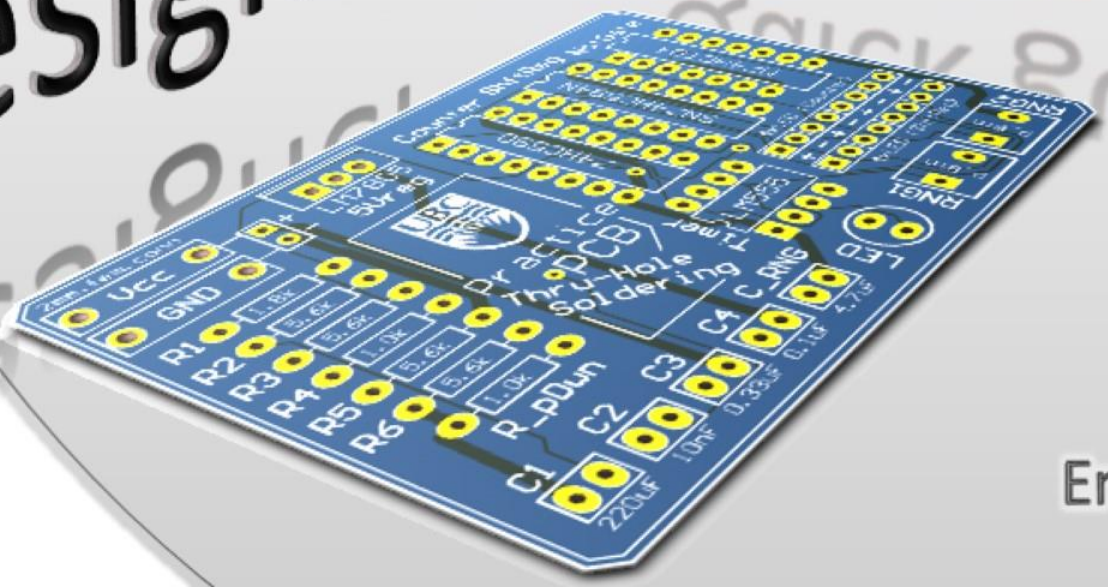
AndyLiu

Altium Designer 14

Quick guide



ECE

Engineering Services

9.21.15

http://eng-services.ece.ubc.ca/

PCB Design

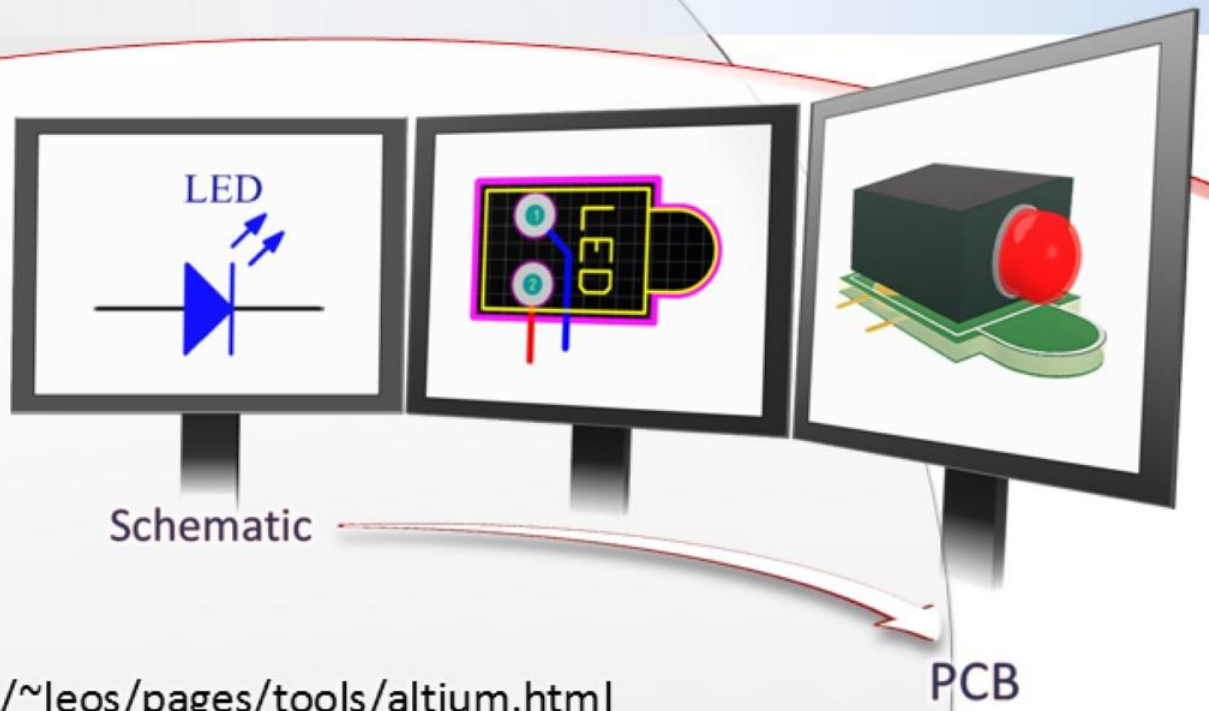
Endless possibilities

Contents

Altium - PCB designer tool

A complete product development system, Altium includes:

- Circuit design & simulation
- PCB design
- Signal integrity analysis



Some resources:

- http://www.ece.ubc.ca/~leos/pages/tools/altium.html

ECE download site:

- https://download.ece.ubc.ca/

'Altium Live' account creation site:

