

What's New in AutoTRAX Design Express (DEX)

20th. November, 2011

- Pressing Footprint placement vector now a wide read line with arrow head.

19th. November, 2011

- Pressing space or 'R' key when moving selected rotates them by 90 degrees. Pressing the 'M' or 'v' key mirrors about vertical center, pressing 'H' key mirrors about horizontal center.
- Selecting a track/net highlights all pads connected to it and dims the others.
- Interactive layout now works. Again pressing space key rotates footprints.
- As you move footprints, a read bar shows displacement for perfect part placement.

18th. November, 2011

- Added context menu for schematic symbols and footprints to show data sheet for the part. Note: data sheet url must be added to be part to see the menu item.
- Properties panels for footprints and schematic symbols now contains data sheet and part description controls.
- Schematic router improved. Far less unexpected automatic routing of wires!

16th. November, 2011

- Improved Smart Pan.
- Improved auto-dimming for PCBs.

14th. November, 2011

- New selection manipulator.
- Improved and optimized Gerber output.
- Some UI tweaks.

13th. November, 2011

- Speed improvements and reduced memory usage.
- Very significant redraw speed improvements on large PCB of about 2.5 times.

11th. November, 2011

- Improved autolayout, and place footprints around border speed.

10th. November, 2011

- Selecting a pad or footprint now turns on auto-dimming and highlights selected.

9th. November, 2011

- Selecting a terminal on a schematic will select the corresponding pad in the PCB.
- Selecting a pad on a pcb will select the corresponding symbol terminal(s) on all schematics.
- Part Builder now works in projects. This lets you change a footprint by selecting either the footprint or the footprint's symbol.

5th. November, 2011

- Selecting a footprint on a pcb will select the corresponding symbols(s) on all schematics.

- Selecting a symbol on a schematic will select the corresponding footprint on the pcb.
- Selecting a track on a pcb will select the corresponding node on all schematics.
- Selecting a wire on a schematic will select the corresponding track on the pcb.

These are great to see corresponding nets/nodes, symbols/footprints.

To see this in action you will need to have a schematic and pcb viewports, ideally tiled (see panels menu) or either the pcb or schematic floating. (double click on it's tab).

You can change the settings in the Edit/Select tab – click on the small arrow bottom left.

4th. November, 2011

- When manual routing, all fully un-routed nets are hidden. This makes routing far easier.
- Added command to symbol terminal context menu to convert it's connection to inter-wire connections. Right click on the symbol terminal...

3rd. November, 2011

- Printing, like plotting, can be scaled.
- Added command to symbols context menu to convert all it's connections to inter-wire connections. Right click on the symbol...

2nd. November, 2011

- Nets can now be delete in PCBs. Associated nodes in schematics are deleted.
- Adding a net to a PCB will add a wire connecting associated symbol(s) in schematics. Currently only works if symbols are in the same schematic. Will change it to work for symbols on different schematics.
- About dialog now has button to show the settings directory.
- Added 2 menu items to the Node ID context menu (right mouse button down over Node ID) to auto position selected node ID or auto position all node IDs.
- Double click on the following buttons in the ribbon view menu applies to all sheets/viewports:
 - View Rulers
 - View Grid
 - View Guides
 - View Node Names
 - View Grid Reference
 - View Title Block
 - View Page

30th. October, 2011

- Added command to rotate selected by defined amount. (Edit/Rotate menu).
- Added command to scale up/down selected by defined amount. (Edit/Scale menu).
- Added new commands for add new sub-system and add existing sub-system.
- Tabbed pages now have icons.
- Symbol terminal names and pin numbers can now be move individually.
- Holding now the control key switches picking to sub-picking. Releasing the key return to normal picking.

28th. October, 2011

- Added request new part button to Parts menu.

26th. October, 2011

- Added short-cuts manager to Panels/Menu

25th. October, 2011

- New option for Electra router – see router settings popdown.
- Added plot button to the application menu. – plot your footprints...

23rd. October, 2011

- New capture for Symbol Builder.
- Adding no connects to symbol terminals now works.

20th. October, 2011

- Group command now adds selected objects direct to symbols and does not create a group. The symbol now contains the grouped item. Use this to customize parts in place on schematics.

18th. October, 2011

- Speed up editing text types with properties panel.
- General speed up of the whole application

14th. October, 2011

- Added application settings to set the default color for PCBs and schematic. See the settings panel.

13th. October, 2011

- Added global applications units setting to settings panel. You can set default units for new schematics and pcbs.
- Settings panel now show values in user defined units. Previously it was restricted to inches.

12th. October, 2011

- Classic menu improved and brought up to speed with the ribbon menu.

8th. October, 2011

- Gerber New optimization added for tracks which removes excess vias and tracks when the vias overlap.

7th. October, 2011

- Gerber output is now true type fonts. You can optionally use stick fonts.
- Gerber output text now more accurate. This gives you true WYSIWIG if you use the True-type fonts.
- Symbol terminal editor grids now have resizable columns that remember their widths.
- Bug fixes for Symbol terminal editor grids.
- Text positioning is now more accurate. This gives you better quality artwork for schematics and PCBs.
- Fix to quad pack pin numbering.
- Background color and sheet color restored.

- Improved text overstriking interface.
- Status bar's zoom slider now works.
- Improved graphics with very small text.

1st. October, 2011

- Fixes to solder mask for gerber output.
- Add tented via option to gerber settings.
- Fixed bugs with gerber output parameters being ignored.

30th. September, 2011

- Improved gerber output for rounded and elliptical pads.
- DEX startup improved. You will no longer see panels/menus drawn in patches. Now you see the splash screen and then the main window rapidly appears after a short time delay while DEX initializes.

28th. September, 2011

- New Copper fill now draws correct, was orange border.
- Solder mask output improved.
- Notes now draw correct on copper layers.
- Added browse buttons to libraries panel for datasheet and supplier.
- Added check box to layers panel to view pads in yellow, or not.
- Plotting improved.
- Pin description column added to terminal editor.

28th. September, 2011

- New Part Wizard not saves and restores splitter positions for the symbol and footprint wizards.
- Terminal edit grid now instantly update viewports when terminal details are changed.
- Fix in part wizard where supplier details were not saved.

26th. September, 2011

- New checklist panel added. This lets you check your schematics.
- Add 'U' or 'u' for undo last capture and 'R' or 'r' for redo last capture when using the part builder to capture text from say Adobe PDF files.
- Ruler visibility now saved and restored per viewport.

20th. September, 2011

- New download latest dialog. Allows you to go direct to web site.
- Bug fix for Electra router.
- Panel reset only affects current workspace.

19th. September, 2011

- Part Wizard panel renamed PartBuilder to avoid confusion with the Part Wizard.
- Part Builder panel now shows symbol and footprint editors in a tabbed view.
- Bug fix for adding wires in schematics, cursor color now correct.
- Bug fixes/ improvements for workspaces.
- Panel visibility algorithm changed. Should eliminate panel show failing.

