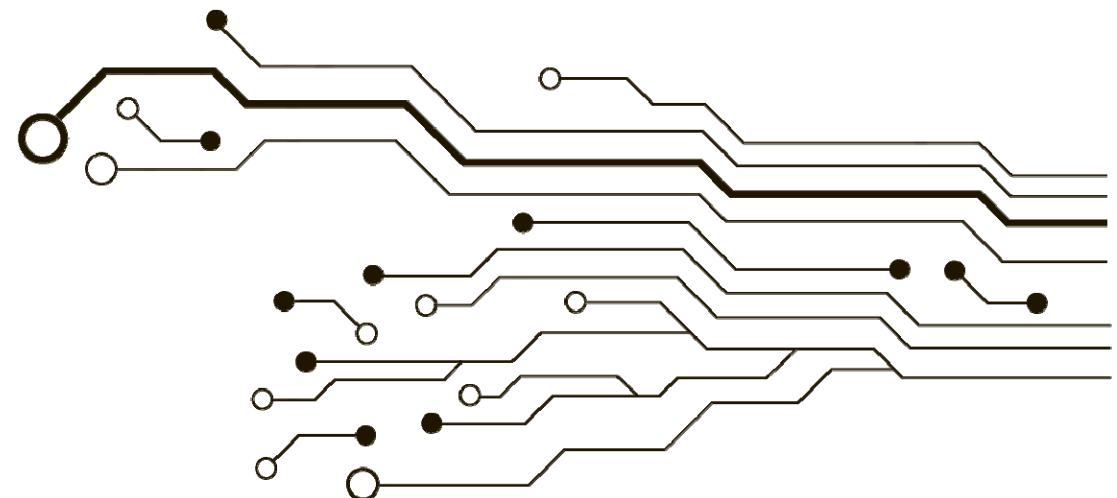


PCB Design through Altium Designer & Eagle

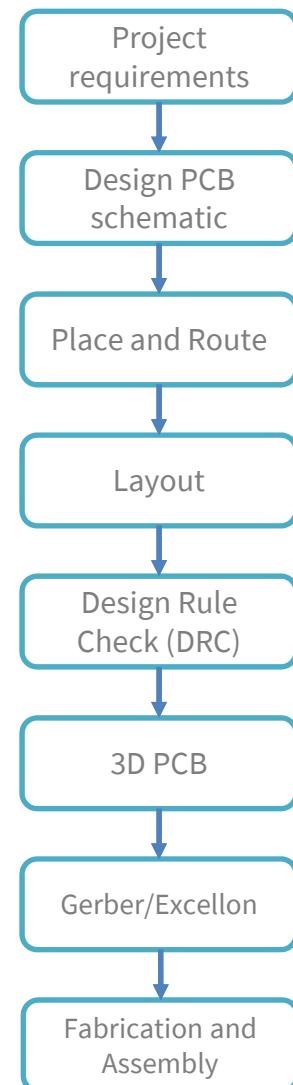
William Fornaciari

08/2018



PCB Design Workflow

- **Project requirements:** definition of the characteristics of the project and of the board (dimensions, number of layers...).
- **Design PCB schematic:** design your PCB schematic
- **Place and Route:** placement of components and tracking of interconnections.
- **Layout:** complete geometric / mechanical description of the board.
- **3D PCB:** rendering and viewing PCB board in 3D.
- **Gerber/Excellon:** describes the geometry and characteristics of copper interconnections for each board layer, while Drill File (in format Excellon) describes all the characteristics of the Vias to be drilled on the board. These files are supplied to PCB manufacturer.
- **Fabrication /Assembly:** Gerber and Excellon files are supplied to the PCB house that deals with the manufacture and assembly of the PCB board.



Altium Designer PCB Design

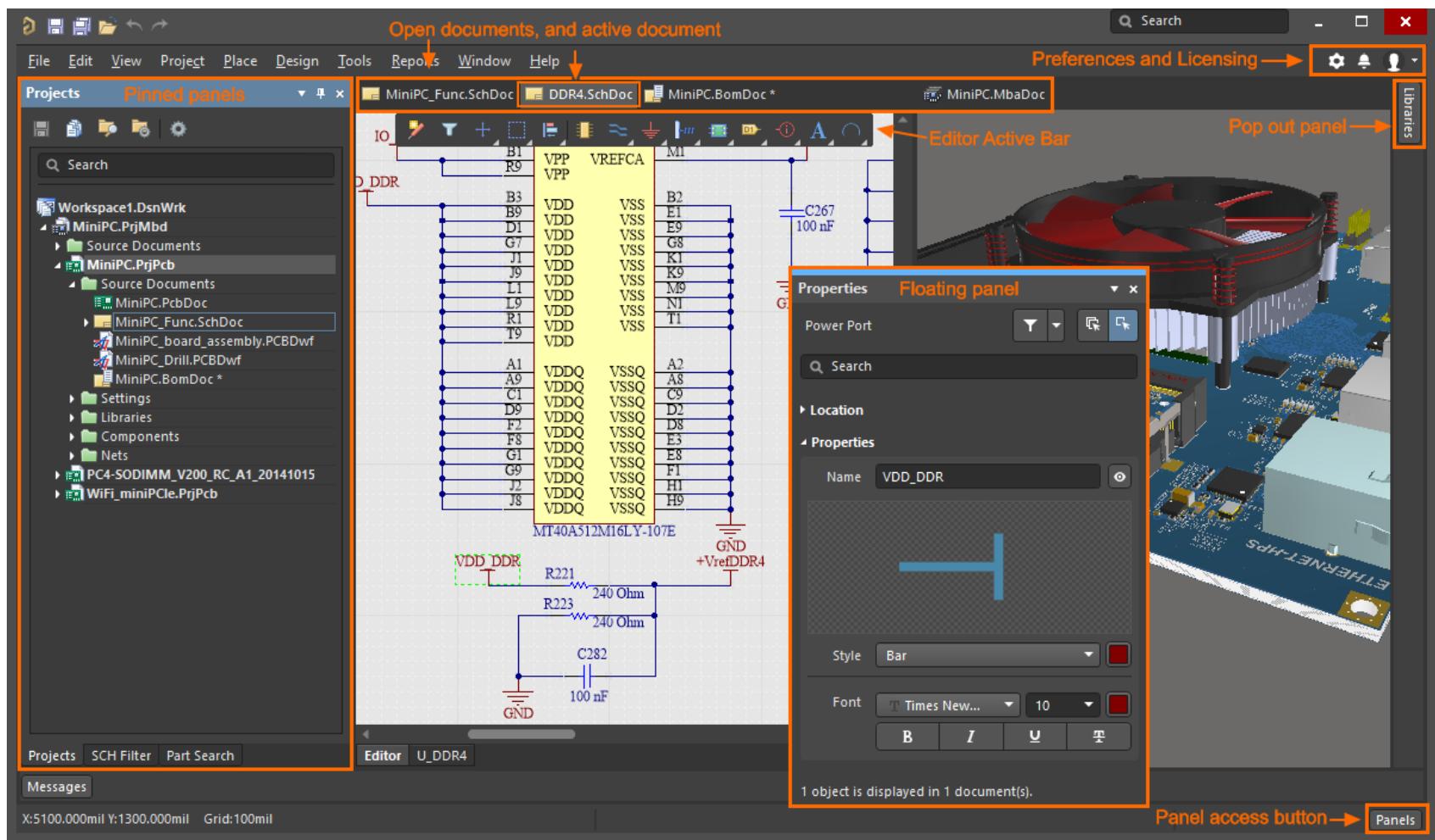
- Altium Designer is one of the most popular of the high end PCB design software packages on the market today. It is developed and marketed by Altium Limited. Including a schematic, PCB module, and an auto-router and differential pair routing features, it supports track length tuning and 3D modeling.
- It includes tools for all circuit design tasks: from schematic and HDL design capture, circuit simulation, signal integrity analysis, PCB design, and FPGA-based embedded system design and development. In addition, the Altium Designer environment can be customized to meet a wide variety of users' requirements.



Altium Designer PCB Design

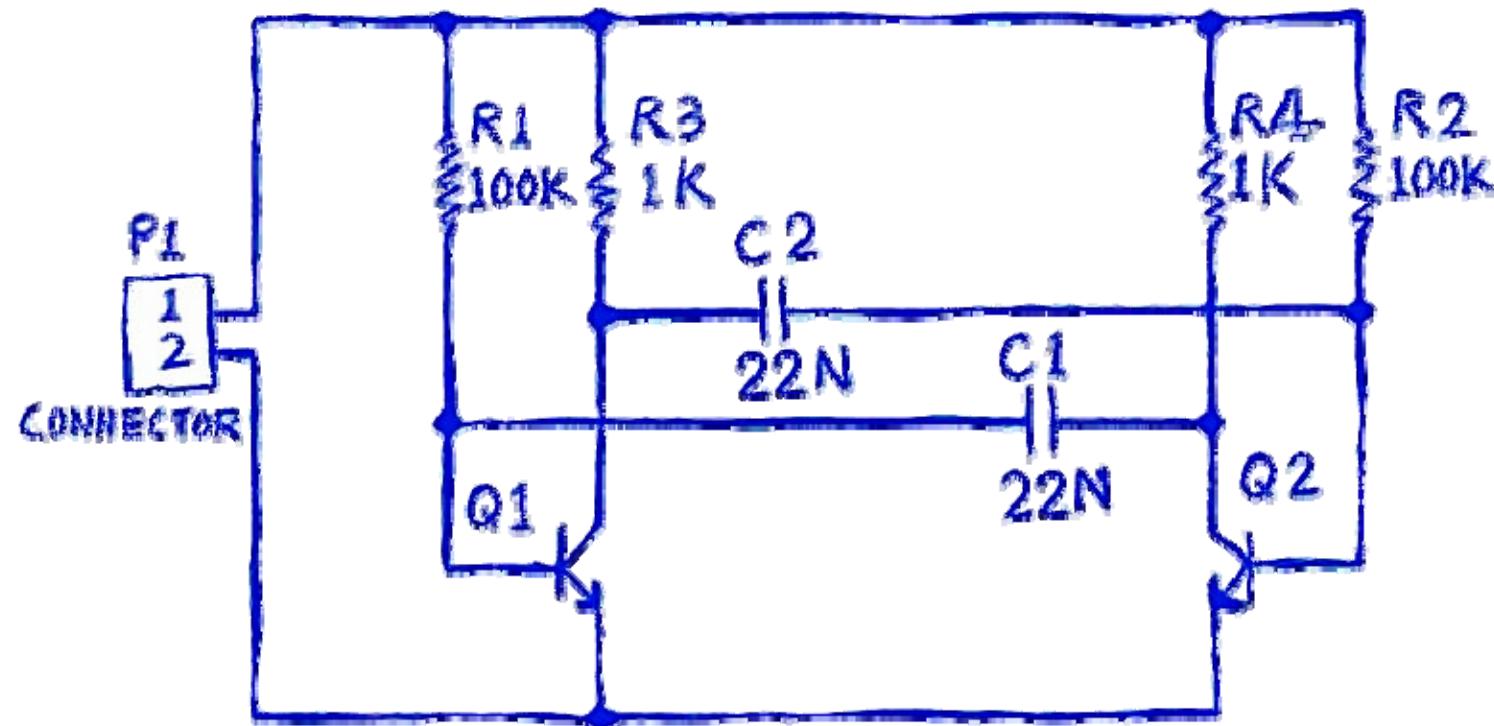
The Altium Designer environment consists of two main elements:

- The main document editing area.
- The Workspace Panels. There are a number of panels , that by default some are docked on the left side of the application, some are available in pop-out mode on the right side, some are floating, and others are hidden.