- http://live.altium.com/#signin

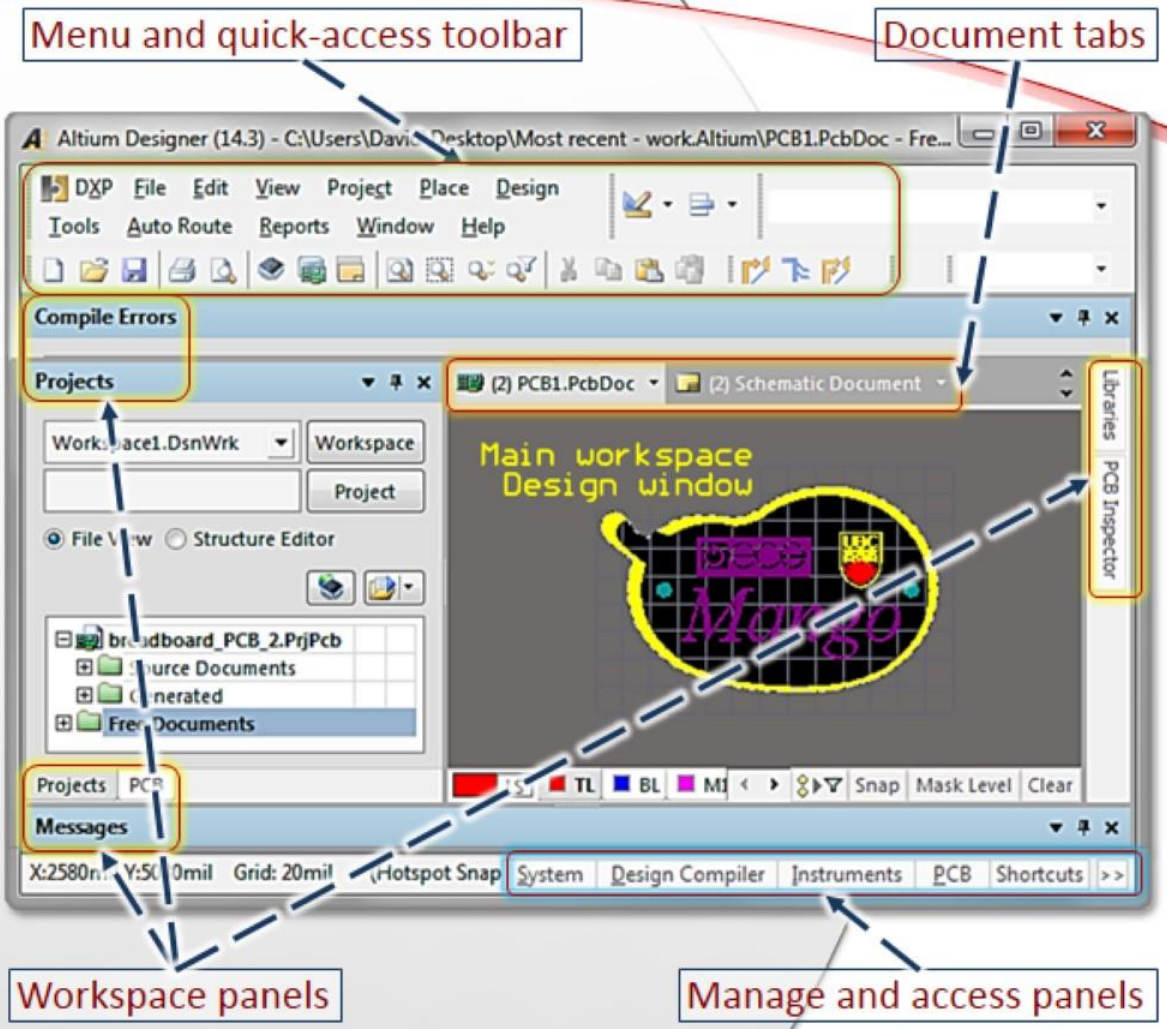
A mouse with a scroll-wheel and multiple large monitors are recommended when working with Altium designer.

Save often as the software licensing server may occasionally disconnect.

Altium – environment

The Altium environment and some of its features of note - the layout may be adjusted to suit personal needs or preferences.

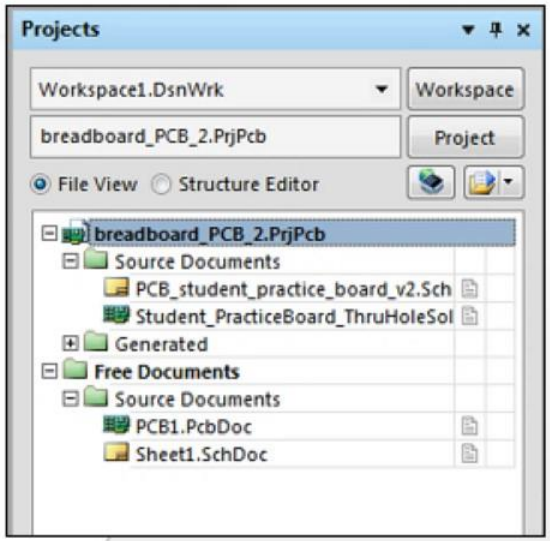
Tools and features may change depending on the type of document currently active

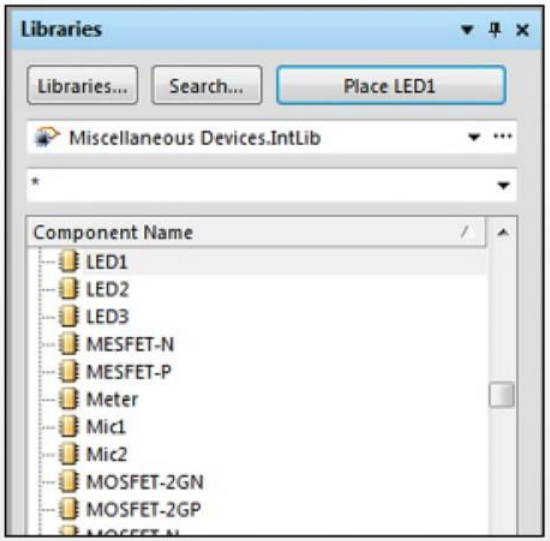


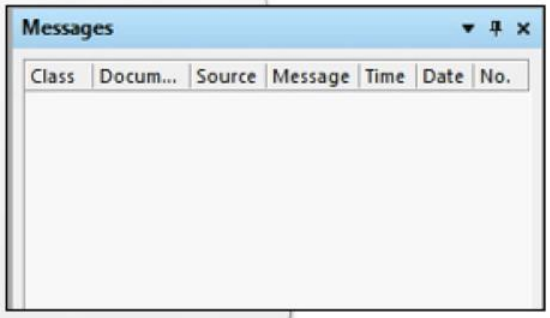
Workspace – general panels

Workspace panels could be either document-specific or global and are meant to improve user efficiency.

Panels could either be left floating or be docked together (adjoined or tabbed) along the edges of the workspace.







Projects

- View projects and associated files
- View free (unassociated) documents

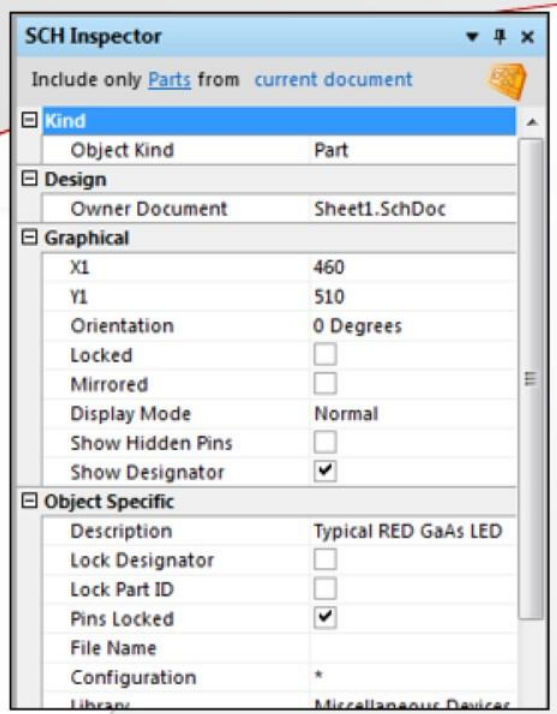
Libraries

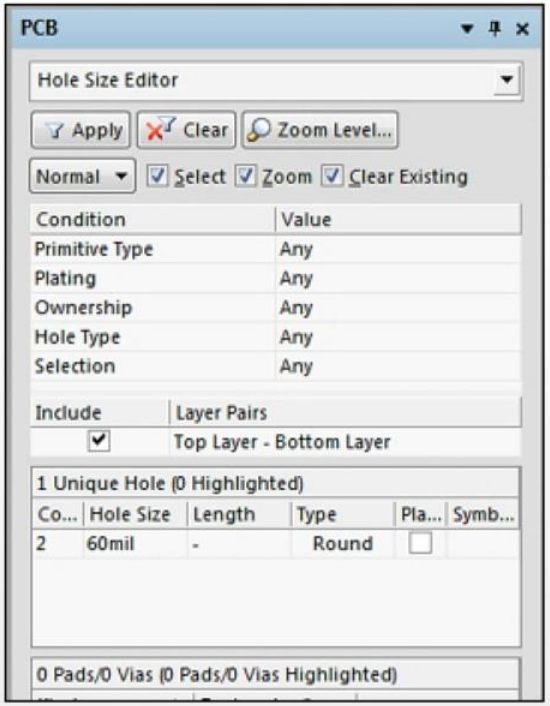
- Locate designs from pre-compiled libraries
- Toggle 'footprint' types
- Create or modify designs

