE].

EXCHANGE A PACKAGE: Delete it from the layout and insert a different one from the database. Wire ends are kept.

MOUNTING DRILL: Drawing functions:

CHECK PROJECT: and M2 upon an error line or a marker explains what's wrong. Help: Catchword: „Check project".

INSERT A LOGO: Docu: Menu Actions/ Load bitmap into a rectangle. Copper: Menu File/Input-Output formats/File transfer/Documentation/Bitmap as a Symbol/Package. Treat like package.

CREATE MANUFACTURING DATA:

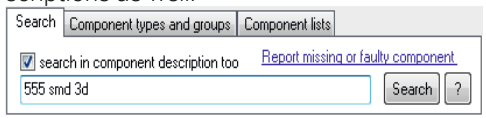
PRINT: Fade out unused layers. Open menu: File/Print... and tick Sharp black/white in the print dialog.

GERBER DATA: The gerbers being created can be read by RS274D and RS274X machines. Also see File/Input/Output formats/Production/(X-)Gerber and Drill Output (PCB Out). Gerbers also can be imported.

ISOLATION MILLING: File/Input/Output formats/Production/Isolation milling. Formats besides HPGL: X-Gerber, NCP (Isel), CNC ISO 6983 DIN 66025 (G-Code, Mach3), PCB (CharlyRobot), CBT (Colinbus), Excellon

DESIGN A FRONTPANEL: Start File/Start-Assistant, then New Project: "Alu-Front panel". Or Menu Actions/Front panel, then follow the menu entries. Toolbar visible: see menu View/Toolbars. Front panel layers in most cases: 28, 29, 30 and 31.

USE THE QUICK FIND AND SEARCH RESULTS OF THE COMPONENT DATA BASE: Open component browser by [Ins] and enter to the search line: Enter e.g. 555 smd 3d. Now TARGET searches the component names. If the box is ticked, descriptions as well.



TARGET recognizes the word smd standing for a mounting technique. Further categories are e.g. spice and 3D for the existance of respective models, texas e.g. for a producer, dip8 for a package form. Speech marks switch off this behaviour. Upper or lower case does not play a role in the search.

COMPONENT BROWSER CONTEXT MENU: M2 on the name of a component listed in the search results of the component browser offers additional options.

FIND THE COMPONENT DATA BASE: Windows Start button, Programs, TARGET 3001! Vxx <edition>, components

CRATE A MOLDED INTERCONNECT DEVICE (MID): Draw a schematic first. Now import a 3D body by schematic MENU ACTIONS/PREPARE MID... Place each of the packages to the 3D body by the use of this icon (left). Turn it by highlighting it and pressing keyboard key [t] for turn. Use key [d] for displacement.

CALCULATE THE AIR WIRES resulting from the schematic by using this icon. The SNAP-ON-PAD and SNAP-ON-TRACK functions help hitting the connections well. DRAW A TRACK by the use of this icon (left). Every click makes a vertex which you later easily can move by this „DRAG A CORNER" icon (left) or keyboard key [g]. Also key [e] for edit works.

DRC THE DESIGN RULE CHECK [DRC] approves to have all packages placed, and all connections made properly. It also checks whether all pads have contact to the bottom of the body and whether all widths and distances are kept according to the constraints of the schematic. [BOM] creates the bill of material together with angles and orientations of all parts in use. [DSP] gives information for dispenser control regarding soldering paste etc. [STEP] creates the project data for production or for demanding a quote.

PUSH&SHOVE: Move existing traces during routing (select icon in Sidebar to add to the bend modes or use [o] for options).

REVERSE ENGINEERING: Redesign/trace an existing circuit by placing a ciruit photo under it (menu Actions/Reverse Engineering/Set Image to Board). Automatic schematic derivation possible, including distribution of related assemblies to user-defined schematic pages.

TARGET 3001!
easy to use.
all in one.

If you get stuck at a certain point please check our Website or find a solution using the Documentation:

HTTPS://IBFRIEDRICH.COM/
HTTPS://DE.TARGET3001.COM/

You also might call us during German daytime:
+49 (0) 6659 919 444

USA residents please call:
+1 530 763-2299

You also might send us an eMail with your project file attached.
TARGET@IBFRIEDRICH.COM