18th. September, 2011

- DEX now has workspace buttons in the Panels tab of the ribbon menu. This allows you to quickly switch between 3 workspaces layouts. *This allows you to*

save 3 separate workspaces for projects, part editing and artwork editing. You can renumber the buttons by click on the small captions button at the bottom right of the workspace buttons. The workspaces are automatically set when you select them. No need to manually save. I have kept it to 3 buttons for now.

16th. September, 2011

- Parts library now compressed.
- Added button to File popup menu to decompress individual files.

14th. September, 2011

- Files save compressed by default.
- Fixes for 3D objects on the bottom side of a PCB.
- Ehen sending parts and projects, they are always attached as compressed files.

13th. September, 2011

- Fixed bug when plotting, scale would be incorrect if preview zoomed in/out.

12th. September, 2011

- Added improved tooltips when adding new entities.
- Dynamic input improved when adding new objects.

24th. August, 2011

- Added Send project via email to the application menu.
- Improved internal routing for single and double sided boards.

22nd. August, 2011

- Added view from bottom for PCBs. It's in the PCB menu, far right.
- Improved the Router popup dialog. Click on the bottom at the bottom right of the menu.
- Added start manual routing on top checkbox to the Router popup dialog. This allows you to select which layer DEX will try to use when it starts routing.

20th. August, 2011

- Print and Plot preview now greatly improved.
- Print preview now show list of sheets to print.
- Plot in Tools tab for PCBs now plots negative/reverse and scaled.

18th. August, 2011

- PCB Plot now plots correctly to scale. Drill targets on pads and PCB border with alignment targets added.

17th. August, 2011

- Gerber import to PCB. You can set color for the imported entities. Entities imported into a group.
- Added a new background layer to the PCB.

13th. August, 2011

- Improved buses. Better optimization and editing.

11th. August, 2011

- Added optional drill centers for PCB plots. Set using Cam Tab corner button.

- Added right context menu item to rulers to delete all guides.

10th. August, 2011

- Added command to context menu of library panel to set the library location.
- Improved synchronization of node names (also fixed some bugs with copper pour).
- Selected un-routed track segments now drawn as cyan.

9th. August, 2011

- Added teardrops to PCB tracks. This is optional.
- Teardrops can be restricted to pads.
- Teardrops can be restricted to pads and round/elliptical pads.
- Teardrops can be turned on/off.
- Fixed bug where grounds connected to parts would not connect all terminals together.
- Fixed several other node-net synchronization issues.

8th. August, 2011

- Added Elliptical pads.
- Added optional teardrop endpoints for PCB tracks.
- Added snap to grid keyboard shortcut 's' or 'S'.
- Added snap to objects keyboard shortcut 'o' or 'O'.
- Added snap to guides keyboard shortcut 'g' or 'G'.

7th. August, 2011

- Extended Autorouter setup dialog to include Electra temporary directory and auto-delete Electra files.

5th. August, 2011

- Improved manipulators for moving small objects.

4th. August, 2011

- Added show/hide viewport tooltips button to the status bar.
- Add button to turn on/off smart pan to the status bar.
- Improved moving of text objects that are small. Previously they might scale instead of move.

3rd. August, 2011

- Optimized linear beziers for PCBs etc.
- Fix to auto-layout. Pre-rotated footprints were placed incorrectly.

2nd. August, 2011

- Double click on edge of PCB border, copper pours, PCB cut-outs will add a vertex and put you in edit mode.
- Dragging ends of curves and polylines together will set them as closed.

31st. July, 2011

- Dimension text now reversible from properties box or right context menu.
- Only projects are automatically reloaded on start. Parts are not.
- Grid now can be set to the snap grid. (was always as units). See grid settings popup.

- Added grid settings popup to the View/Snaps Per Grid button group. Click on the small arrow at the bottom left.
- Bug fix when adding lines, start cursor was invisible (PCBs only)
- By default PCB grid is set to snap spacing. Schematics are set to units.

30th. July, 2011

- New layout for most properties dialogs; space optimized.
- Number edit boxes improved, e.g. 1.99999999 displays as 2
- New Spice Probe (see Simulation ribbon tab).

19th. July, 2011

- Improved manual wire routing on schematics and node optimization.

17th. July, 2011

- Added no footprint button for symbols. Used for purely logical symbols like ground.
- Added Wires properties popup for Wires menu.

16th. July, 2011

- Port symbol replaced with round symbol.
- Bug fixes.

13th. July, 2011

- Ports and inter-wire connectors can now contain multiple lines.
- Improved sub-systems.
- Bug fixes.

11th. July, 2011

- Sub-circuit symbols and ports now connect together on PCB.
- Added sheet preview at the base of the Project Panel. Double clicking on sheet will display a viewport containing the sheet. Previously a single click did this.

10th. July, 2011

- Sub-circuit symbols and ports.
- Loads of bug fixes.

6th. July, 2011

- Fix to moving Node Ids. (schematics).
- Margin added to interface connector text.
- Added Sub-systems.
- Fix for restoring cascaded/tiled viewports.
- Dragging parts from the library – smart panning will occur for parts with only 1 symbol.

28th. June, 2011

- Improvement in Project Manager panel. Bug fix for adding child sheets and deleting sheets.

30th. June, 2011

- Classic menu interface improved.

28th. June, 2011

- For the horizontal and vertical ruler and the origin, Drag to add a guide. Ctrl+drag to set the origin. Double click to reset the origin.
- Increased tooltip display time from 5 to 10 seconds.
- Graph grid major lines drawn correct when the origin is offset.
- Improved graph grid for PCBs.

25th. June, 2011

- Improvements to standard router.

27th. June, 2011

- Added scroll bars to 2D viewports. By default these are not displayed. You can turn them on/off using the View/Scroll Bars menu item or right click on a viewport and check/uncheck the view scroll bars item. Visibility persists between sessions.

24th. June, 2011

- Fix for visible of routed layers on PCBs. Sometimes track would not appear when turning on/off layers. Tracks were there, but just not shown on the screen. Gerber output was ok.
- Bug fix for selected track segments sometimes displaying multiple selections.

20th. June, 2011

- Project archive directory changed from XXX to XXX.Backup where XXX is the project name. Copy or rename your old XXX directory. This was done to prevent confusion when exporting to PDF as the save dialog tried to save to the backup directory
- !Several bug fixes.

13th. June, 2011

- Fix for spiral inductors and antennae.

12th. June, 2011

- New Text editor.

10th. June, 2011

- Release 1.3.00 – New UI libraries.

6th. June, 2011

- Floating viewport (by dragging on their tabs or double clicking on their tabs) now change the contents of the Ribbon menu to suit the viewport type.

30th. May, 2011

- Added view 3D, PCB and schematic commands to the viewport context menu.
- Added view color bar commands to the viewport context menu.
- Stopped display of multiple viewports with 3D.
- Add help button now link to correct help topics.

28th. May, 2011

- Added group and ungroup commands to the viewport context menu.
- Added pick and sub-pick commands to the viewport context menu.
- Added align and distribute pick commands to the viewport context menu.
- Fixed bug in curves that were incorrectly drawn.

- Several other bug fixes.