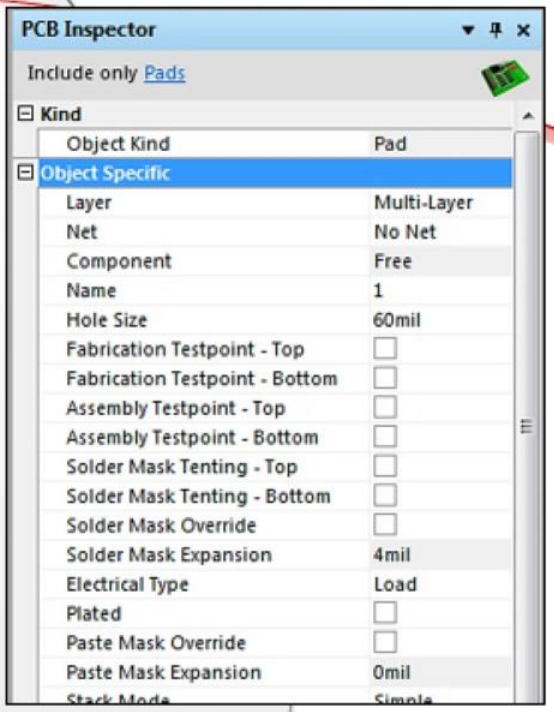
Messages

- View status info and results
- Locate and highlight warnings and errors

Workspace – Sch & PCB panels







SCH Inspector

- View or edit object properties in schematic files – affects selected objects

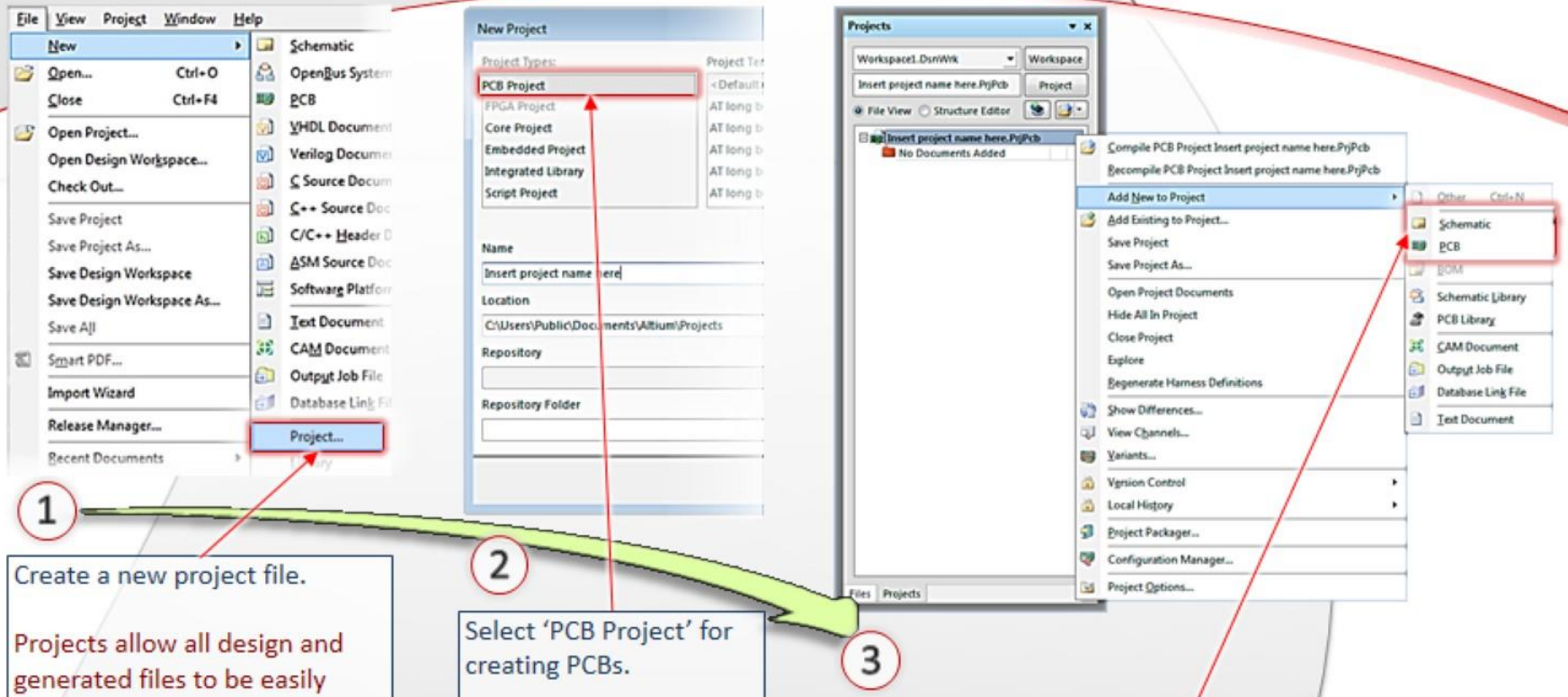
PCB

- View or edit board or object properties in PCB files – could affect all objects

PCB Inspector

- View or edit object properties in PCB files – affects selected objects

Project – starting a new PCB



Create a new project file.

Projects allow all design and generated files to be easily grouped together in a single location

Select 'PCB Project' for creating PCBs.

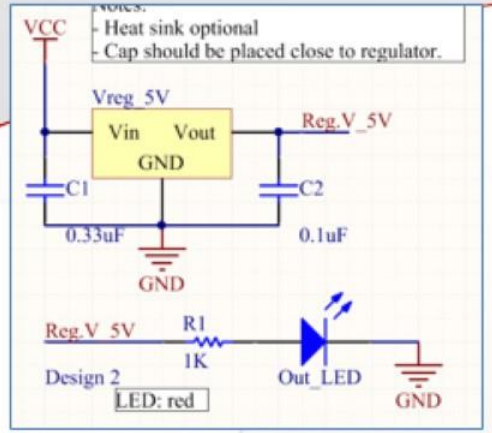
(Select 'Integrated Library' if you wish to create or modify library designs.)

Create both a schematic and a PCB file for your project.

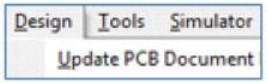
Schematic and PCB files are linked in a project, so changes made to one file could be imported to the other file.

Altium can detect any differences between the two files.