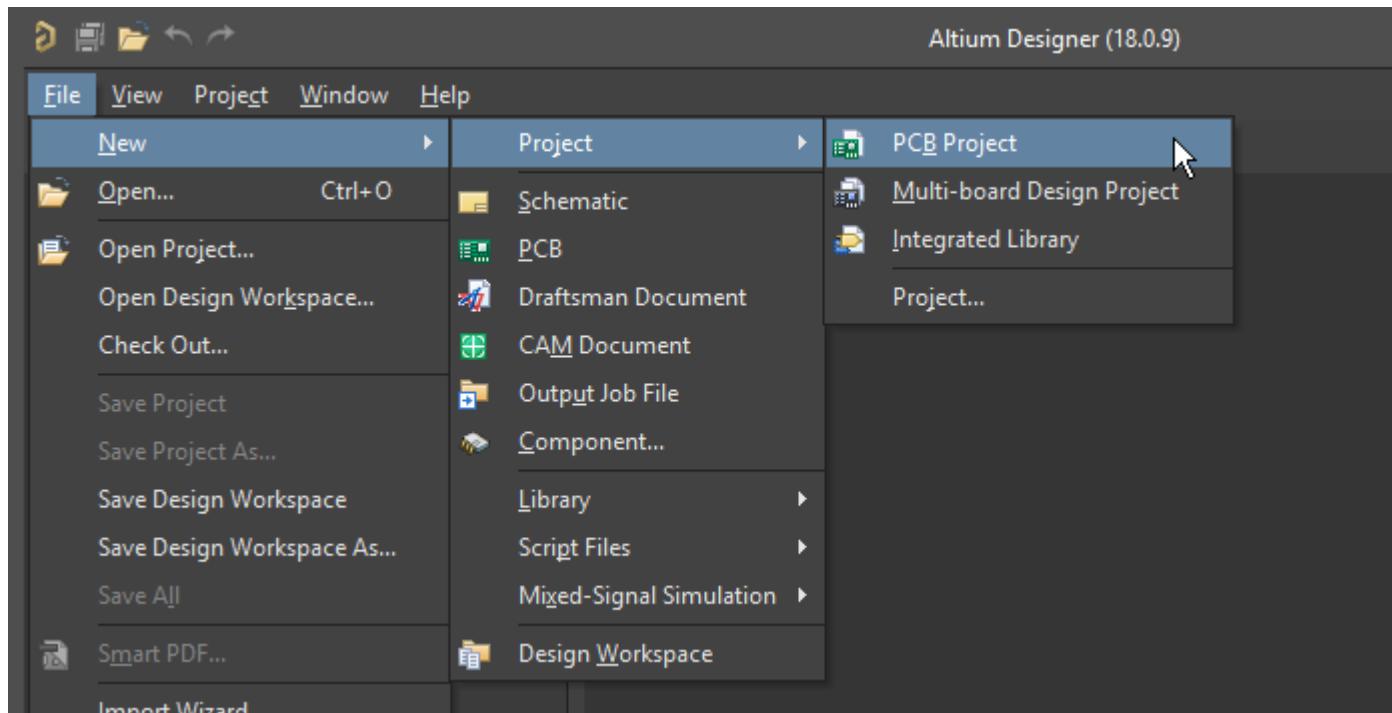
The Design

The design that will be creating for the schematic and then designed for a printed circuit board (PCB), is a simple astable multivibrator. We will refer to this schematic for the PCB design.



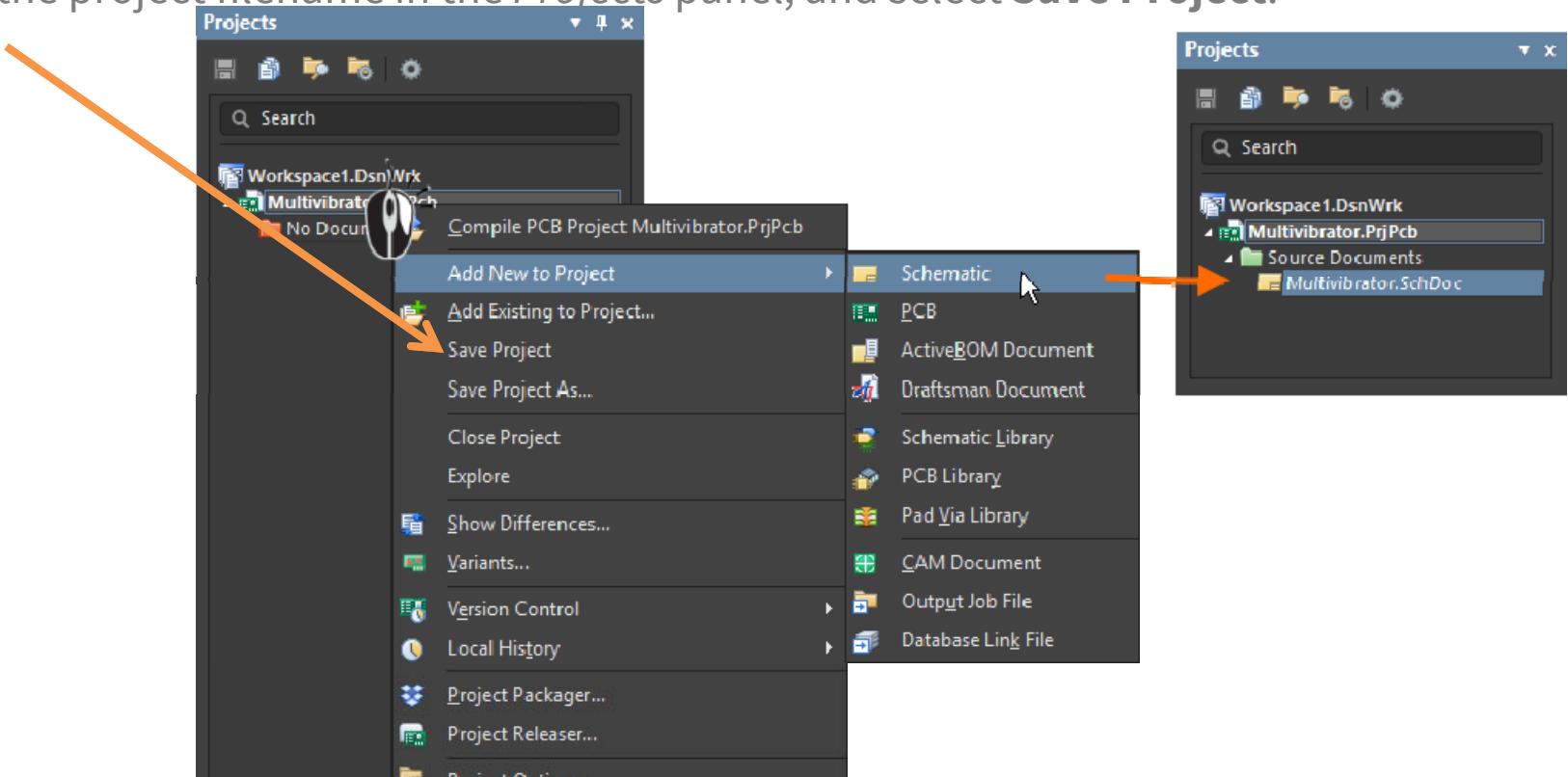
Creating a New PCB Project

- Select File » New Project » PCB Project from the menus.
- Select File » Save Project from the menus to open the Windows Save As dialog. Use the dialog to navigate to a suitable location, and enter the name Multivibrator in the File Name field.



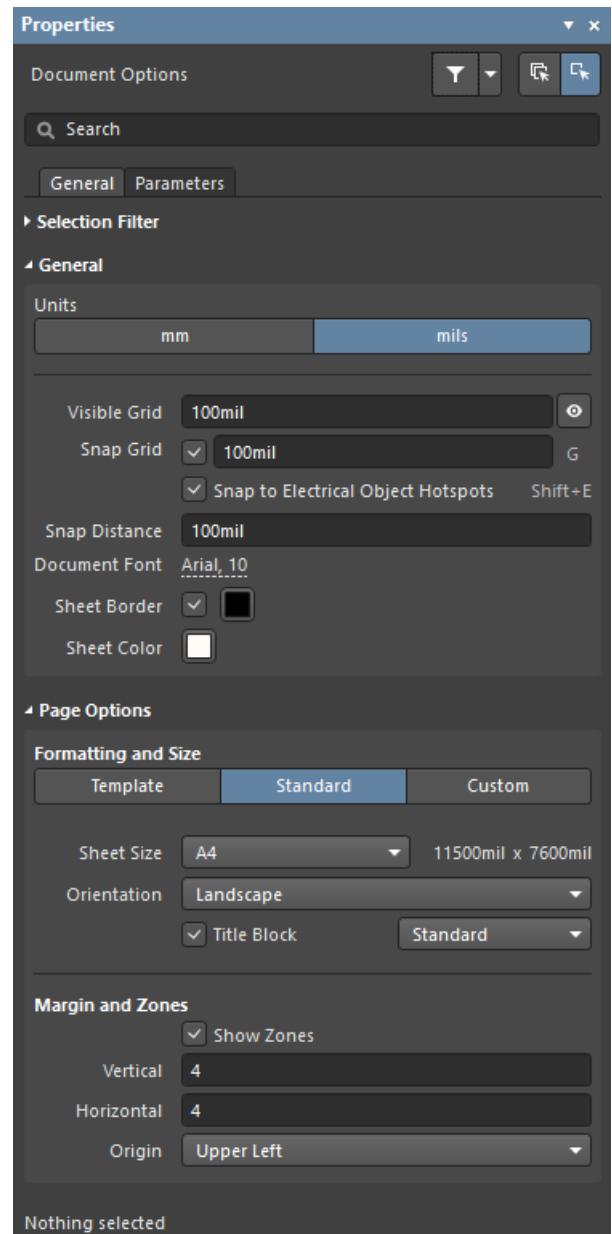
Adding a Schematic to the Project

- Right-click on the project filename in the *Projects* panel, and select **Add New to Project** » **Schematic**. A blank schematic sheet named **Sheet1.SchDoc** will open in the design window
- To save the new schematic sheet, select **File** » **Save As**. Type the name **Multivibrator** in the **File Name** field and click **Save**.
- Since you have added a schematic to the project, the project file has changed too. Right-click on the project filename in the *Projects* panel, and select **Save Project**.



Setting the Document Options : Sheet Size, Snap & Visible Grid

- Set the sheet size to A4, this is done in the **Page Options** section.
- Confirm that both the **Snap** and **Visible Grids** are set to 100mil.
- Save the schematic by selecting **File » Save** (shortcut: **F, S**).



Components and Libraries in Altium Designer

- Altium Designer components can be:
 - created in and placed from local libraries, or
 - placed directly from the Altium Content Vault, a globally accessible component storage system that contains thousands of components, each with a symbol, footprint, component parameters and links to suppliers.
- **Schematic Library:** Schematic component symbols are created in schematic libraries (*.SchLib), stored locally.
- **PCB Library:** PCB footprints (models) are stored in PCB libraries (*.PcbLib), stored locally. It includes Pads, Overlay Component, Glue Dots etc.
- **Altium Content Vault:** The Altium Content Vault is much more than a library. Components are stored in the cloud, accessible from anywhere. It includes: symbol, footprints, component parameters, and links to suppliers.