26th. May, 2011

- You can now set the library location in the settings panel.
- Various bug fixes.

26th. May, 2011

- Fix for rotated footprints routing in Electra, pads were in the wrong position in the Electra file.
- Fix for embedded rotated objects in groups. Group did not rotate correctly.

24th. May, 2011

- Parts bin changed to internally keep the parts in their own container.

23rd. May, 2011

- Improved XML read/writer to reading/saving DEX XML files (Linq compatible).

23rd. May, 2011

- Added angled guide lines. Similar to point/vertical and horizontal guides.

20th. May, 2011

- I have completed the Part Bin panel.

For demo video see <http://www.kov.com/Tutorials/?PartsBin>

You can add parts to the parts bin by:

1. Dragging the parts file from the Windows Explorer (or similar application) onto the Parts Bin or
2. Dragging the part from the Parts Library panel onto the part bin or
3. Right clicking on a part in the Parts Library panel and selecting 'Add to Parts Bin' or
4. Right click over a part on a schematic and select 'Add to Parts Bin'.

To add a part from the parts bin to a schematic, drag it from the parts bin onto the schematic.

TIP. When first setting up a design you could drag all the parts you wish to use onto the parts bin. Then select the schematic page you wish to place the part then drag it onto the schematic from the Parts Bin.

19th. May, 2011

- Fixed problem where it was possible to add transparent lines etc. These would not be visible. They are now forced to the default pen/fill colors.

18th. May, 2011

- Added Time Scheduler/Calendar application. See Tools menu.

16th. May, 2011

- New command to re-snap all schematic electrical items to the grid. This is in the Edit menu. This is useful in tidying up designs.
- AutoTRAX EDA import fixed for error importing Footprint Values and symbols that had been transformed.

11th. May, 2011

- Settings and settings panels improved.
- Polyline/polygon optimizes after edges removing redundant intermediate vertices in parallel edges (including inclined).
- Fixed moving polyline/polygon edge problem.
- Moving curve segment snaps to grid.
- Improved installer.

30th. April, 2011

- Double clicking on line converts to polyline and inserts a new vertex which you move with the mouse.
- Double clicking on polyline inserts a new vertex which you move with the mouse.
- Double clicking on curve inserts a new vertex which you move with the mouse.
- Dimension improved.
- Context menus. 59 new context menus for each selected object. This really helps productivity.
- Polyline/polygon edit improved, includes convert to curve.
- Curve edit improved includes convert to polyline/polygon.
- Loads of bug fixes.

27th. April, 2011

- Added rotate/mirror commands to the viewport context menu.
- Dimension can change fonts.
- Text in dimensions can be positioned by dragging it. Dimension lines remain fixed.

27th. March, 2011

- Greatly Print preview now show multiple pages. Watermark, orientation and margins saved between sessions.

24th. March, 2011

- DEX now does not display dialog stating checking for updates. This is now optionally done in the background. DEX starts up faster.

27th. February, 2011

- Greatly improved manual routing for tracks that connect to multiple pads.

22nd. February, 2011

- Moving/editing entities in the viewport by dragging with the mouse displays parameters in tooltip.
- Manual routing improved. Editing tracks now optionally snaps to grid. Previously moving segments did not.

15th. February, 2011

- Added Electra autorouter to the installer.
- Fixed page settings for schematics (showed PCB settings).

10th. February, 2011

- Fixed problem with Spice Analysis Graph not sizing correctly.
- Fixed frequency input for sin wave generator.

21st. January, 2011

- Added PDF export.

27th. December, 2010

- Added output to CNC. See tools menu. This will generate 2 CNC files ending in Top and Bottom for both sides of a PCB.

5th. December, 2010

- Bug fix for autorouter not running at all.
- Export to Scaleable Vector Graphics – Tools tab.

29th. November, 2010

- Automatic update now disabled. You can do it from the Home menu tab next to the license button.
- Optional numeric input of parameters while adding lines and polylines.

27th. November, 2010

- Fixed sign of electrolytic capacitor.

23d. November, 2010

- Fixed redraw bug. Multiple useless redraws.

20th. November, 2010

- You can now set a page scale for individual sheet. See ribbon menu View/Page/corner drop down button. The page scale lets an A sheet contain a design of size save 1100" x 850", this would be a scale of 64. Now you can even do your floor plans with DEX!

19th. November, 2010

- Dimensions can now be added linear or aligned with an offset.
- Added button on the left of the status bar to turn on/off dynamic input.

18th. November, 2010

- Added overstrike ability to symbol terminals.

16th. November, 2010

- Fixed bug with text editor. It was set to right to left.
- Dimensions can now be added with an offset.

15th. November, 2010

- DEX now optionally archives all changes to a design to a directory with the same name as the design. You can change this via the settings panel in Project/General/Automatic Archive.

14th. November, 2010

- Starup improved. Less flashing dialogs etc.

- Part finder, selected part not flashes at 5 times per seconds. Compared to the past when it flashed > 100X for fast graphics!

13th. November, 2010

- Setting signal name on tracks updates names on nodes in the schematics.
- Create new part shows part wizard.

12th. November, 2010

- Hide/Show color panel on bottom of application window works.
- Adding lines, polylines, arc, rectangles, ellipses, noteboxes, symbol borders now display interactive dimensioning during construction.
- Adding rectangular and elliptical pcb borders and cutouts now display interactive dimensioning during construction.
- Adding guides shows position tips during addition.
- Adding pcb pads shows position tips during addition.
- Bug where PCB border would not display in 3D is fixed.
- Note boxes draw current on PCBs.
- PCB properties dialog now sets pcb color correctly.

7th. November, 2010

- All languages now work.
- Panel contents fill available control space.
- Speed up part symbol placement. Dialog does not appear.

6th. November, 2010

- Removed floating 3D viewport command. Double click on a viewport tab will float it. Also removed extra main window command for same reason.

26th. October, 2010

- NC drill files format changed to 0.1 thou resolution. (D decimal places).

17th. October, 2010

- Fixed bug where manual routing would not swap layer when space bar pressed.

15th. October, 2010

- Fix for NC drill files where part is rotated.

9th. September, 2010

- User group and Mantis button added to Ribbon Home menu.

22nd. August, 2010

- Library files are not overwritten on update.

14th. August, 2010

- DEX now uses .Net 4 (previously it used .Net 3.5 SP1).

8th. August, 2010

- Library parts now installed with the installer.

24th. July, 2010

- Printing now will print full page or all objects if show page is off.
- Moving footprint and symbol values and references now no longer snaps to grid.

18th. July, 2010

- Faster startup for ribbon menu version.

17th. July, 2010

- All properties dialog layout adjust for multilingual text.

13th. July, 2010

- Improved Spanish translation thanks to Ramon Barber. Work still in progress.

11th. July, 2010

- Added Japanese and Ukrainian

4th. July, 2010

- Added support for the following languages in the UI (Not the help file as yet):
 - English
 - Chinese
 - Czech
 - Danish
 - Dutch
 - Finnish
 - French
 - German
 - Hungarian
 - Italian
 - Norwegian
 - Polish
 - Portuguese
 - Russian
 - Slovenian
 - Spanish

This is just a first pass and they are machine translations (like Google/Yahoo). Will improve over the coming week.