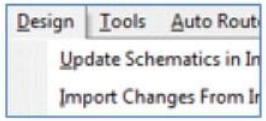
Project – from Sch to PCB 1/2

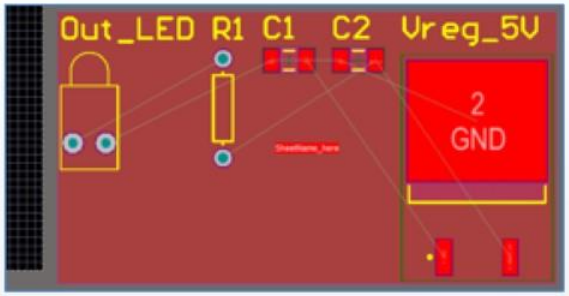


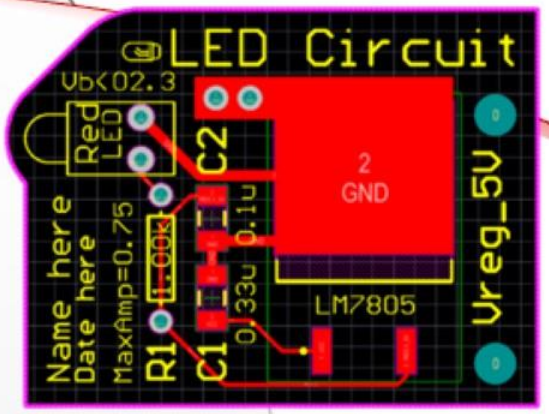
Update PCB from Schematic file

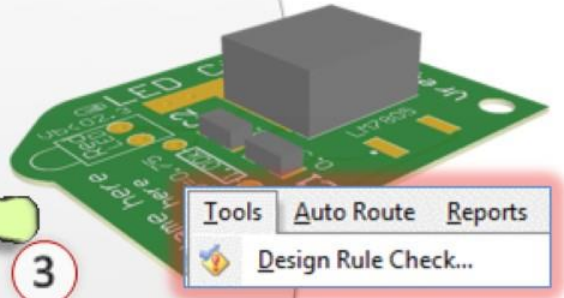


Update PCB from PCB file









1

Create your schematic diagram and compile your design for errors.

The properties of objects could be altered. Download or create new libraries for access to a greater variety of designs.

Circuits can be simulated if simulation files are included.

2

Import the schematic into the PCB file – this option is present in both files.

Altium will generate a (red) room by default; there is an option to omit the room in the import menu.

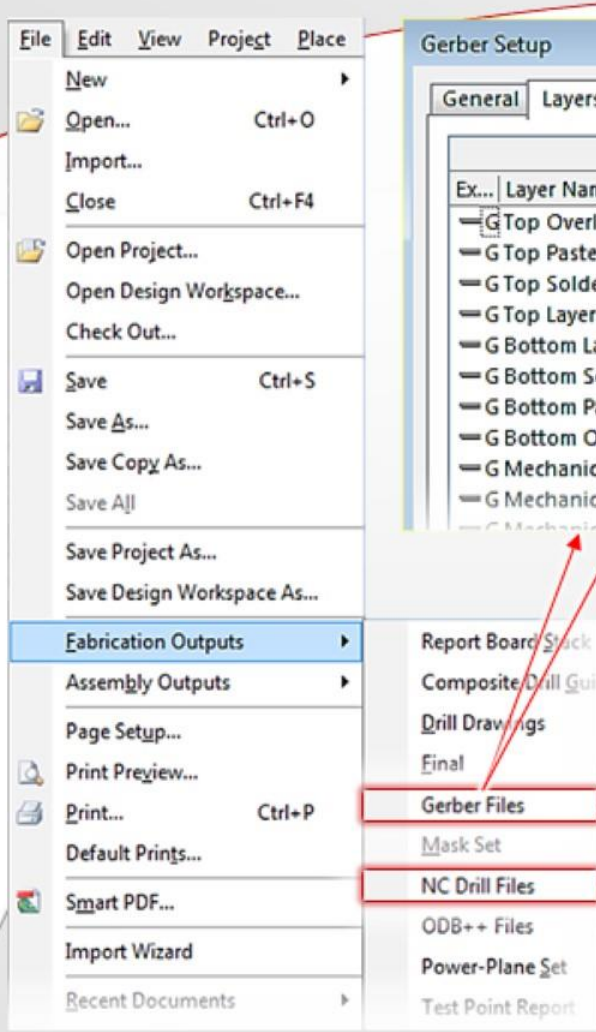
Changes to the schematic design could be done by importing from the PCB file.

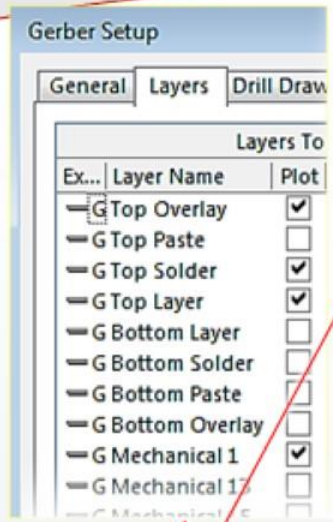
3

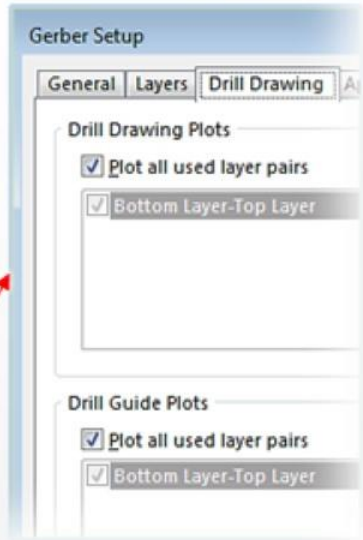
Create your PCB design on the (black) PCB surface. Check your finalized design.

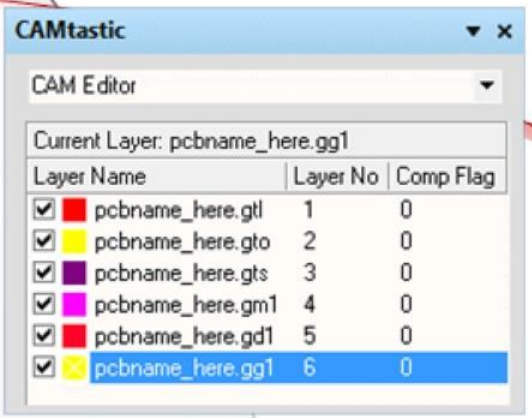
Rooms designate where components may reside – for small designs, a room should cover the entire PCB. Rooms may be hidden from view.

Project – from Sch to PCB 2/2









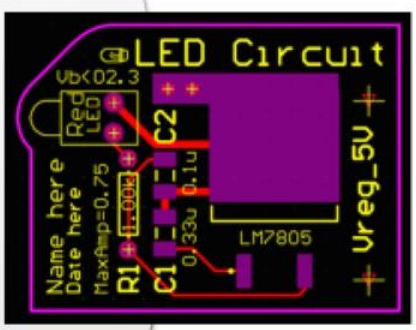
4

Generate fabrication files (Gerber, NC Drill) for your PCB design.

Gerber Setup
- Pane 2: select the PCB layers you want manufactured
- Pane 3: select both 'plot all used layer pairs'

NC Drill Setup:
- the default settings may be used

5



View generated fabrication files and check for issues in Cam Editor (Camtastic).

Changes or corrections to PCB designs could be done in Camtastic.