Accessing Components

- The two panels that are used to access components are shown below.

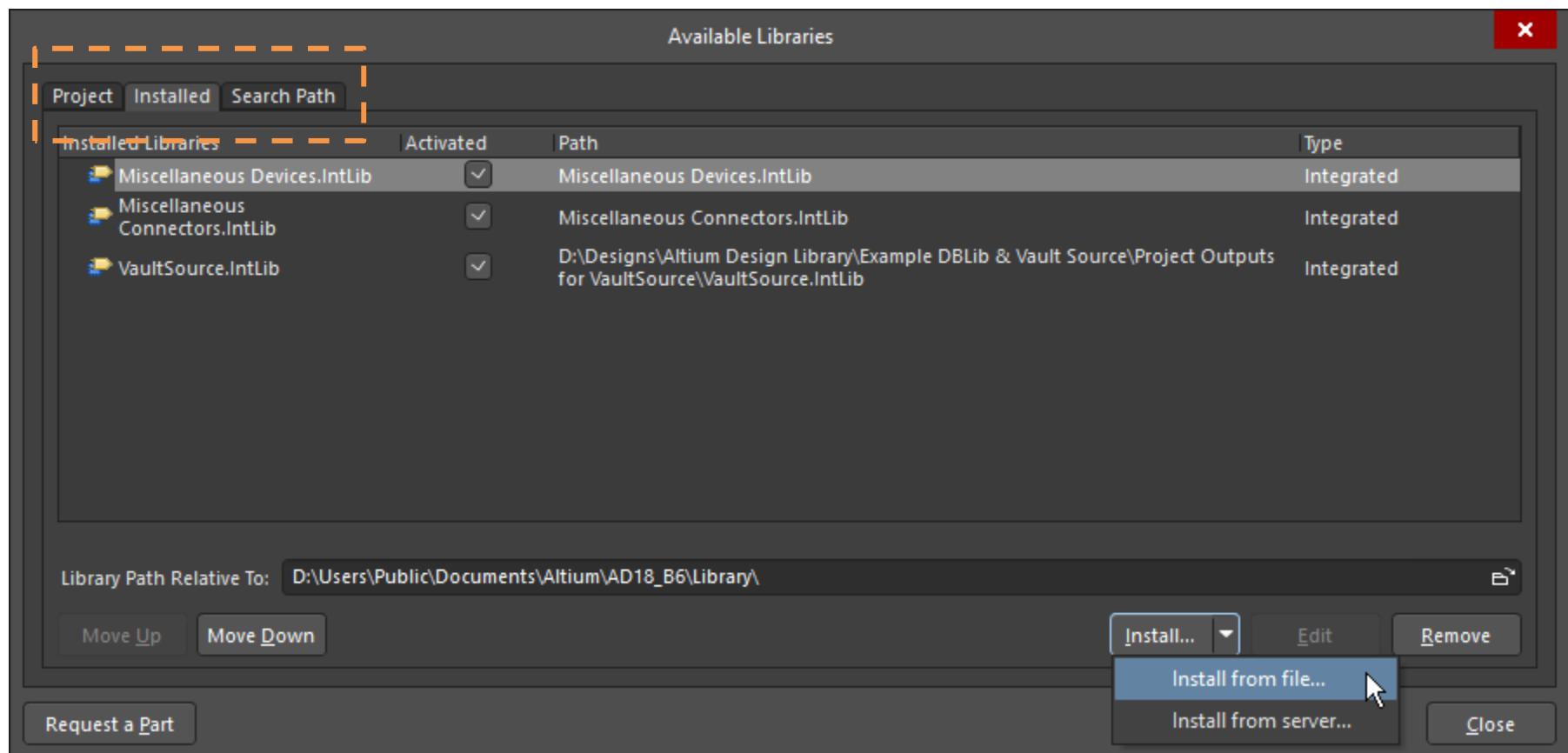
The screenshot displays two panels from the Altium Designer interface:

Libraries Panel: Shows a list of components in the "Miscellaneous Devices.IntLib" library. The component "2N3904" is selected, showing its description as "NPN General Purpose Amplifier", footprint as "TO-92A", and various model types including Signal Integrity, Simulation, and Footprint.

Explorer Panel: Shows the "Altium Content Vault" search results for "Unified Components\Components\ON Semiconductor\Discrete\Bipolar Transistors\General Purpose Transistors". The results table includes columns for Revision ID, Revision State, Name, and Description. The component "BC547C" is highlighted in the list. Below the table, detailed component information is shown, including Datasheet, DataSheetVersion, Manufacturer, and PackageDescription. To the right, there are 3D visualizations of the component's package and symbol.

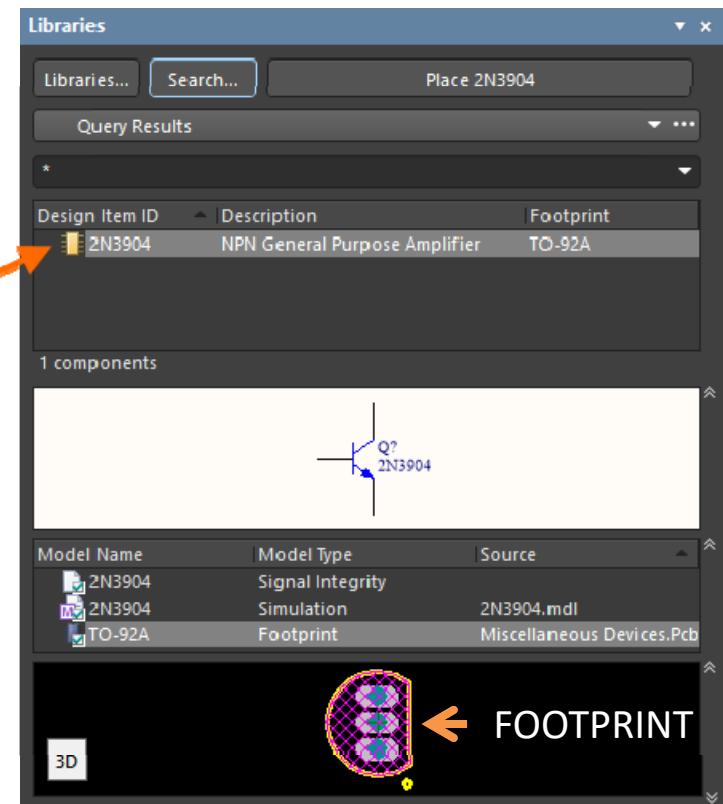
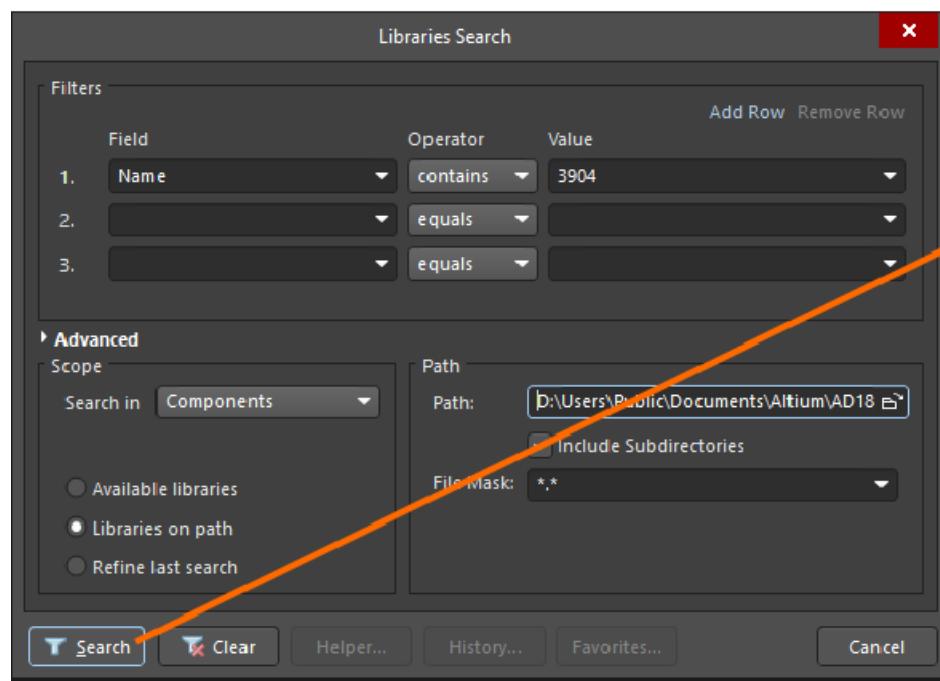
Making Libraries Available to Access the Components

- The libraries that are available include:
 - Libraries in the current project** - if a library is part of the project, then the components in it are automatically available for placement within that project.
 - Installed libraries** - these are libraries that have been installed in Altium Designer, their components are available for use in any open project.



Finding a Component in Libraries

- Search for the component using the Libraries **Search** dialog. You can search across:
 - installed libraries (**Available libraries**)
 - libraries on the hard drive (**Libraries on path**).
- Select components and click *Place 2N3904* (or double clicking or click & drag) bar to remove it onto the schematic paper. Set them to any direction based on your needs.



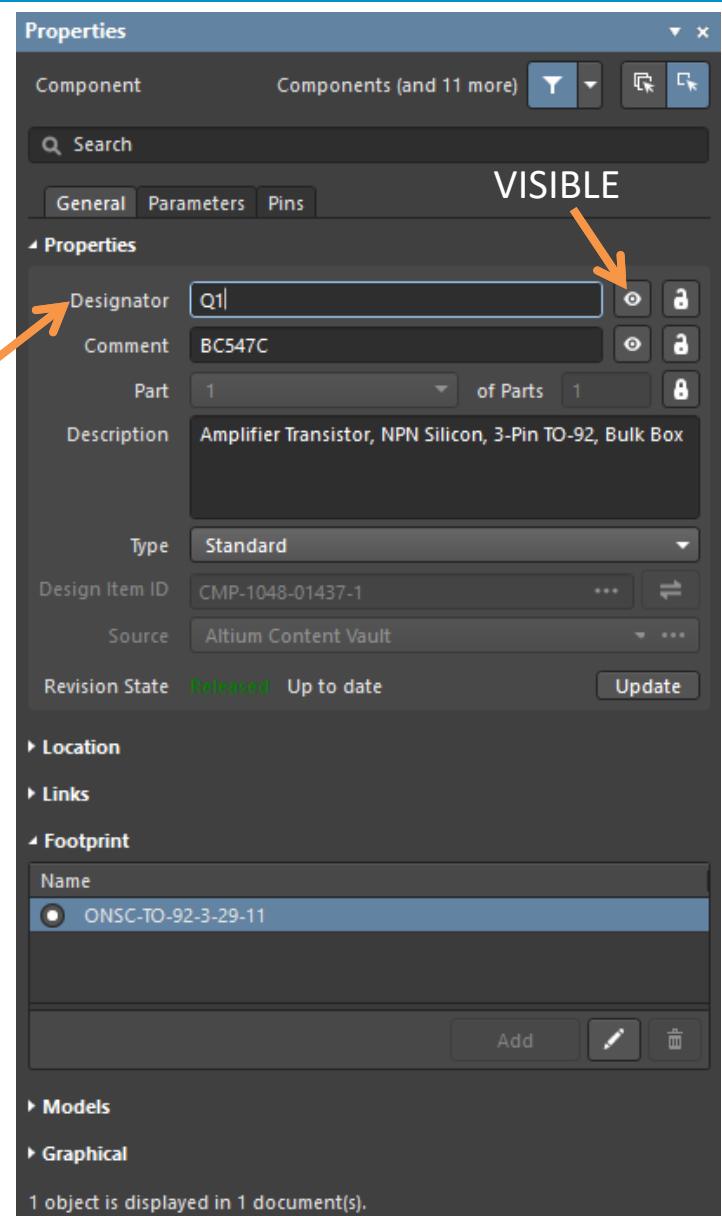
Multivibrator Parts

- The next step is to search the Altium Content Vault for the following components to use in the Multivibrator circuit.