To change the language, use the Application menu in the Ribbon. Click on the round DX icon, the language selection is at the bottom of the menu.

Not all dialogs have been adjusted to take into account varying widths for text. In addition Right to left e.g. Chinese controls are not done. I expect this to be done early this week.

14th. June, 2010

- Node ids did not move correctly when the node was in a group.
- You can turn off automatic wire routing in the sheet settings panels.
- Copy and paste now works with electrical entities such as symbols, wires etc. Does not work on PCB sheets, only schematics.

12th. June, 2010

- What's new now displayed for new builds. (this document).
- Moving symbols and groups containing symbols did not follow the mouse exactly and sometimes groups 'broke up'. Now fixed.

9th. June, 2010

- Adding parts to schematics, custom pads added to parametric footprints would not appear. Now fixed.

21st. May, 2010

- Option to display ribbon menus in Office 2010 style or 2007 style. (Panels Menu).

14th. May, 2010

- Copper pour regions can now be nested.

13th. May, 2010

- Adding copper pour regions did not fill, fixed.
- Adding auto-sized copper pour region to rectangular PCBs not works correctly.

12th. May, 2010

- User interface speed up.
- More parts – see CMOS + TTL

3rd. May, 2010

- Bug fix where extra schematics sheets were created when opening a new main window.

2nd. May, 2010

- New installer for updates. Easier installs less dialogs. All automatic. No residual install dialog.
- New main window button moved to panel menu and added to classic menu.
- Focus problem where main windows would jump in from of others fixed.

1st. May, 2010

- New Part Wizard dialog no longer stick on top.
- Added multiple windows aimed at multiple monitor useage. See the Application menu. Currently only works for ribbon menus.
- Installer will now not overwrite existing Electra installation.

29th. April, 2010

- Footprints can now be placed on the bottom of the PCB as well as the top. Use the footprint's properties dialog in the properties panel.

28nd. April, 2010

- Off-page connectors are noe inter-page connectors.

22nd. April, 2010

- Enabled sub-picking if invisible symbol terminals using window select. (Hold doen left mouse button and drag the mouse over the terminal.
- Improved bus editor properties dialog.

14th. April, 2010

- Added all tutorial videos to local help.

10th. April, 2010

- Added right mouse click context menu to 3D view.
- Bug fix to footprint wizard, library parameters click did not update part.
- Footprint/Symbol text values preserved when changing parameteric part type.

8th. April, 2010

- UI improvements to part wizard. To make your life easier.
- Adding symbol border to part design sheet auto-aligns terminals etc.
- Pick and place output file updated for footprint placement point. Rotation of footprint also output.
- Footprint editor now allows viewing and editing of footprint rotation as well as the footprint placement point

6th. April, 2010

- Added auto-position symbol reference and value button to Symbol Wizard.
- Added buttons to hide/show all symbol terminals to Symbol Wizard.
- Added more parts and artworks. You will see lots more added over the next few weeks.
- Terminals not attached to a symbol border can have their orientation set.
- Added button to line style editor to set width to schematic wire width. This helps in creating artwork in parts editor.
- Copy and paste of groups fixed.

5th. April, 2010

- Added new capture tool to Symbol Wizard side panel.
- Click to capture test from another application.
- Move cursor to application.
- Hold down and drag left button to select text.
- On release, text is used to set terminals.
- Terminals start at the first one and when all terminals are set, capture will end.
- Works well with Adobe PDF files.

3rd. April, 2010

- New library structure. Added several parts and artworks for creating parts.

2nd. April, 2010

- Added new file format '.art' for saving library art to drag and drop. Includes all graphics entities together with pads, symbol borders and symbol terminals. Great to use to quickly build custom parts etc.
- Faster opening of parts in same application windows when possible.
- Bug fix for adding pads with width and height locked.

31st. March, 2010

- Added toolbar buttons and status bar to 3D viewer for viewports.
- Option to view 3D part with semi-transparent board.
- More UI improvements.
- Improved automatic symbol build from footprint.

29th. March, 2010

- Improved rebuild symbol from footprint. Now has option for 1,2 or 4 sided symbol
- Dragging parts onto schematics, pressing the 'space' button rotates 80 degrees. Pressing arrow keys mirrorssymbol about the vertical or horizonntal axis.

27th. March, 2010

- More UI improvements.
- Bug fixes for auto-sizing footprints.

26th. March, 2010

- Parts menu items now all have footprints (for non-virtual symbols) and 3D packages.
- It is now possible to very easily make custom footprints. Add pads and the silkscreen, footprint reference, footprint value and courtyard automatically adjust.
- Preview can now have 3D attributes in PCBs. These include lines, polylines, curves, rectangles, images and ellipses.
- Improved part creation.
- Load of bug fixes and user interface tweaks.

22nd. March, 2010

- Bug fixes and user interface improvements.

18th. March, 2010

- Several small but significant UI improvements to setting the color of entities.

17th. March, 2010

- Added 4 color selector styles. Color swatch, RGB sliders, HSL sliders and HSL color wheel. The color selector now allows transparency.

16th. March, 2010

- Preview of new line and fill color editor controls in the properties panel. Color Ring control not fully functional yet.

8th. March, 2010

- DEX automatically checks for updates.
- Added show targets checkbox to PCB border properties dialog.
- Added set shape button to PCB border for copper pour properties dialog.
- Added set origin to PCB border lower left to ruler menus and the PCB properties dialog.

8th. March, 2010

- New install program.
- Added thermal relief checkbox to copper pour properties dialog. If uncheck, pads and vias connected to the copper pour region will merge as single hole else thermal spoke will be added.

6th. March, 2010

- Reduced target size for PCB.
- PCB targets and sheet visible initially disabled for PCBs.
- Gerber Viewer print can now print black on white. (menu option).
- Improved print layout to maximize paper usage.



5th. March, 2010

- Pads connect to copper pours via thermal relief if no track with same signal name as the copper pour connects to the pad on the copper pour layer.
- Tracks and vias with same signal name as the copper pour merge with the copper pour.