Sch – features of designs

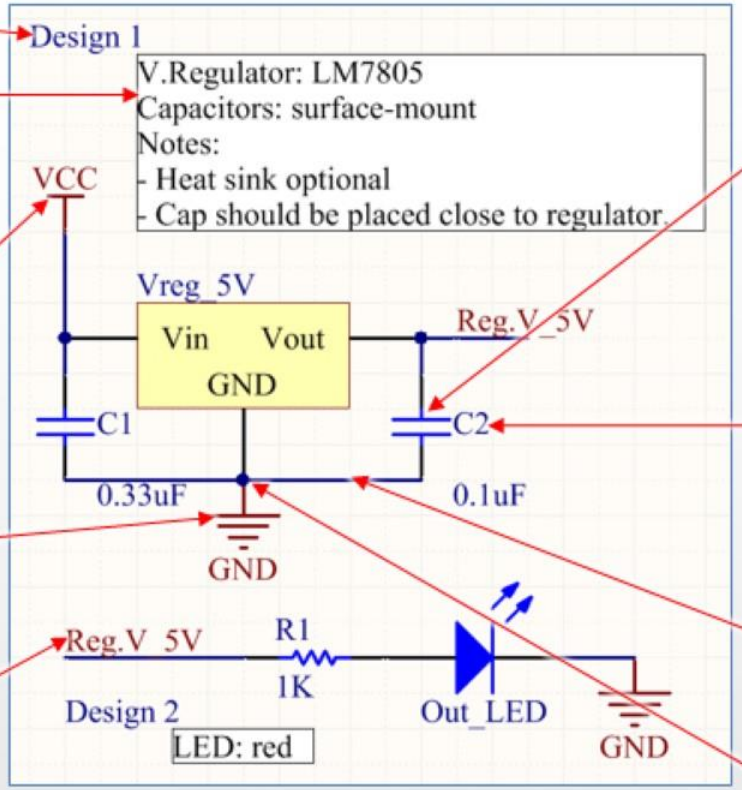
Text string: a text object

Text frame: a text box object

VCC net label: a type of 'net label' specifically for the input voltage; to adjust the input voltage for components with an absent power pin, edit the object to unhide the pin or to adjust the pin's connection

Power port: a type 'net label' specifically for the circuit's reference point

Net label: an electrical node – all nodes with the same name are electrically connected



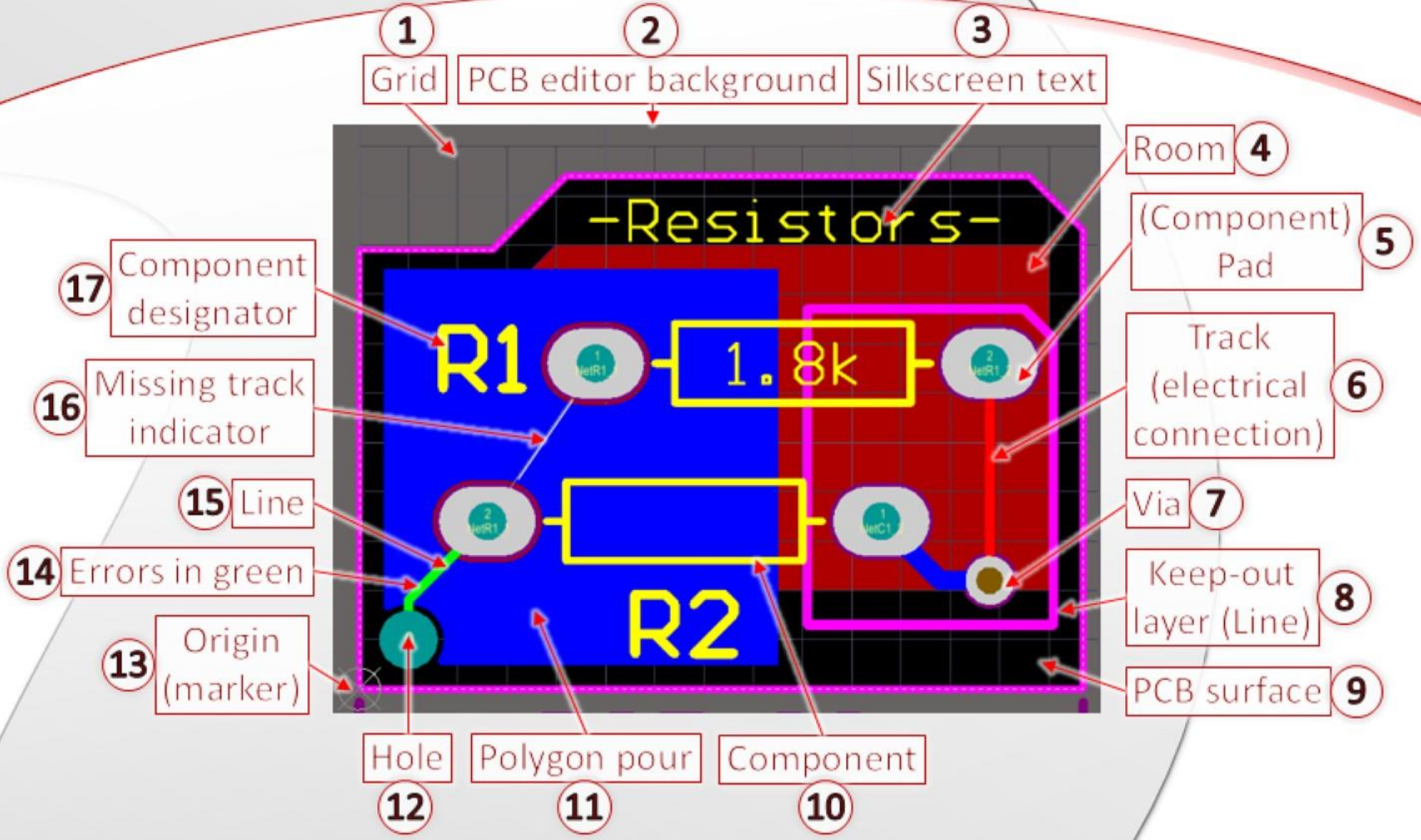
Component: a design from a library file; integrated libraries contain both schematic and PCB representations of components

Component designator: a component's unique ID; when placing multiple copies of the same component, designators auto-increment if the ID is set prior to placement – [Tab]

Wire: an electrical connection

Junction point: a dot that indicates that a wire has an end point on another wire or crosses a component pin