DESIGNATOR	DESCRIPTION	LIB. COMPONENT NAME
Q1, Q2	NPN transistor (BC547 or 2N3904)	CMP-1048-01437-1
R1, R2	100K resistor, 5%, 0805	CMP-1013-00122-1
R3, R4	1K resistor, 5%, 0805	CMP-1013-00074-1
C1, C2	22nF capacitor, 10%, 16V, 0805	CMP-1036-04042-1
P1	2-pin header, thruhole	CMP-1024-00327-1

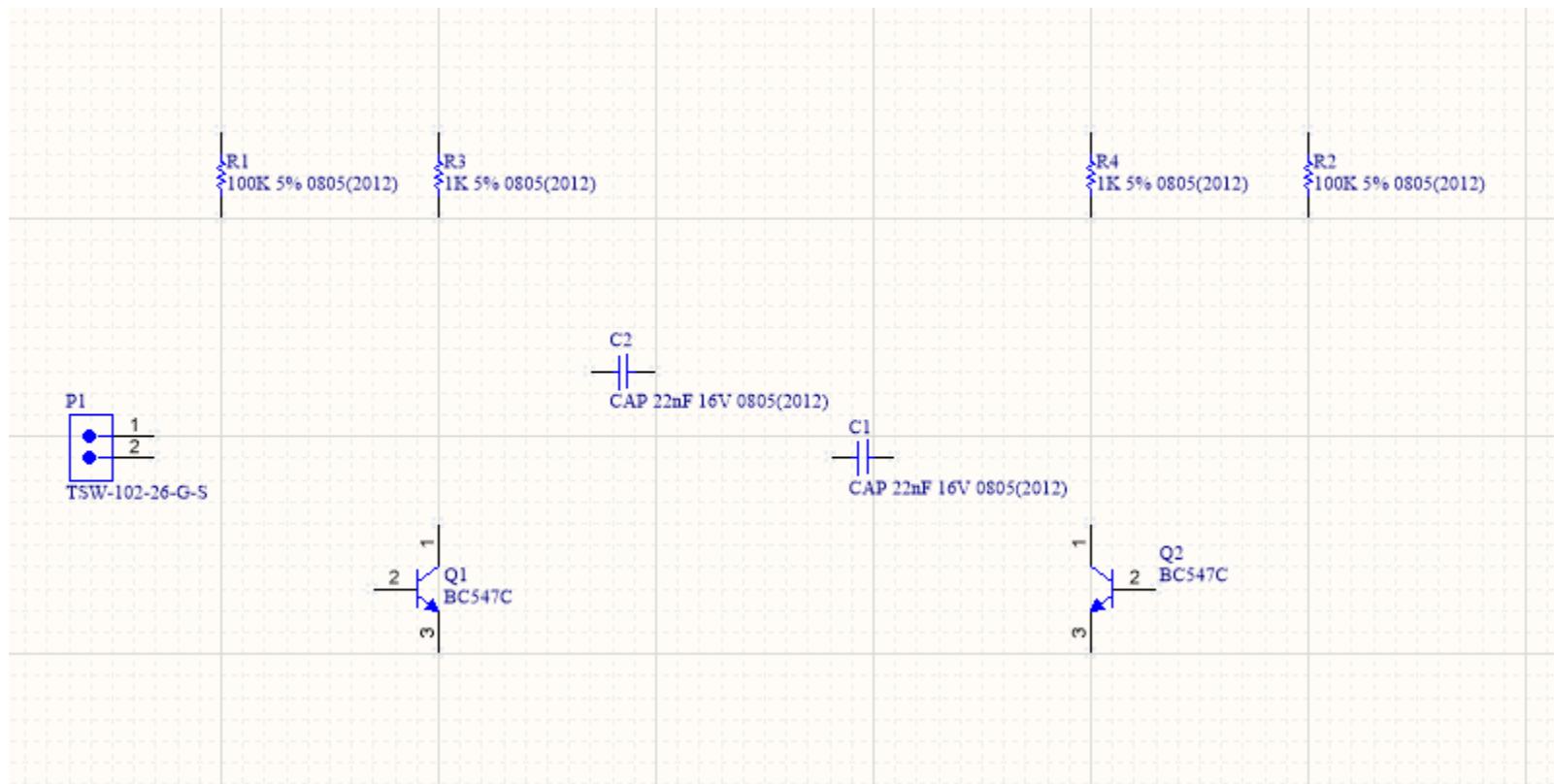
Placing Components on the Schematic

- To EDIT component properties: before placing the part on the schematic you can edit its properties, which can be done for any object floating on the cursor. For example, while the transistor is still floating on the cursor, press the **Tab** key to open the interactive *Properties* panel.
- Set the **Designator** to Q1, and the **Comment** to be **Visible**.
- Placement Tips:
 - **Spacebar** to rotate
 - **X** to flip (X-axis), **Y** to flip (Y-axis).



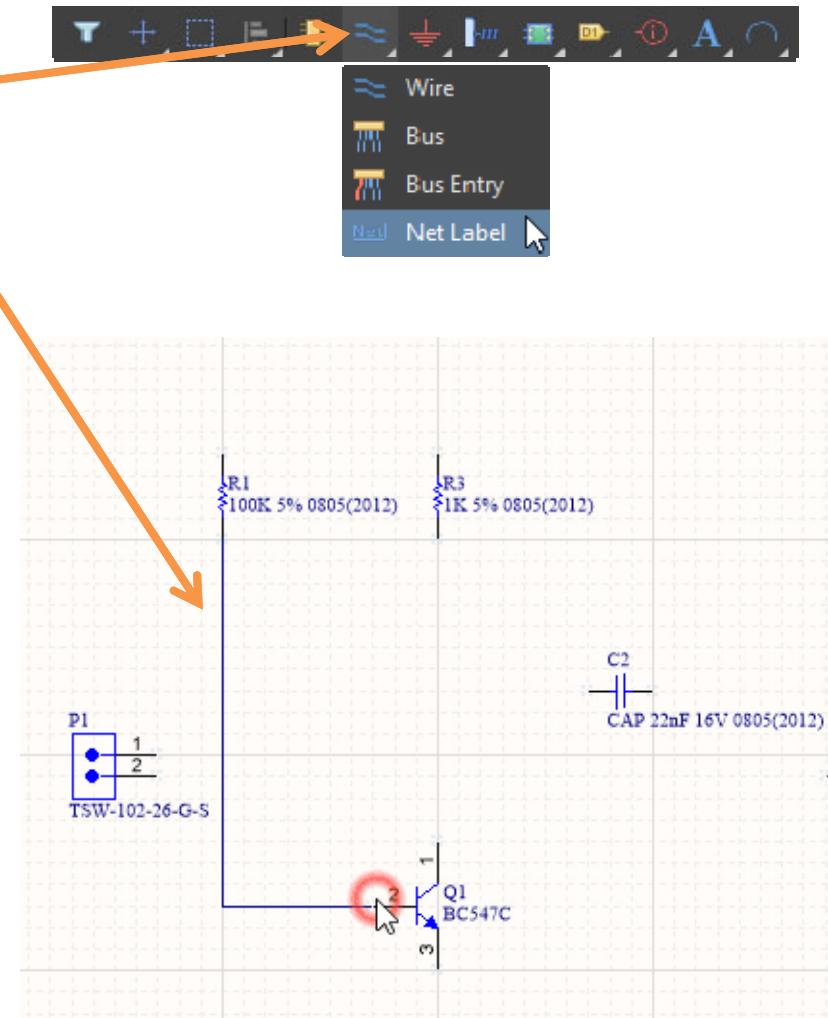
Placing Components on the Schematic

- Once you have placed the components, the schematic should look like the image below.



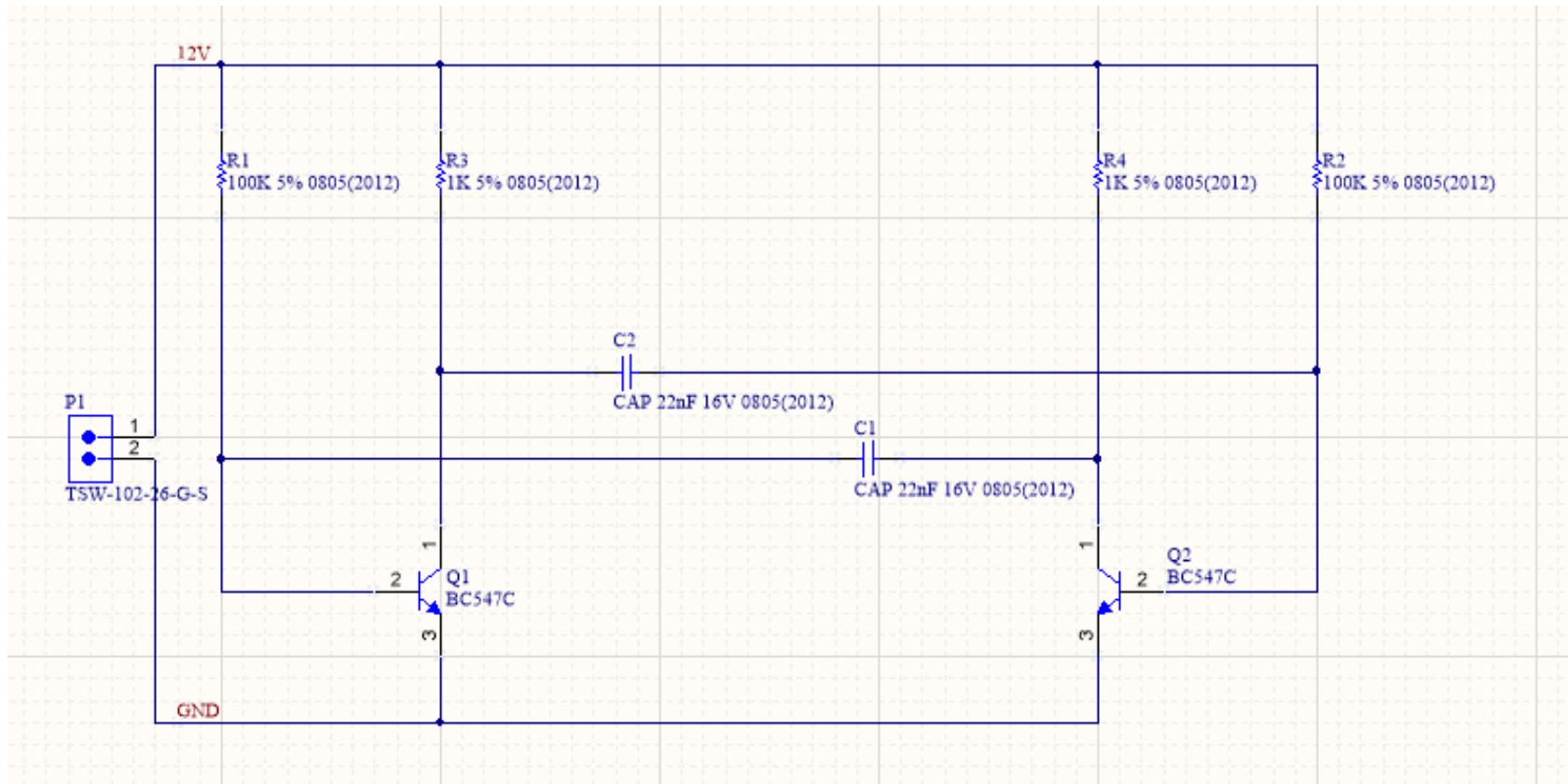
Wiring

- Click the button on the Active Bar **Place » Wire**
- Click the **Left Mouse Button** or press **Enter** to anchor the first wire point.
- Position the cursor over e.g. the base of Q1 and then click or press **Enter** to connect the wire to the base of Q1.
- When you have finished placing all the wires, **right-click** or press **ESC** to exit placement mode.