- Pads with the same signal name as a copper pour connect via thermal relief unless a track is available to connect them.
- Added print function to Gerber Viewer.
- *4th. March, 2010*
- Added project object hierarchy tab to the Source View panel. Multiple nodes with same signal name now all connected together on the PCB with the same net, even if the separate nodes are not connected together.
- You cannot delete a net/track on a PCB that is associated with a node/wire on a schematic. Deleting a footprint will delete the schematic symbols and all connected wires on the schematic and tracks on the PCB.
- *1st. March, 2010*
- Added project object hierarchy tab to the Source View panel. This allows you to select/view and edit objects by the position in the DEX database.
- Added Hole entity to the DEX database for drill holes only. Views as hole in 3D. Not output to Gerber so no Gerber design rules error for no copper ring. Outputs to drill file. Absolutely no relation to pads or vias. More akin to cutouts but do not appear in Gerber board profile; only appears in NC drill file
- *28th. February, 2010*
- Added Pan, View Contents and Units ribbon buttons to Gerber Viewer. Also added status bar.
- Manual Routing now smart pads when routing near the edge of viewports.
- Added menu link to PDF manual for DEX.
- *27th. February, 2010*
- Added Cam Settings popup to Tools/Cam ribbon menu group.
- Added Gerber Settings to Cam Settings popup. Options to: set holes sizes on Gerber pads and vias, and show/hide targets and plot labels.
- Gerber viewer now displays thermal reliefs/power planes correctly as promised.
- Gerber viewer - when mouse is over a pad/via shape (actually a Gerber flash) the tooltip shows the aperture D number etc.
- ***The Gerber output is now done. (...famous last words!).***
- *26th. February, 2010*
- Gerber viewer now stores files in Program Data directory. This allows .msi installer to work.
- Improved measure distance tool.
- Added measure distance tool to the Gerber viewer.
- Gerber output of power planes implements. Works with the Gerber Viewer(not fully as thermals are drawn incorrectly, will fix this weekend)
- Gerber Viewer now uses Gerber Flash entities.
- Gerber output now has optional holes for pads.
- *23rd. February, 2010*
- Smart panning on highly magnified views fixed. Previously would jump violently.
- Measure distance tool improved.
- Power planes look like they are now full working except for Gerber output. Will fix tomorrow.
- *21st. February, 2010*
- Added cursor world position display to status bar (Ribbon menu).

- Net/tracks and vias will now draw thermal relief on signal planes (inner layers on boards with 3 or more layer) .Gerber output not done. More work to do on this. Will finish this week.
- *21st. February, 2010*
- Improved visual quality of copper filled areas in the Gerber Viewer.
- Improved Gerber output.
- Improved DRC checker.
- *20th. February, 2010*
- Added button to PCB border properties dialog to set origin to the bottom left of the PCB border.
- Fixes for selecting and editing the PCB border.
- You can now set the color of the PCB. Shows in populated and 3D views.
- Generation of Gerber files with large copper pour was VERY slow. Now it's very fast.
- Gerber quality improved to 0.1 thou. - 2.54 microns.
- *19th. February, 2010*
- Design Rule Checker now checks for un-routed track segments and optionally will route and optimize tracks and nets.
- Improved PCB ribbon menu page.
- Bug fixes for manual routing.
- Properties dialog for Track improved. Added un-route buttons and width setting controls for net, track and selected segment.
- Added Show/Hide menu item to origin, and vertical rulers context menus.
- When setting origin from origin, and vertical rulers, origin is displayed for both coordinates.
- *18th. February, 2010*
- New context sensitive ribbon pages for symbols, footprints, schematics and PCBs. These group commands together for each design context. It really makes you more productive.
- Manual routing and auto-dimming works great.
- *17th. February, 2010*
- More improvements on manual PCB routing, vias and track widths.
- *15th. February, 2010*
- Picking objects not works correctly on PCBs. Previously wrong objects were selected. This really affected manual re-routing.
- Work in progress on track width, and manual routing.
- *12th. February, 2010*
- New help file.
- *11th. February, 2010*
- Several bug fixes and code optimization.
- *8th. February, 2010*
- Adding parts now sets footprint reference correctly.

- Renumber parts now works with PCB order. Schematic and Footprint references are set in bands so they are ordered for easier location. You can use this to fix any footprint references that are in error.
- Symbol properties dialog, symbol reference dialog and symbol value properties dialog now update all sibling symbols and the associated footprints.
- Footprint properties dialog, footprint reference dialog and footprint value properties dialog now update all associated symbols as well as the footprint.
- *6th. February, 2010*
- Added Symbol/Footprint finder tabbed control to the World View panel. This lets you quickly locate them. It will display a viewport containing the symbol/footprint and flash it. The World View will also update to show where on the sheet you are.
- Added World View control to the DRC panel.
- *5th. February, 2010*
- Design Rule Checker panel. Completed. Click on error and viewport will display error at its center.
- Design Rule Checker can now automatically fix PCB design errors (pads, vias, holes and track minimum size).
- *4th. February, 2010*
- New Design Rule Checker panel added. First set of design rules done. Shortly will add lists of errors in the panel so you can easily highlight them. The Design Rule Checker (DRC) is an essential part in ensuring Gerber output will be ok.
- Gerber output improved. See Gerber Viewer.
- *3rd. February, 2010*
- Gerber export to top and bottom resist as well as board profile with cutouts implemented.
- Gerber viewer improved.
- *31st. January, 2010*
- Import AutoTRAX EDA files placed SMT pads on silkscreen layer. Now fixed.
- PCB Track Optimizer improved. Previously it would remove vertices incorrectly.
- New web site completed. <http://www.kov.com>
- Improved Gerber output. Now works with transformed objects.
- Fix to gerber viewer for negative coordinates.
- Add Wiki and Blog tabs to Forums dialog.
- *25th. January, 2010*
- Forum moved.
- *20th. January, 2010*
- Axial parts now appear correct in 3D.
- Web site moved and alternate created <http://www.autotrax.net>
- *12th. January, 2010*
- Fixed bug in importing old EDA library. Re-run it to fix if needed.
- Optional outer guard ring on spiral antenna.
- *10th. January, 2010*
- Changed Mantis bug reporter to work with uploaded designs and also to show uploaded designs in the viewport on the left.

- *9th. January, 2010*
- User Import of old AutoTRAX EDA library now adds associated footprints to parts.
- *8rd. January, 2010*
- User group dialog now combines the Yahoo! user group and the User's Forum.
- Added new Mantis rich-client dialog for viewing and reporting bugs. This is far easier than using a browser. You need to have a valid support license to use it.
- *3rd. January, 2010*
- Created new user group. User group button on menu now opens up a compact browser without all those adverts!
- *2nd. January, 2010*
- Accounts dialog updated.
- Speed bug fixed.
- DEX now compatible with new web services.
- *27th.December, 2009*
- Added Spiral Arial parametric part.
- Added Spiral Inductor parametric part.
- *21st. December, 2009*
- Title Block shows page size.
- Title Block now displays revision as Prefix:SaveNumber. SaveNumber is the number of saves. Prefix is set in title block properties dialog.
- Title Block shows sheet name,
- Title Block shows design wide document number.
- Help file additions.
- *19th. December, 2009*
- **DEX now tested and works on XP, Vista, and Windows 7 including both 32 and 64 bit operating systems.**
- *16th. December, 2009*
- Added shared community library.
 1. Display the library panel.
 2. Add your parts to 'My Shared Parts'
 3. Click on  to uploaded your shared parts. That's it!
- When you start DEX it automatically updates the community library folder on your machine.
- To manually download latest Community Parts click on .
- *8th. December, 2009*
- Part Wizard now keeps symbol terminal names when changing footprint parametric model type or model parameters.
- Improved silkscreen images for parametric footprints.
- *7th. December, 2009*