PCB – features of designs 1/2



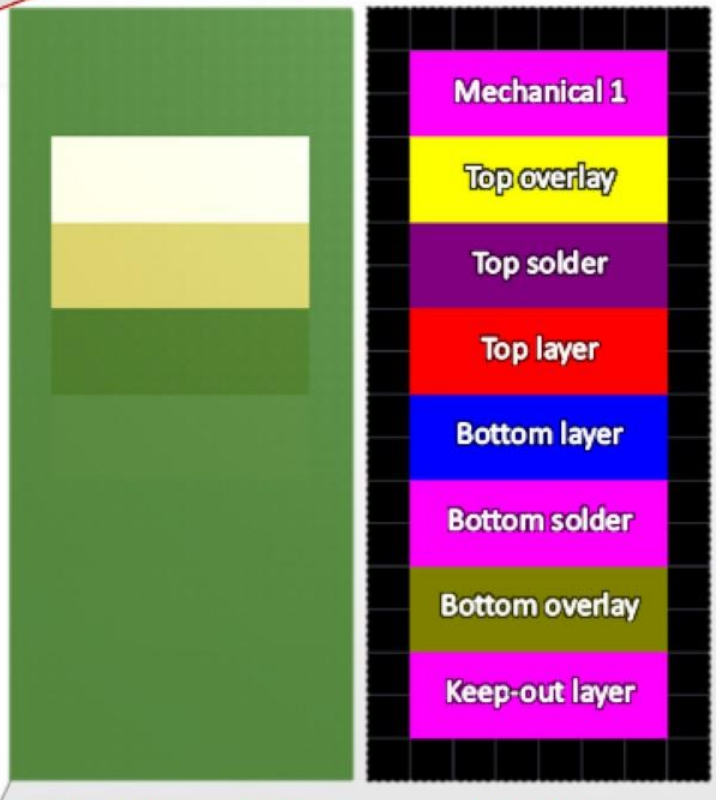
A small PCB and some features of note

PCB – features of designs 2/2

1. **Grid**: helps align or measure objects – it could be readily changed
2. **PCB editor background**: default workspace background
3. **Silkscreen text**: PCB surface text or images are placed on top or bottom overlay layers
4. **Room**: defines where (specific) components could be placed – not necessary for small designs
5. **(Component) Pad**: pads allow the placement of through-hole or SMD components on a PCB
6. **Track (electrical connection)**: a type of line that acts as signal connection – tracks are distinct from lines and are placed with a different command
7. **Via**: used for electrically connecting different layers on a PCB – different via types exist
8. **Keep-out layer line**: could specify region where specific objects may or may not reside
9. **PCB surface**: represents the physical surface of a PCB – mechanical layer 1 is used for PCB outlines
10. **Component**: a library file "footprint" of a component – a PCB component representation
11. **Polygon pour**: a region on a layer – could be used for signal planes or heat-sinks
12. **Hole**: a pad set to non-plated and pad size to 0 – generally used for mounting purposes
13. **Origin**: defines the origin of the design – useful in designing and manufacturing PCBs
14. **Errors in green**: design rules set the design constraints; design errors or violations are shown in green
15. **Line**: general-purpose drawing object – a line that connects objects electrically is a track, a similar/related object
16. **Missing track indicator**: indicates a missing electrical connection for designs imported from schematic files
17. **Component designator**: designators are linked to their respective components and could be modified from either schematic or PCB files

PCB – board layers

Common layers in PCB editor



3D view - top 2D view

Mechanical layers are **not manufactured**; they represent details such as outlines, dimensions, or fabrication information.

Total number of mechanical layers: 32

Overlay layers (silkscreen layers) are used for PCB surface text or images.

Solder layers represent the protective solder masks on PCBs; any object placed on this layer will **remove** parts of the solder mask. Solder masks are what gives PCBs their colors.

Electrical (**signal**) connections are created on the top layer or the bottom layer – for PCBs with 2 layers.

Possible number of signal layers: 32

Possible number of internal power planes: 16

The **keep-out** layer defines the boundaries of regions where certain objects may or may not reside.

PCB – layer menus

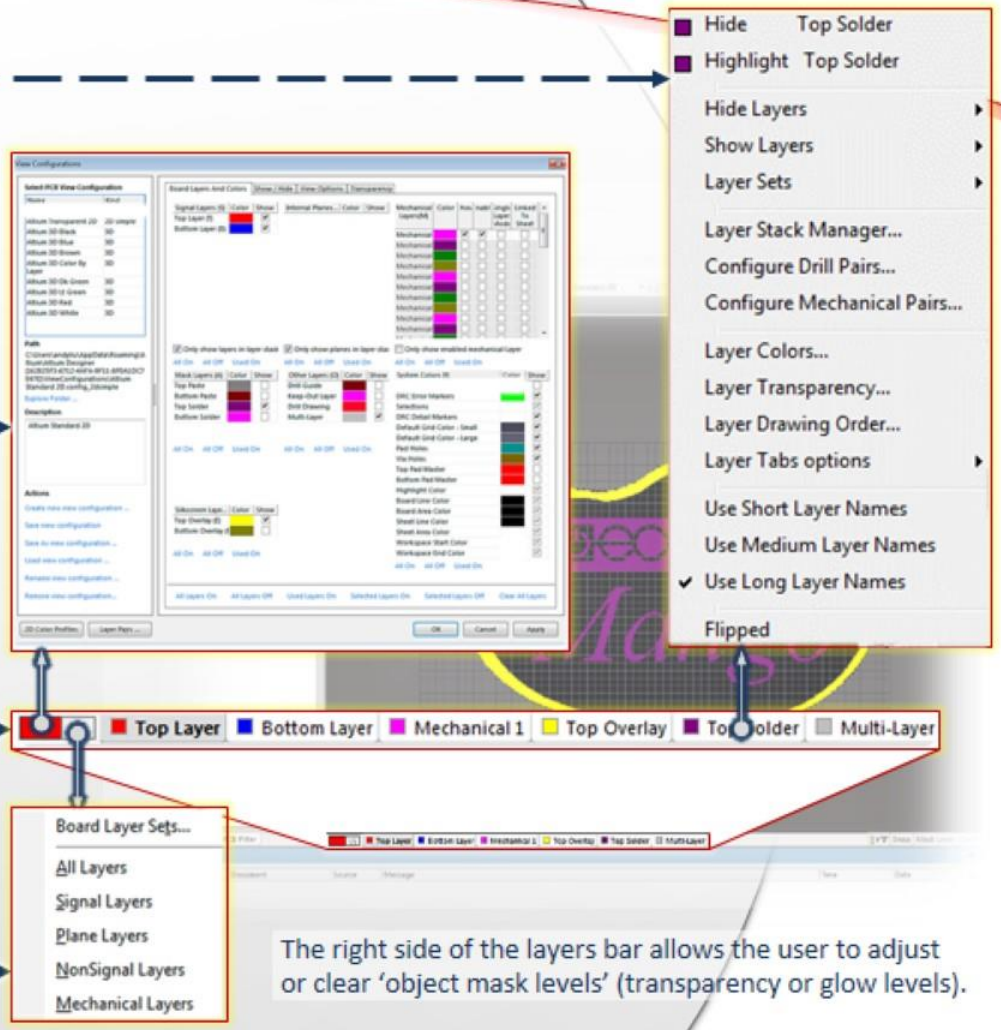
[mouse.right-click] – manage the chosen layer or other specific layers

View configurations menu – for setting and managing:

- PCB views and colors
- layer colors
- hide or inactivate layers
- hide rooms

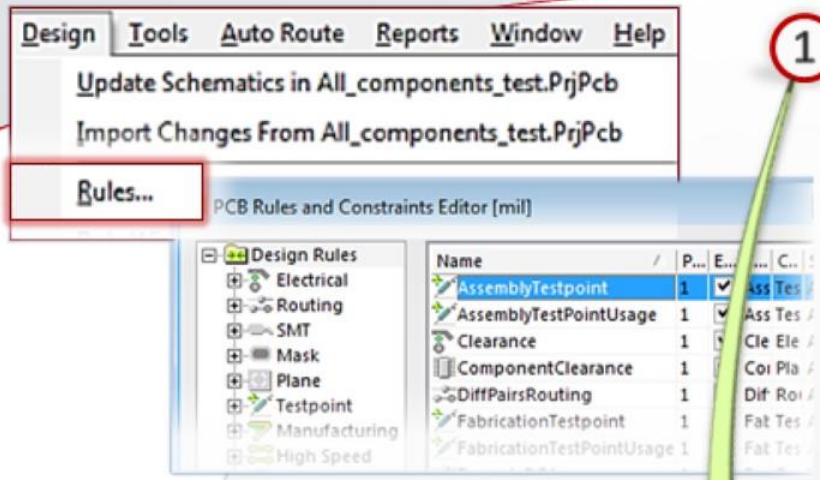
Layers bar – shows current layer and allows user to quickly toggle between layers

Manage layer sets menu – toggle sets of layers



The right side of the layers bar allows the user to adjust or clear 'object mask levels' (transparency or glow levels).