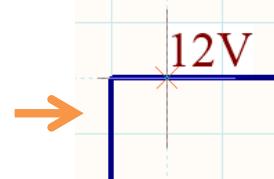


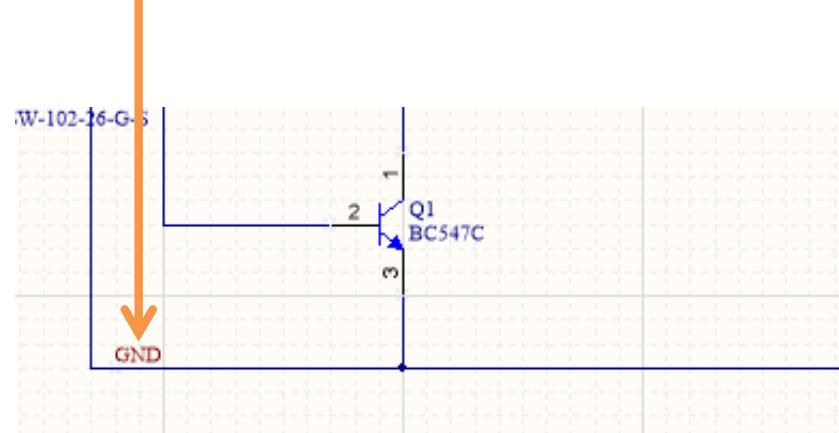
Wiring

- After wiring the schematic should look like the image below.



Nets and Net Labels

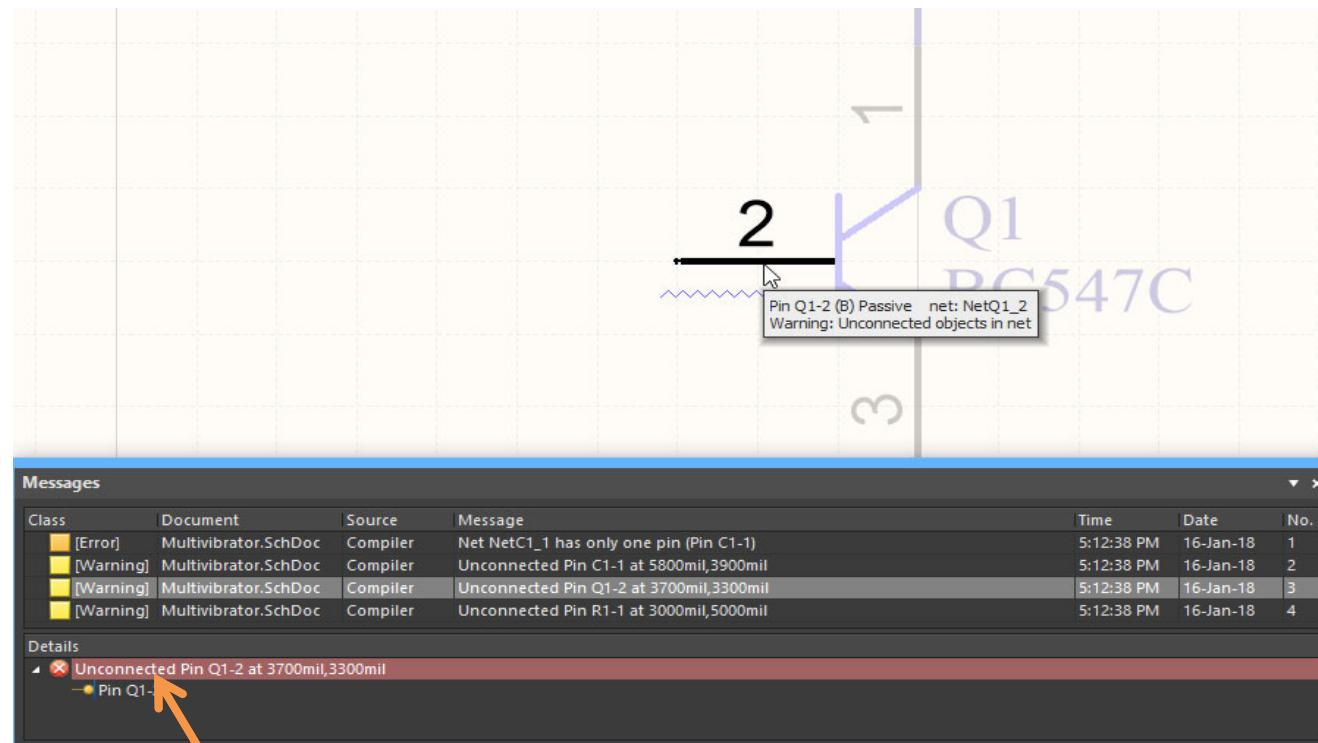
- Each set of component pins that you have connected to each other now form what is referred to as a *net*. For example, one net includes the base of Q1, one pin of R1 and one pin of C1.
- Click the  button (**Place » Net Label**). A net label will appear floating on the cursor.
- To edit the net label before it is placed, press **Tab** key to open the *Properties* panel.
- Type 12V in the **Net** field, then click the Pause button () to return to object placement.
- Place the net label over a wire (upper wire on the schematic). 
- Press **Tab** and type GND in the **Net Name** field and press **Enter** to return to object placement mode.
- Place the label on lower wire on the schematic.
- **Right-click** or press **ESC** to exit net label placement mode.



Check For Errors

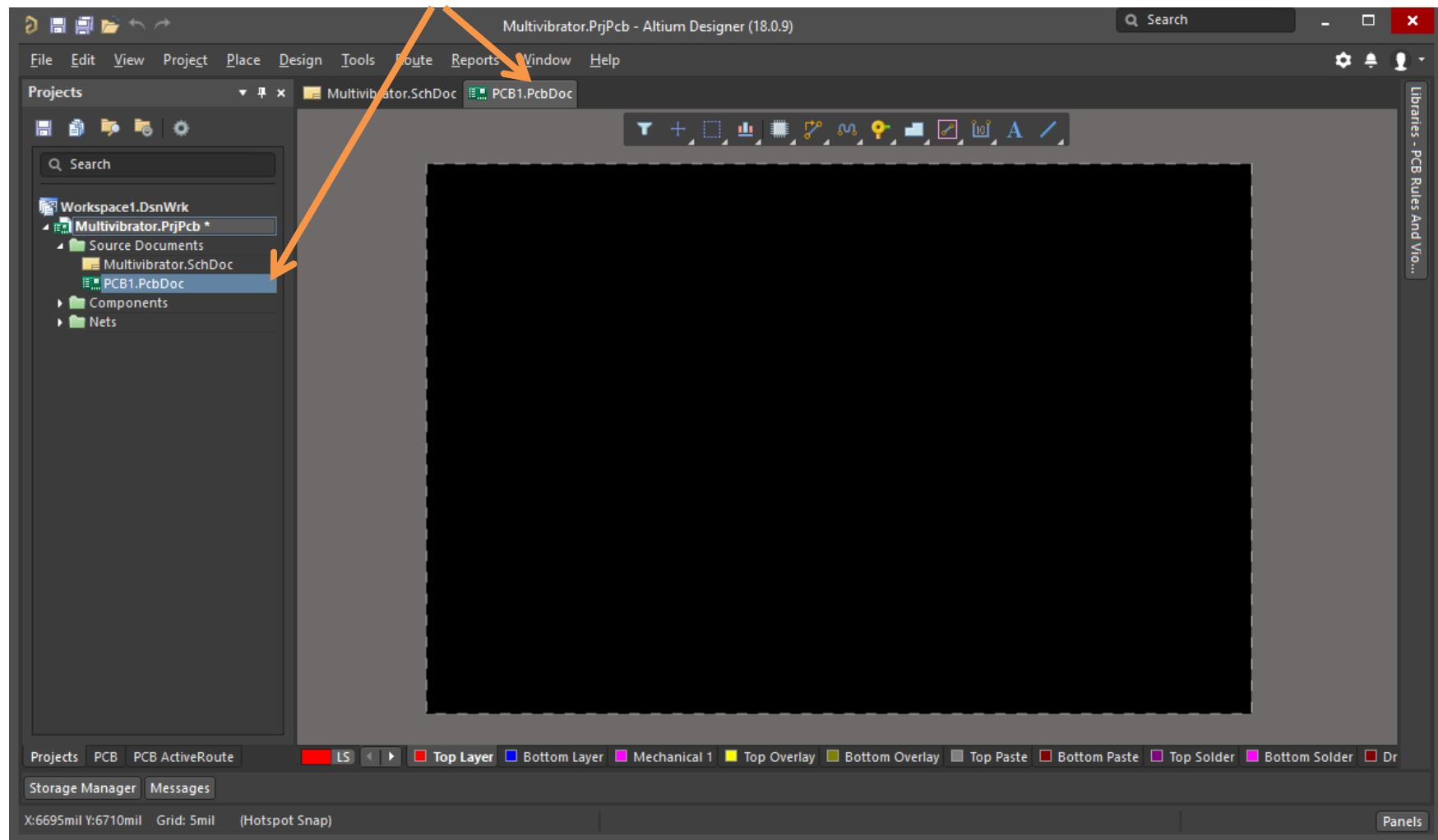
Compiling the Project to Check for Errors

- To compile the project and check for errors, select **Project » Compile PCB Project Multivibrator.PrjPcb**. Use the Messages panel to locate and resolve design warnings and errors - double-click on a warning/error to cross probe to that object, in case of any error.
- Once fixed error, it will be necessary to recompile the project (**Project » Recompile PCB Project Multivibrator.PrjPcb**)