- Improvements to schematic router. Wires to wire junctions previously may not route.
- *6th. December, 2009*
- Improvements to symbol wizard: Include select terminal in grid selects terminal on sheet. Also added extra controls to see more parameters for the part terminals.
- *5th. December, 2009*
- Improved part wizard panels and parametric parts. Added tab with lists of predefined parts. Added some more parts.
- *2nd. December, 2009*
- Improved schematic routing.
- Moving terminals in schematics now keeps them to the symbol border and reroutes.
- Moving parts, reroutes.
- Resizing symbol borders reroutes.
- *30th. November, 2009*
- Added command to show/Hide footprint courtyards in view menu.
- Added command to show/Hide footprint placement points in view menu.
- *28th. November, 2009*
- Improved graphics for rectangles, symbol border etc. using adaptive anti-aliasing.
- Improved import of AutoTRAX EDA schematics and PCBs. Part symbols now are matched to part pads. Multiple sheets imported. Viewports created for each sheet.
- Selecting footprint pin numbers for symbol pins improved.
- Paste works correctly and other bug fixes.
- *25th. November, 2009*
- Loads of UI improvements and fixes. Added PCB header footprint wizard and radial component wizard.
- *22nd. November, 2009*
- More improvements for graphics speed.
- *21st. November, 2009*
- Gerber output improved.
- New standalone installer.
- Improved graphics speed when adding lines, wires, parts, etc.
- *20th. November, 2009*
- Transfer old parts now prompts for source and destination.
- Gerber output done, Gerber viewer done except for output parameters.
- *19th. November, 2009*
- Added command in Tools tab to transfer old AutoTRAX EDA parts into the DEX library. The files are converted and put in group My Parts. Once move you can move them using the DEX libraries panel.
- Several usability improvements and bug fixes.
- *18th. November, 2009*

- A lot of improved to parts wizard. (Symbol and footprint).
 - **Things are looking good. I expect DEX to get 'first light' by Thanksgiving.**
 - New library location \program data\Kovac Software\DEX\Library. You old stuff is in \program data\Kovac Software\AutoTRAX\Library; I suggest you delete it.
- *16th. November, 2009*
 - Symbol and Footprint panel merged into 1 part wizard panel. The appropriate wizard displays depending on context.
- *12th. November, 2009*
 - Replaced Guid attributes in XML with long unsigned integers.
- *11th. November, 2009*
 - Rotating parts (without symbol border e.g. resistors) by multiples of 90 degrees works correctly.
 - A few teaks and bug fixes.
 - If you find DEX updates ad there is no date entry, then it means that the update includes only tweaks and unreported bug fixes.
- *9th. November, 2009*
 - Changed selected object panel so that is shows selected sheet properties is no object is selected.
 - Added new tab to sheet properties to allow you to add notes for each sheet in the design.
- *8th. November, 2009*
 - Improvements to footprint and symbol wizards. 3D packages improved.
- *7th. November, 2009*
 - Reload last project fixed.
 - Missing library for copper pour added.
 - First pass of footprint modeller done.
- *3rd. November, 2009*
 - Working on part creation and modifying the part/footprint data. You may see parts with a pin name of "?" this means there is no pad associated with it. This is work in progress and should be fixed in the next few days.
- *28th. October, 2009*
 - Copper pour speed increase for copper pours over sub-sections of the PCB.
 - Progress dialog box appears during copper pour fill. User can cancel.
- *27th. October, 2009*
 - Curve creation and drawing improved.
 - Added tension control to curves.
 - Adding curved pcb cut-out now works.
 - Move curve control points improved.
 - Current layer automatically turned on when adding objects.
 - If pcb cutouts extended outside the pcb border, tracks were drawn incorrectly in 3D. Now fixed.
 - Drawing style changed for keep-out regions to semitransparent bricks.
 - New icons for copper pour, keep-out and cut-out menu buttons.

- *26th. October, 2009*
- Added option in Sheet Settings to view TPH pads as solid yellow or yellow outlined.
- Pasted objects now offset by grid spacing, so with copy/paste the pasted objects are separated from the originals.
- *25th. October, 2009*
- Bug in copper pour for rotated footprints fixed.
- Unfilled copper pour now outlines as thick dashed lines rather than thin lines.
- Copper pour rectangles can now have rounded corners.
- Copper pours can be scaled and rotated.
- 3D view updates improved when changing PCB. 3D view now more in-sync.
- *24th. October, 2009*
- Install website moved.
- *23rd. October, 2009*
- PCB cutouts in 3D implemented.
- PCB cutouts on PCB Sheets show through to grid in populated view.
- *22nd. October, 2009*
- Improved Copper pour.
- Bug fixes.
- *18th. October, 2009*
- Pasting objects leaves them all selected. Make moving easier.
- Color bar left and middle button reversed to be same as AutoTRAX EDA.
- Color bar now enabled for lines.
- If selected objects have only fill or line color then both left and right buttons act the same in the color bar.
- *17th. October, 2009*
- Added Electra router capabilities. You will need Electra installed and you should select Panels/Settings/Project then enable Electra.
- Pads now drawn with yellow outline rather than solid yellow. This helps show layer.
- Wires to pads with holes displayed now draw correctly.
- Via holes drawn.
- *9th. October, 2009*
- Improved schematic auto-router.
- *3rd. October, 2009*
- AutoTRAX EDA buses import correctly
- Import of multi-line text in AutoTRAX EDA fixed.
- *2nd. October, 2009*
- More improvements in reading of AutoTRAX EDA files.
- Improved switching between menu style.
- This update will reset your login.
- *1st. October, 2009*

- Improved reading of AutoTRAX EDA files, in particular PCBs. More improvements to follow.

- *29th. September, 2009*

- Improved management of toolbar persistence.
- Graph grid not as pronounced.

- *28th. September, 2009*

- Classic menu bars improved. Only relevant menu and toolbar items are visible depending on sheet type.

- *27th. September, 2009*

- Classic menus now automatically set for different sheet types.
- Panels automatically set for different sheet types.

This means you can customize the toolbars and panels for each sheet type and the customization is remembered.

The different sheet types are:

1. None. (Used when no sheet is displayed)
2. Schematic.
3. Schematic Part.
4. Footprint.
5. PCB.
6. Text Sheet.
7. 3D view.

**Settings now saved in C:\Users\XXXX\AppData\Local\Kovac Software\DEX.
Where XXX is your account name.**

- *18th. September, 2009*

- Classic menu bars now mostly implemented. 216 items out of 386 items implemented plus support, e.g. Customization and persistence.
 1. To switch from Ribbon menu to classic click on Menu/Classic button in the Panels ribbon category.
 2. To switch from classic to ribbon click on Panels/Switch to ribbon Menu.

- *13th. September, 2009*

- A lot of internal code optimization. Tedious work, but well worth it.

- *4th. September, 2009*

- Color bar no longer has title bar. This saves a lot of screen space. It can only be docked to the bottom of the application. You cannot move it. You can hide it from the Panels menu.

- *3rd. September, 2009*

- Popup menu added to Graphics in Page menu. Lets you set page size, edit the grid reference parameters and set the title block.
- Fix to schematic wire auto-router for imperial page sizes.