PCB – design rules & checks



1

Open the 'PCB Rules and Constraints Editor' to set the design rules and constraints.

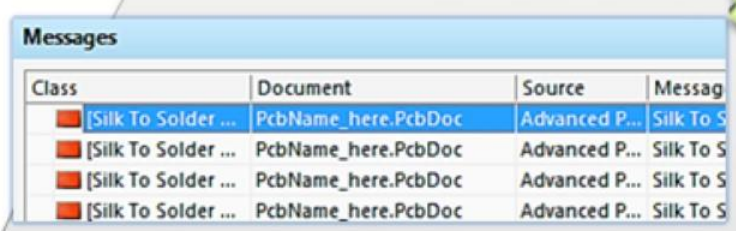
Different designs have their own unique requirements and different manufacturers possess different machines and fabrication techniques – rules should be customized for every PCB design.

Rules could be imported or exported, and rules **only apply** to the current PCB file. Apply rule changes before closing the rules menu.

2

Check for design errors or violations.

A finalized PCB design should have no errors or violations.



3

Examine the 'Design Rule Check' (DRC) generated report file or check the 'Messages' panel to view the results of the check.

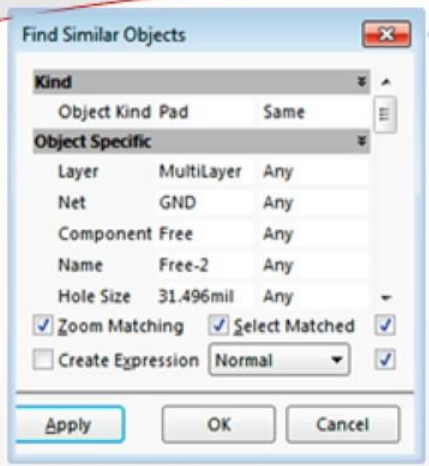
You may have to check the 'Create Report File' option (in the 'Design Rule Check' menu) to generate the DRC file.

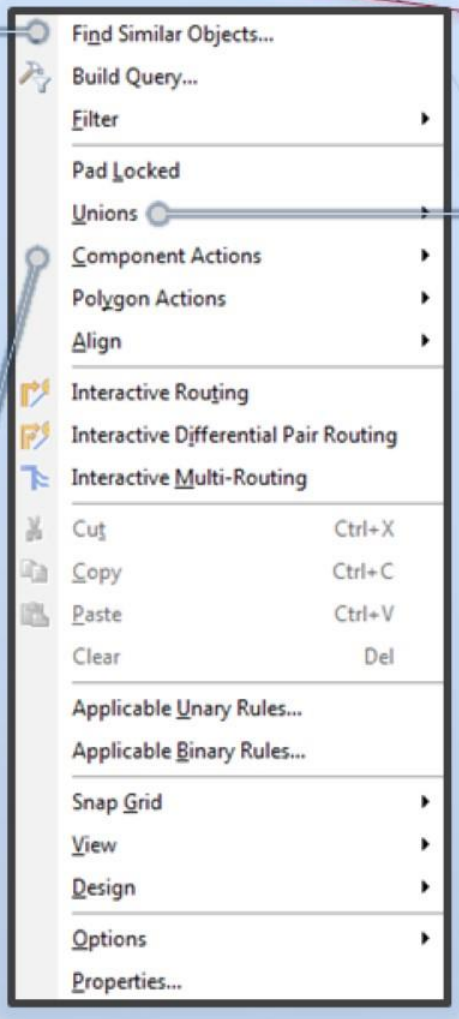
Double-click violations in the 'Messages' panel to zoom in on them.

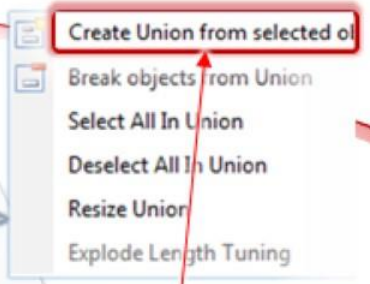
PCB – some useful tools

The 'Find Similar Objects' menu allows certain objects to be located and allows multiple objects to be selected at once.

The menu, in combination with the Sch or PCB Inspector panel, allows entire portions of the design to be modified instantly.





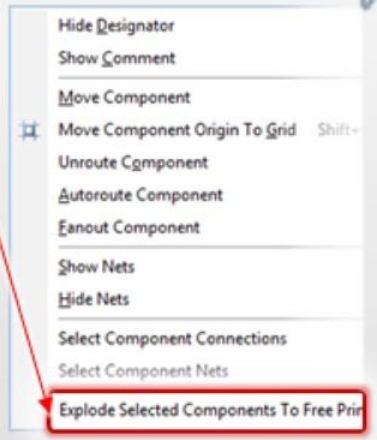


Objects in a union are grouped together – moving one object in a union moves the rest in the union.

To move select objects within a union, the 'move all selected objects' command, [m]+[s], is required.

An exploded component is reduced to free unbound primitive objects and is no longer considered a PCB 'footprint.'

There is no command to return exploded designs to their original state – besides the undo command.



Menu accessed by right-clicking on selected objects

Sch & PCB – some useful tools

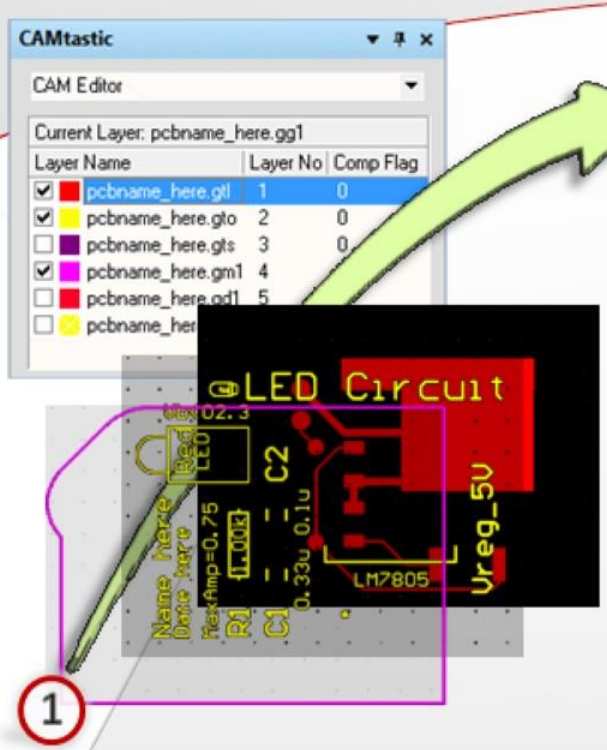
Schematic tool		Shortcut
Selection	**Find similar objects menu**	**[Shift]+[f]**
	Select all outside region	[e]+[s]+[o]
View	**Fit schematic to screen**	**[v]+[f]**
Object layout	**Rotate object**	**[Space]**
	Change wire placement mode	[Shift] + [Space]
	Break/split wire	[e]+[w]
Schematic	**Edit properties during design placement**	**[Tab]**
	Alignment menu	[a]

Press [Esc] or right-click the mouse in the middle of an action to cancel it – applies to both Sch and PCB.