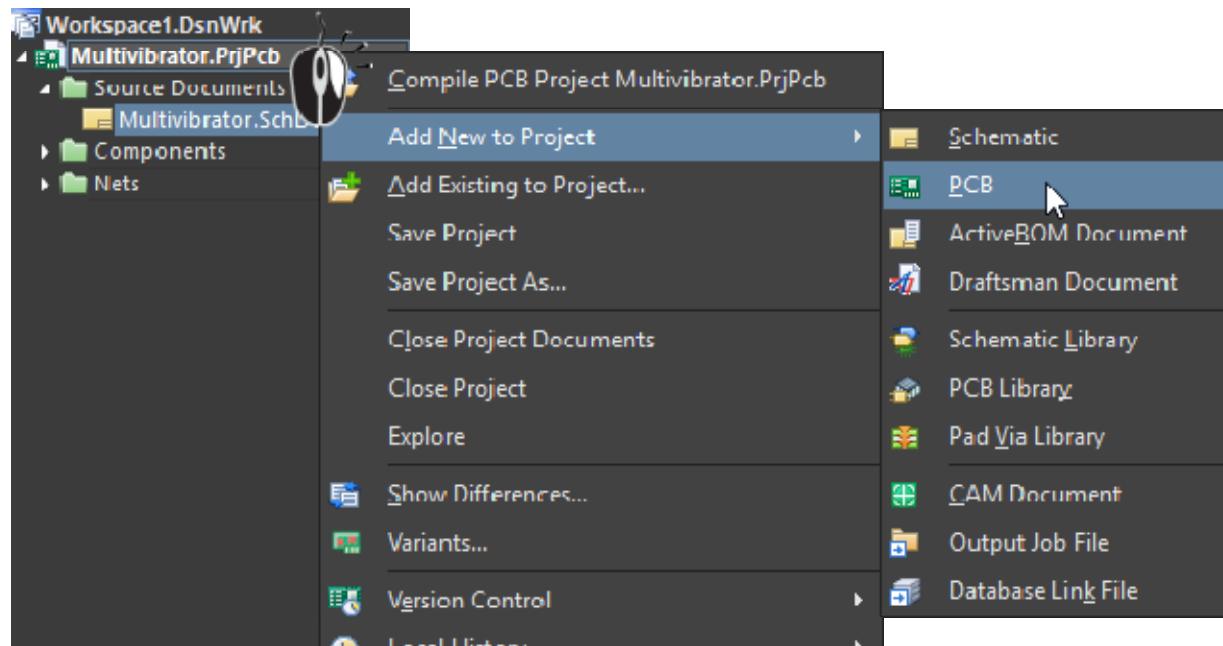
Creating a New PCB

- Before you transfer the design from the Schematic Editor to the PCB Editor, you need to create the blank PCB, then name and save it as part of the project.



Creating a New PCB

- A new PCB can be added to the project via the *Projects* panel **right-click** menu, select the **Add New to Project » PCB** .

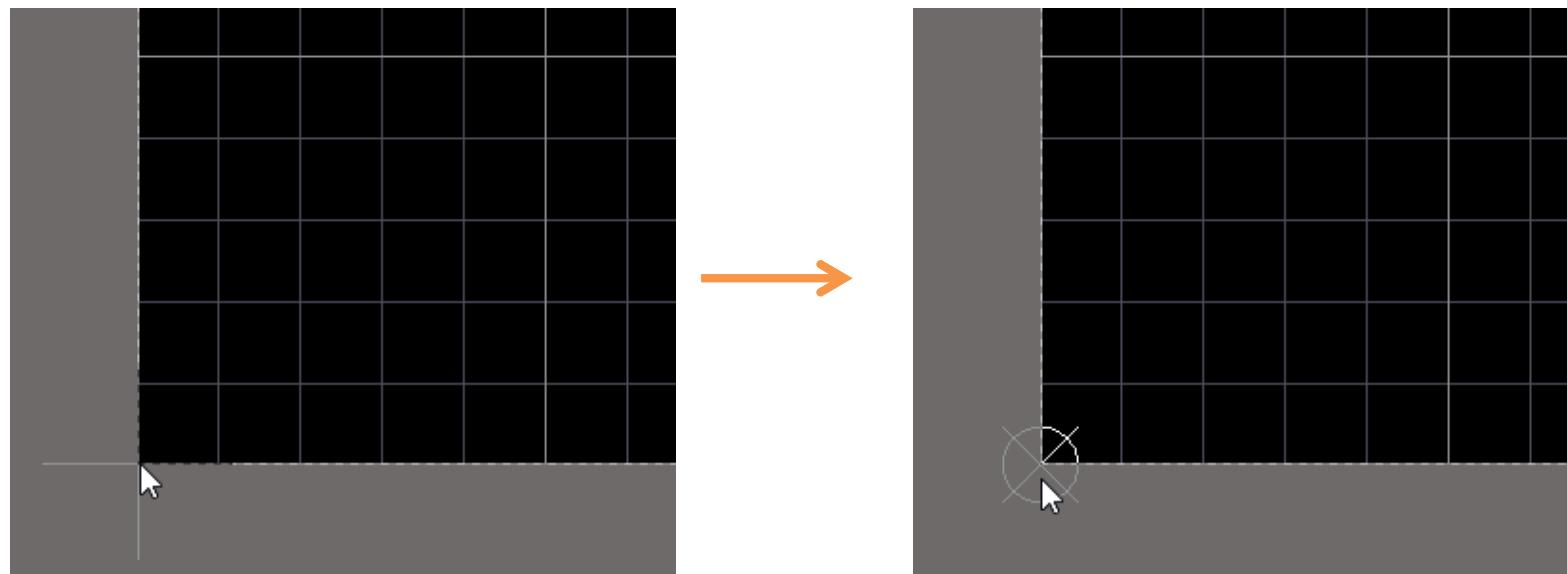


- The PCB will appear as a Source Document in the Project, as shown above. **Right-click** on the PCB icon in the *Projects* panel to select the **Save As** command, naming it Multivibrator.
- Adding the PCB has changed the project, so save the project too (**right-click** on the project filename in the *Projects* panel, and select **Save Project**).

Configuring the Board Origin

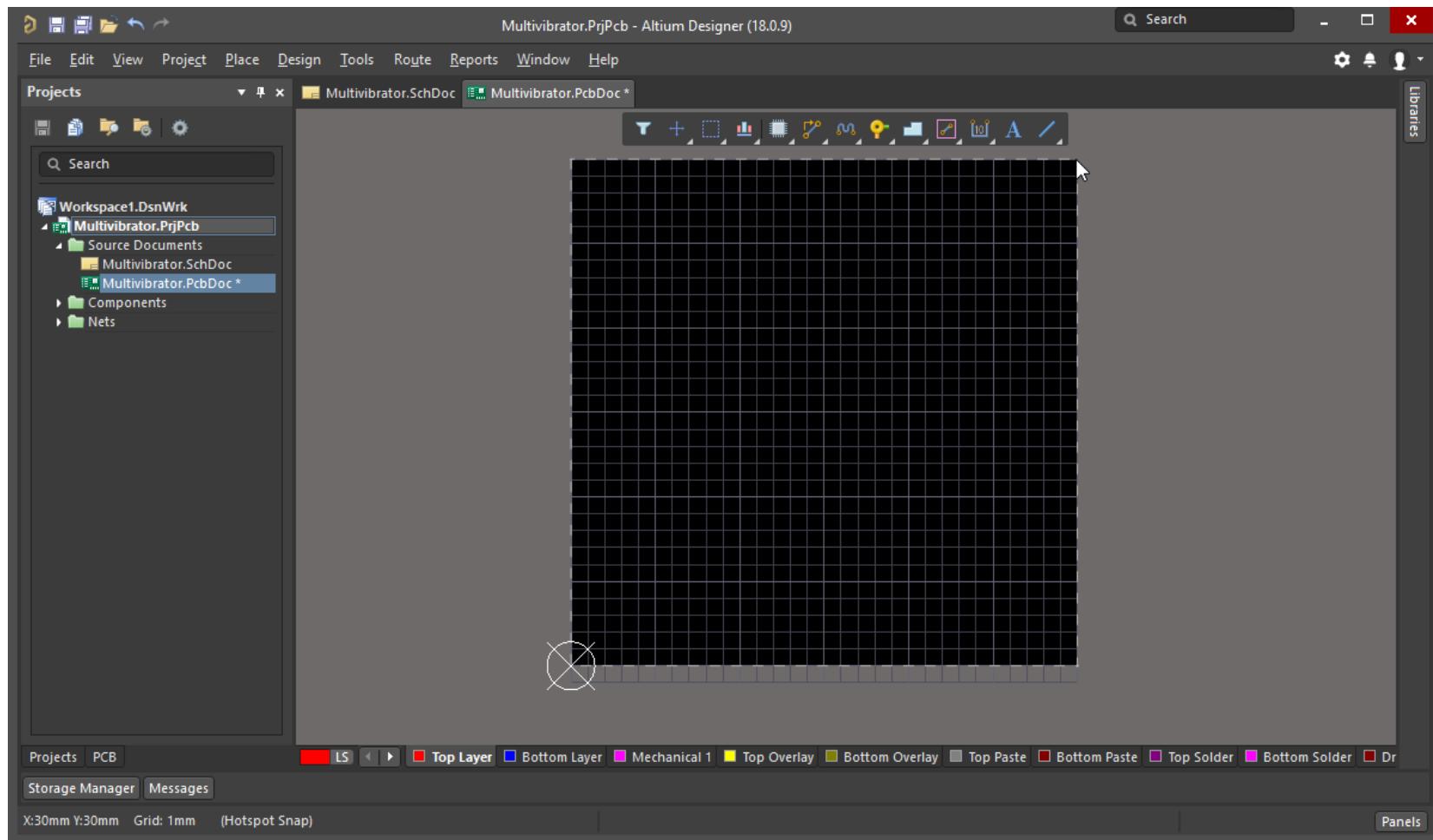
Setting the Origin and the Grid

- There are two origins used in the software, the Absolute Origin, which is the lower left of the workspace, and the user-definable Relative Origin, which is used to determine the current workspace location. Before setting the origin, Keep zooming in to the lower left of the current board shape until you can easily see the grid - to do this position the cursor over the lower-left corner of the board shape and press **PgUp** until both the Coarse and Fine grids are visible, as shown in the images below.
- To set the Relative Origin, select **Edit » Origin » Set**, position the cursor over the bottom left corner of the board shape, then **left click** to locate it.