- *2nd. September, 2009*

- Popup menu added to Graphics in Add ribbon menu. Selects default.
- All undo/redo states named. Can be selected from popup in Undo/Redo ribbon menu.

29th. August, 2009

- Main window shows correct size on first show.
- Main window restored correctly on multiple monitors.

28th. August, 2009

- Added pen size for graphics (rectangles, ellipses, lines etc.) menu items in Add ribbon menu.
- Improved some icons on dark backgrounds.

28th. August, 2009

- Added pad size (width and height) menu items in Add ribbon menu.
- Added pen and fill color for graphics (rectangles, ellipses, lines etc.) menu items in Add ribbon menu.

23rd. August, 2009

- Bug fixes to adding parts like 7400 series TTL.

21st. August, 2009

- UI improvements to add line,ellipse,dimension,image and note.
- Dimensions can now have:
 - Text showing length.
 - Prefix + Text showing length.
 - Prefix only. Useful for referencing dimension elsewhere.
 - No text or prefix.

22nd. August, 2009

- Cut, delete, copy and paste working.

16th. August, 2009

- 2 new commands in the Edit ribbon menu for schematics. Re-wire selected and re-wire all. These rewire schematic wires. No extra ones are created, and none are deleted.

15th. August, 2009

- Autorouter places footprint on grid.
- Center drawing in schematics moves wires correctly. Were slightly off grid.
- Print preview now remembers whether to print page or contents.
- parts automatically rewired in schematics when rotated by 90/180/270 or mirrored.

12th. August, 2009

- Added floating 3D viewport.

10th. August, 2009

- 3D now fully working.
- 3D menu now renamed View.

8th. August, 2009

- Source view panels now has 2 tabs. First shows xml of the project, the second show the 3D xaml for the project.

2nd. August, 2009

- Curved PCB now correctly drawn in 3D.
- PCB border now pickable.
- Wires and terminals in schematics are now sharper.

1st. August, 2009

- Improved 3D.

26th. July, 2009

- Spice modeller now works on 64 bit Vista and XP.

23rd. July, 2009

- 3D viewports viewpoint mouse movement now working correctly.

19th. July, 2009

- 3D viewports have background and basic viewpoint mouse movement.

15th. July, 2009

- 3D views back. Mouse view settings not available yet.

13th. July, 2009

- DEX now works on 64 bit Vista and XP platforms. Performance problems fixed.
- 3D viewport content disabled. Displays 3D text. Will be reinstated this week.

10th. July, 2009

- Mouse wheel zoom zooms on center of mouse point, not center of viewport.

9th. July, 2009

- Global settings panel and sheet setting panels merged into one panel.
- Grid Reference and Title block editable via Settings panel.

8th. July, 2009

- More help.
- Grid Reference and Title block implemented.
- Editors for Grid Reference and Title block parameter to come later.
- View ribbon menu has buttons for show grid reference, title block and page.

6th. July, 2009

- Partial implementation of help files.

4th. July, 2009

- Autolayout group moved from Route menu to layout menu.
- Implemented 'Prepare for Manufacture'. This creates all gerber files etc and adds them to zip file.

3rd. July, 2009

- Bill of Materials export in Tools/Cam menu implemented.
- Gerber export in Tools/Cam menu implemented.
- Drill file export in Tools/Cam menu implemented.
- Pick and place export in Tools/Cam menu implemented.
- Preview Gerber Tools/Cam menu implemented.
- Gerber export now works with text.

2nd. July, 2009

- Improvements in Gerber viewer.
- Implemented Prepare for Manufacture on tools tab. This '2 click' feature creates a zip file with all files required for board manufacture.

1st. July, 2009

- Export to Spice model from Tools/Export menu.
- NetList Export from Tools/Export menu.
- DXF Export from Tools/Export menu.
- Further improvements to Gerber output and viewer. All should work except to text output.

30th. June, 2009

- Splash Screen not topmost.
- Improvements to gerber output.

29th. June, 2009

- Initial export of ID board - ribbon menu Tools/ Export.
- Gerber Viewer - ribbon menu Tools/CAM View output. This shows gerber output in real time. Text not done yet. This works by exporting all gerber files and reading them back in. Try adding lines to PCB and see them appear in gerber output.

24th. June, 2009

- Bugs fixes to manipulators points in transformed objects.

24th. June, 2009

- Internal changes – changes for export.
- Improved selection transformer, also easier to rotate objects.

22nd. June, 2009

- Internal changes. Added gerber input modules.

21st. June, 2009

- Parts List panel complete.

20th. June, 2009

- Rotate 90/180 and 270 degrees works for all.
- Mirror about vertical/horizontal axis works for all.
- More menu buttons resized and the menu is 99% finalized as for release.
- As DEX is getting closer to release, I have removed some menu buttons for features I will implement them in Release 1.1. These include:
Hyperlinks, rotation snap, standard menu, un-route footprint and track segment, ortho mode, some spice items like oscilloscopes, import and export for DXF, export for SVG and PDF. (you can print to PDF)

19th. June, 2009

- Fixed bug saving designs with locked footprints.
- Group properties now allows you to re-center and resize the group.
- New large buttons on Route menu.
- Locked menu buttons now set on entity status.

- Footprints no longer allow automatic picking of pads.
- Router status move from menu to status bar.
- Bitmaps/images can be dragged from Explorer etc onto sheets.
- Text can now be dragged from outside DEX onto sheets.
- Added scale thumbwheel to note, rectangle, image and group property dialogs.

18th. June, 2009

- Improved library panel. This has separate tabs at the bottom to display the schematic symbol, footprint and 3D package. It also displays a textual description of the part and a separate tab with links to the online datasheet and suppliers web site. You can change the text and the libraries are updated automatically.

See parts in Parts/Cypress Semiconductors/Memory/Async SRAMS

- Improved part wizard.
- Work has commenced on a Gerber viewer. This will read in all gerber files and display them. It will work in real time. This is aimed at ensuring what you get from the board manufacturer is what you want.
- All CAM dialogs are modeless and persist from session to session.
- Improved 3D board view.
- Added icons to some windows UI tabs.
- Some bigger menu icons.

14th. June, 2009

- Added Snap footprints to grid command. (layout menu)
- Added Snap tracks to grid command. (layout menu)
- Snap Parts to grid improved.
- New View PCB button on panels menu.
- Improvements to world view panel.
- Many menu button increased in size.

13th. June, 2009

- All entities can be locked and unlocked. If locked then they cannot be deleted or changed. Use Menu Edit/Lock-Unlock to change.
- World view panel complete.
- Moved Sheets menu group from Edit to Home menu.
- New add text sheet button in Home/Sheets menu.

12th. June, 2009

- Double click on text, note, part values and reference entities brings up text editor.
- Origin now draws in magenta so it's visible on schematics and PCBs.
- 2 new add note buttons in the Add/Shapes menu.
- Add notes and text now prompts for text before allowing you to drag the next into place.
- Spell checker added to text editor and note editor.
- Bug fixes.
- Improvements to reader AutoTRAX EDA files.