In PCB 'single layer view,' hidden layers are protected from being edited.

PCB tool		Shortcut
Selection	**Move all selected objects**	**[m]+[s]**
	Find similar objects menu	**[Shift]+[f]**
	Select all outside region	[e]+[s]+[o]
View	**Fit PCB to screen**	**[v]+[f]**
	Board, 2D, 3D layout modes	[1], [2], [3]
	Flip PCB	[v]+[b]
PCB layer {current layer}	**Single layer view**	**[Shift]+[s]**
	Place track (wire/elec. connection)	[p]+[t]
	Place line (general-purpose drawing)	[p]+[l]
PCB	**Adjust grid size**	**[g]**
	Rotate object	**[Space]**
	Alignment menu	**[a]**
	Measure distance	[Ctrl]+[m]
	Define PCB shape (manual dragging)	[1]+[d]+[d]
	Define PCB shape (from drawn outline)	[d]+[s]+[d]
	Change units (imperial ↔ metric)	[q]
	Refresh	[End]

Camtastic – checking for errors



Using the 'CAMtastic' panel, view layers individually or with respect to one another to check for any issues with the files or the design.

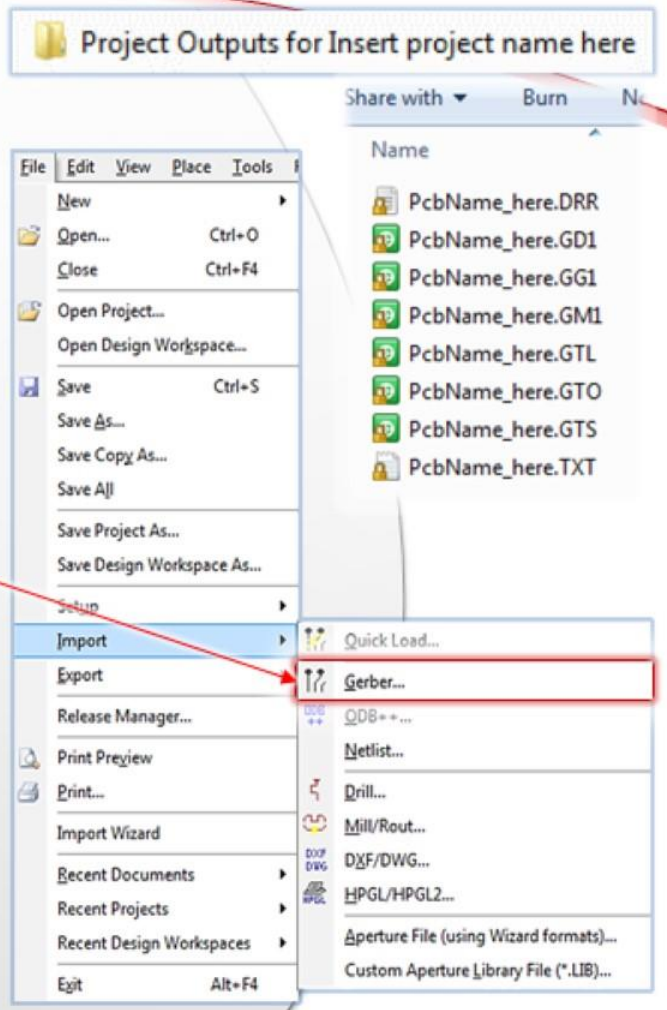
Zoom out far to see if any unwanted design features or objects are present.

2

All of the generated project files should be located in a 'Project Outputs' folder. The Gerber data files for the layers, the .DRR file, and the .TXT file, should be placed in a .zip folder for manufacturers.

To superimpose and check the generated Gerber data files, delete the original Gerber data in Camtastic and import the generated files.

Open the .DRR file to check if the hole sizes are what is expected. To check the NC Drill .TXT file, copy the file then convert its file extension into .GD1 – open the file in Altium and compare it against the generated .GD1 file.



Best practices – things to note

General:

- **Altium may not generate fabrication files properly**
  - ❑ **check over fabrication files**
- PCBs are 'cut out of' panels, so one panel could contain multiple copies of smaller designs – expenses are partially dependent on the number of panels

Manufacturing:

- **Manually route all design-critical traces**
- Connect traces to the center of pads
- Add 'teardrops' to pads

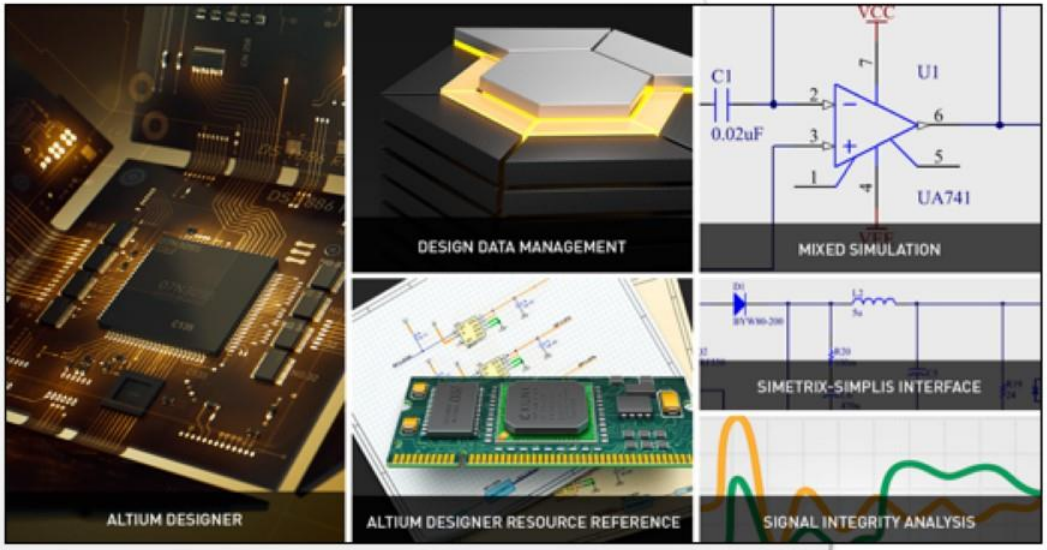
Design:

- Group associated components into 'functional blocks'
- Separate analog and digital components
- Capacitors and inductors may generate EMI
- Place de-caps or op-amp resistors close to their respective components
- Connect all copper regions to a specific signal (net)
- signal planes and heat sinks may be required and should be considered
- Required trace widths are dependent on expected current and temperature
- Shorter traces are preferred
- Avoid 90° bends in traces
- Keep signal traces perpendicular to one another – for adjacent signal layers

Additional info

The Altium online wiki is a powerful resource with detailed guides, examples, and explanations.

http://techdocs.altium.com/



--- Contacts for more info on Altium or these slides---

Dr. Roberto Rosales – robertor@ece.ubc.ca
Tihomir Tunchev – tihomirt@ece.ubc.ca
Andy Liu – andyliu@ece.ubc.ca