Configuring the Board Grid

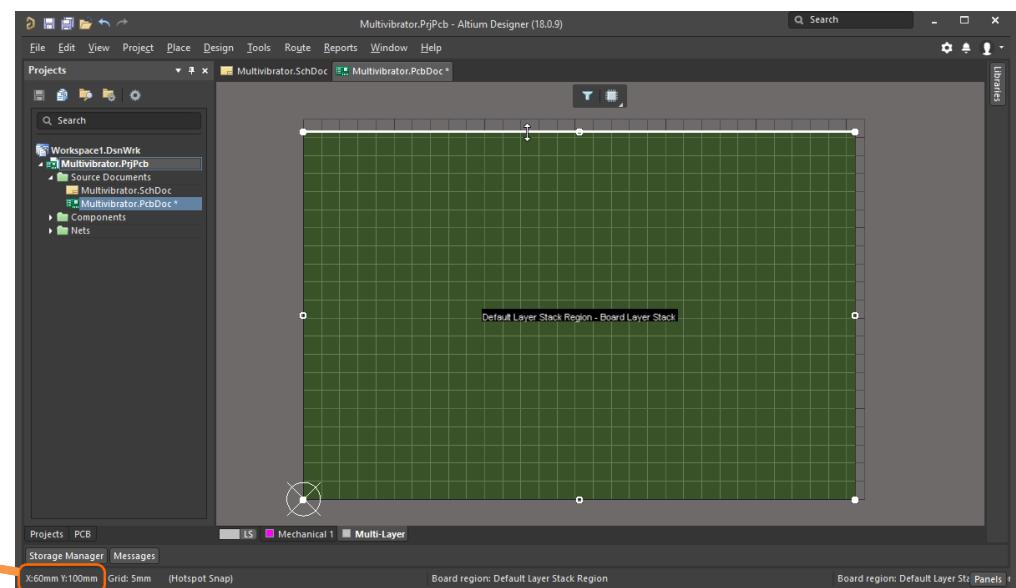
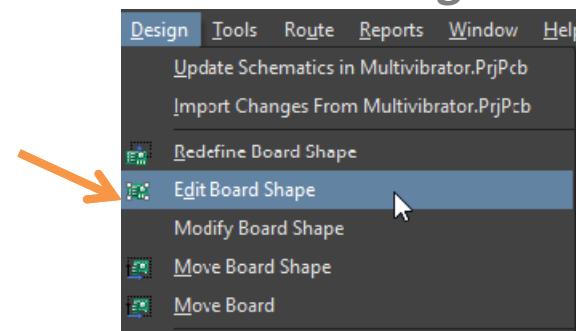
- The next step is to select a suitable snap grid: for this tutorial it will be using a Metric grid. A coarse 5mm grid will be suitable for component placement, press **Ctrl+Shift+G** to open the *Snap Grid* dialog and enter **5mm**, then click **OK** to close the dialog.



Configuring the Board Shape

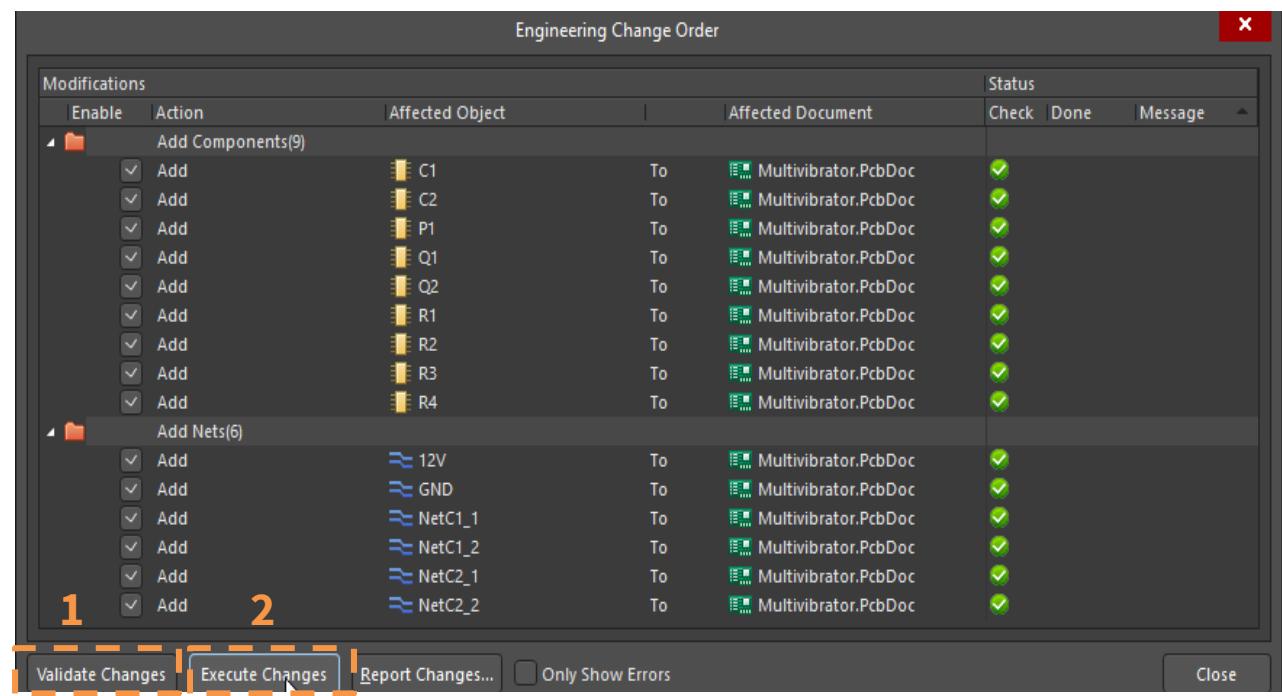
Redefining the Board Shape

- To manipulate the size you need to be able to see the edges of the board, use **Ctrl+WheelRoll** to zoom out a bit more, or press **PgDn**.
- The next step is to change the board shape. To do this select **View » Board Planning Mode** to change (shortcut: **1**). The display will change, the board area will now be shown in green.
- To edit select **Design » Edit Board Shape** from the menu.
- The resize cursor is shown, use the location information on the **Status bar** to help you resize the board to 30mm x 30mm.



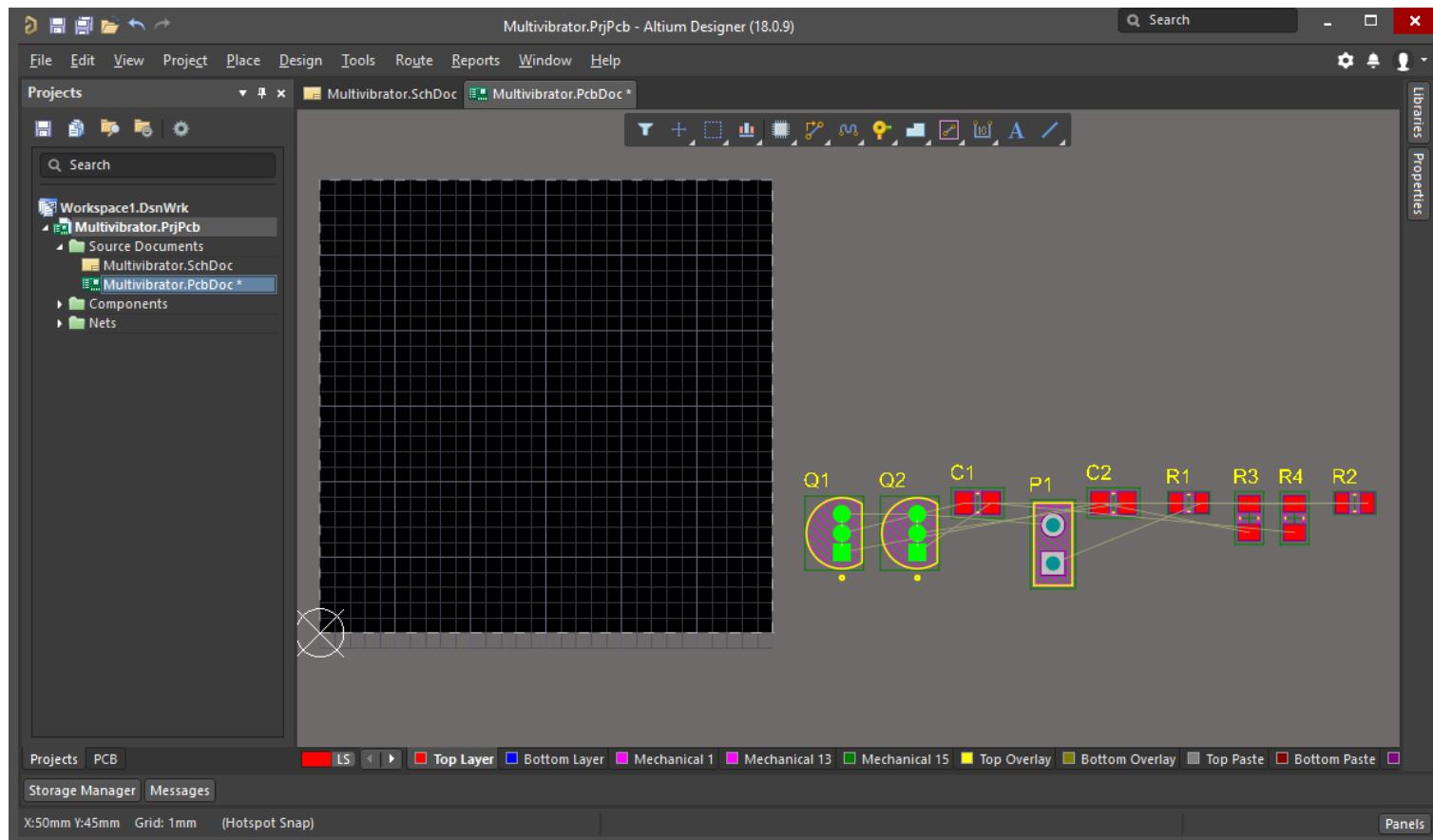
Transferring the Design to PCB Layout

- Select **Design » Update PCB Document Multivibrator.PcbDoc** from the Schematic editor menus.
- Click on **Validate Changes (1)**. If all changes are validated, a green tick will appear next to each change in the **Status** list. If the changes are not validated, close the dialog, check the **Messages** panel and resolve any errors.
- If all changes are validated, click on **Execute Changes (2)** to send the changes to the PCB editor.
- When completed, the target PCB opens with the *Engineering Change Order* dialog open on top of it, and the **Done** column entries will be ticked.
- Click to **Close** the dialog and complete the transfer process.



Transferring the Design to PCB Layout

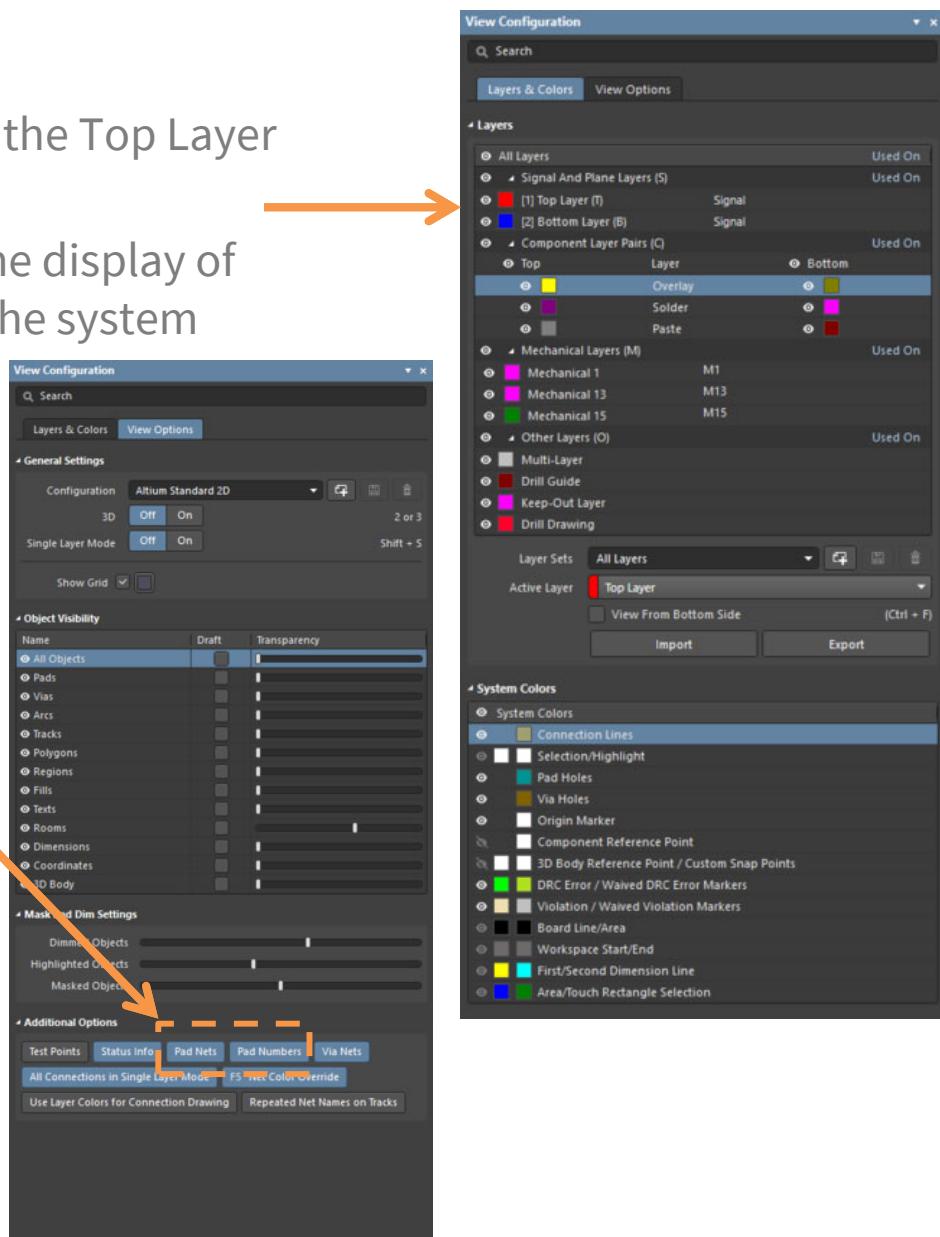
- When completed, the components are placed outside the board shape and the nets are created.