10th. June, 2009

- Print preview and print for source view panel.
- Print preview and print netlist panel.
- Print preview and print for Spice analysis panel.

- Newsletter link on Home ribbon.
- Send Feedback on Home ribbon.
- Wishlist on Home ribbon.
- Tip of the day now links to correct file.
- Help/Search.
- Subpicking works correctly. (Edit menu)
- Menu minimize moved from Home to Panels menu.

8th. June, 2009

- Add snap footprints to grid command in layout menu ribbon.
- Add snap tracks to grid command in layout menu ribbon.
- Net/Wire optimizer removes excess vias and excess tracks vertices.

7th. June, 2009

- PCB picking improved.

6th. June, 2009

- Added auto-dimming option in Route Ribbon tab.
- PCB now drawn in black or green if populated view set.
- Manual routing of tracks done.
- Unroute all done.
- Unroute selected nets done.
- Unroute selected tracks done.
- Unroute selected track segment done.

4th. June, 2009

- Add option to enable/disable spell checking as you type. This underlines misspelt words in red. Right click to see menu of spelling suggestions.
- Fix to error reading PCB contents.

3rd. June, 2009

- Added new text sheet type. Use project panel and add text document. These documents are multi-page and can have pictures etc.
- Added spelling checker for text documents.

31st. May, 2009

- Added new Add Note command and associated properties editor.

27th. May, 2009

- Dimension arrows now draw correct.
- Undo/Redo now works ok with dimensions.
- Added more internal tests – it improves reliability and helps fix bugs.
- Work started on manual routing in PCBs.

26th. May, 2009

- PCB via properties dialog done.
- PCB pad properties dialog done.
- PCB track properties dialog done.

25th. May, 2009

- Adding wires now incorporated smart routing. As you drag the end of the wire, left click fixes a route via point. This allow you to do fully automatic wire routing,

routing via your specified points using left clicks or completely manual via left clicks.

24rd. May, 2009

- Library panel viewport of part now readonly.
- Footprint now read from file correctly. Mapping of pads to Symbol terminals was incorrect.
- Tooltips in the viewport only show in pick mode.
- Add pad command completed and pad properties dialog completed.
- Tabbed views now restored in correct order.
- Scaling objects vertically by dragging center scale point works.
- Rotating objects works correctly.
- Dragging unselected objects keeps them unselected.

23rd. May, 2009

- PCB board in 3D generated correctly for rectangular, elliptical and curved boards.
- Part components now displayed in 3D on the PCB.
- What's New dialog only automatically displayed with new versions.

22nd. May, 2009

- Node (wire) junction properties dialog completed.
- Bus junction properties dialog completed.
- Wire junctions now display as solid circles.
- Group properties dialog completed.
- Moving parts in schematics automatically adjusts wire connections. This is now perfect yet and sometimes a bit slow but works well.

21st. May, 2009

- New vastly improved Add Node functionality. Just click on start terminal/bus and click on end terminal/bus and the node is automatically routed from source to target avoiding all obstacles including graphical objects such as text and circles. This will greatly improve your productivity.

See <http://kov.com/Tutorials/AddingNodes>

17th. May, 2009

- Bug fixes and improvements to adding nodes/buses.
- New cursors for add Bus.

16th. May, 2009

- Added text height increment/decrement button to text type object properties dialog.
- Add bus command completed and bus properties dialog completed.
- Add button to Add ribbon menu to bevel wire to bus connections.
- Buses now have movable Ids.

14th. May, 2009

- NodeID properties dialog completed.
- Node properties dialog completed.
- Improved node naming.
- Add offpage command completed and offpage properties dialog completed.
- Symbol terminal properties dialog completed.

13th. May, 2009

- Improvement to schematic nodes.
- Bug fixed in reading nodes.
- Node IDs autoplace on adding wires.

12th. May, 2009

- Part reference properties dialog completed.
- Part value properties dialog completed.
- Part reference and value can be picked and moved independently.

11th. May, 2009

- ColorBar improved and completed.
- Help now works. Only a few topics completed as yet.
- AutoHide/Show on docked panels now a lot faster.

10th. May, 2009

- Missing SPICE dll and exe now installs. This enables SPICE simulation.

9th. May, 2009

- ColorBar panel now works on all sheets.
- Layer Panel now fully functional.
- Added layer controls to all entities which can be assigned to a layer. Layers only applicable to PCB sheets.
- Show pin numbers for pads implemented.
- Show pad holes implemented.

8th. May, 2009

- Pcb Border, PCB Cutouts, Keepouts and Copper pour shapes can now be:
 1. Rectangular.
 2. Elliptical.
 3. Closed Polygon/Curve.
- Commands added to Add menu tab to add the above.
- Add menu optimized.
- Add keepout command completed and keepout properties dialog completed.
- Add pcb cutout command completed and pcb cutout properties dialog completed.
- Add pcb border command completed and pcb border properties dialog completed.
- Add copper pour command completed and copper pour properties dialog completed.
- Add pad command completed and pad properties dialog completed.
- Added Center PCB command in ribbon menu layout tab.

2nd. May, 2009

- Tooltips over entities now shows units correctly.
- Origin, horizontal and vertical ruler context menus completed.
- Add show/hide rulers to viewport context menu.

1st. May, 2009

- Implemented origin. Includes.
 - Show/Hide origin
 - Move origin in viewport.
 - Origin properties dialog.
 - Delete origin in viewport hides origin (Yes, a bit obvious! ☺)

- Right context menus for horizontal, vertical and origin rulers around viewport.
- Hide/Show rulers in Ribbon menu.
- Bug fix on undo/redo.

28th. April, 2009

- Changed font for this page.
- Units property now works e.g. mm/cm/inches. The units are settable per sheet so a schematic sheet can be in inches and the PCB sheet can be in cm.

27th. April, 2009

- Add text commands completed and text properties dialog completed.
- Text quality now improved using antialiasing.
- Add polyline/polygon commands completed and polyline/polygon properties dialog completed.
- Add open and closed curve commands completed and curve properties dialog completed.
- New Ribbon style dialog for the What's New dialog.

25th. April, 2009

- Fixed resource leakage when adding graphics.

24th. April, 2009

- Separated main application into AutoTRAX.dll this is driven by DEX.exe. This will allow for automated system testing.

22nd. April, 2009

- Add arc command completed and arc properties dialog completed.
- What's New Button on Home Ribbon Menu shows this text dialog.

19th. April, 2009

- This What's New dialog box.
- Add image command completed and image properties dialog completed.
- Add coordinate command completed and coordinate properties dialog completed.
- Add dimension command completed and dimension properties dialog completed.
- Add rectangle command completed and rectangle properties dialog completed.
- Add line command completed and line properties dialog completed.
- Add ellipse command completed and ellipse properties dialog completed.
- Add horizontal, vertical and point guide commands completed and their associated properties dialog.
- Repeat command is now the default. Turn it on/off using the viewport right click context menu (see below)
- Snap to objects now works.
- Viewports now have context sensitive right click menus.
- Status panel now shows directories used.