Configuring the Layer Visibility

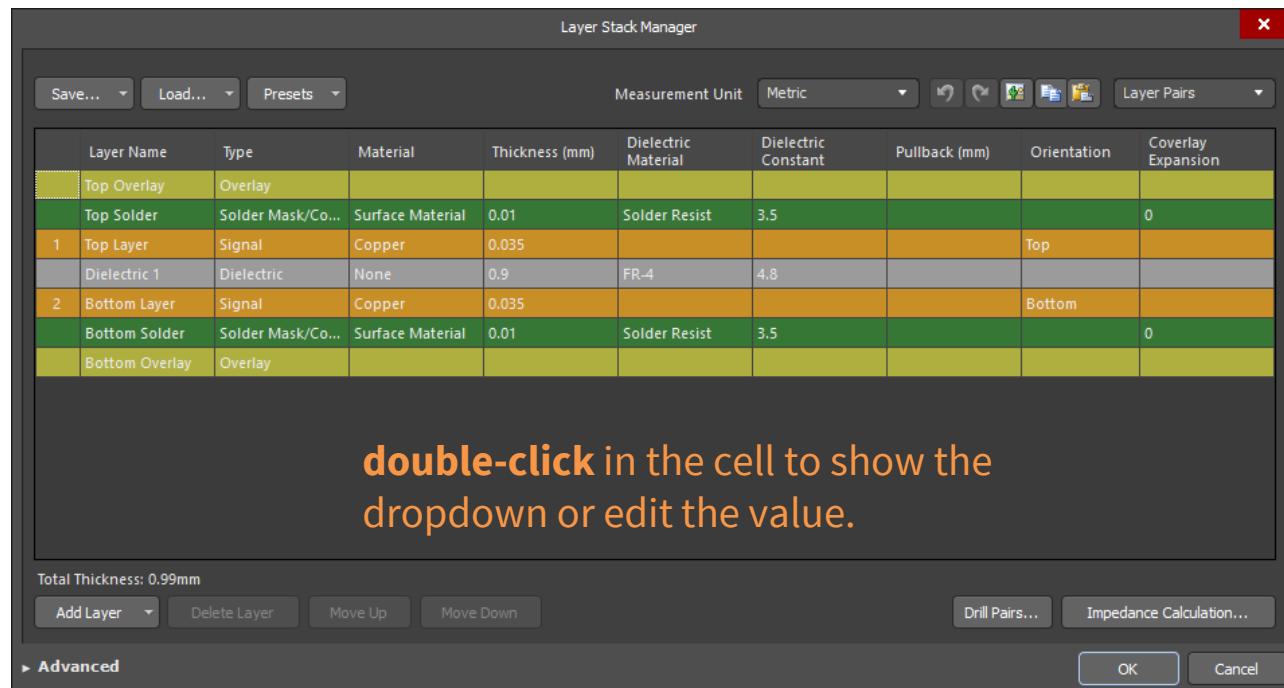
- Open the *View Configuration* panel.
- In the **Layers and Colors** tab, confirm that the Top Layer and Bottom Layer signal layers are visible.
- Note that this panel is where you control the display of the mask layers, the silkscreen layers and the system layers, such as DRC and grids.

- Switch to the **View Options** tab.
- Confirm that the **Pad Nets** and **Pad Numbers** options are enabled.



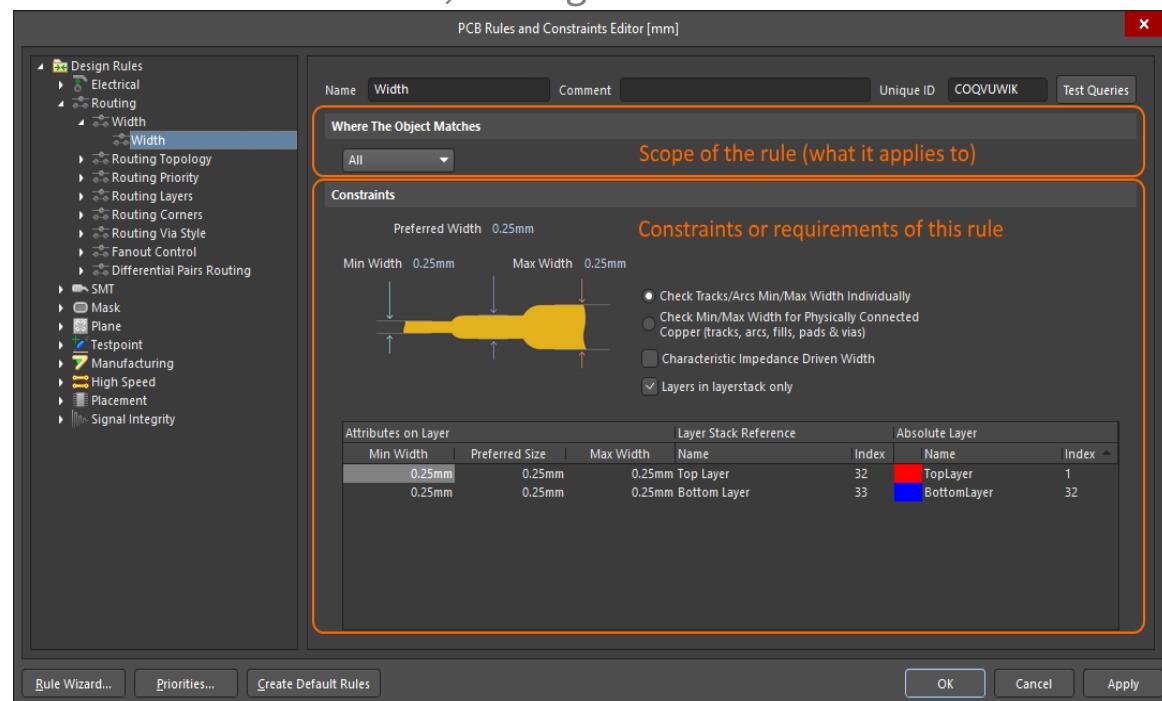
Physical Layers and the Layer Stack Manager

- The properties of the physical layers are defined in the **Layer Stack Manager**.
- The *Layer Stack Manager* dialog is used to:
 - Add / remove signal and power plane layers.
 - Add / remove dielectric layers.
 - Change the order of the layers.
 - Configure the Material type for the physical layers.
 - Set the layer Thickness, Dielectric Material and Dielectric Constant.



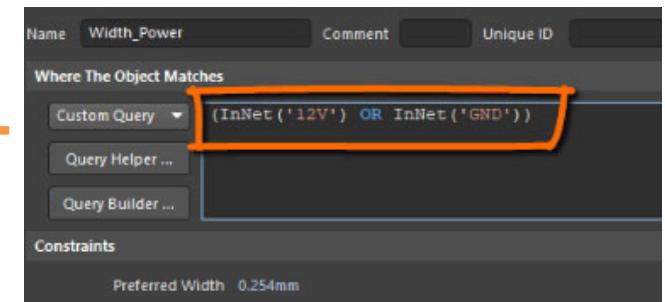
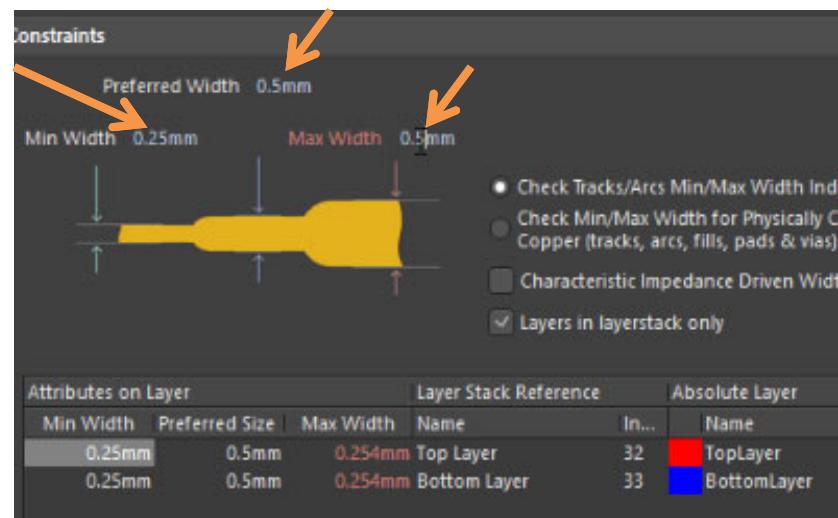
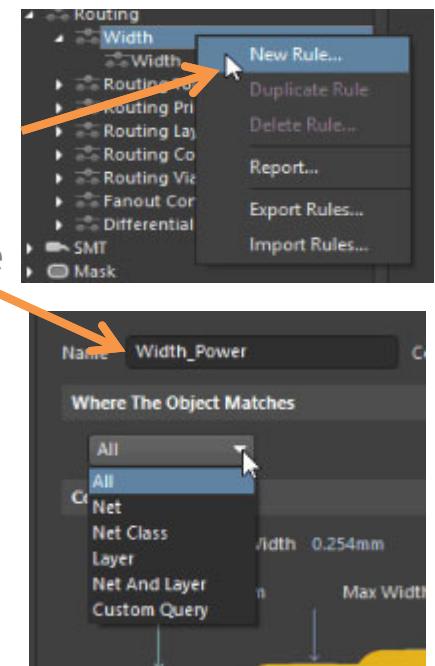
Routing Width Design Rules : Signal Nets

- With the PCB as the active document, open the **PCB Rules and Constraints Editor**.
- Each rules category is displayed under the **Design Rules ->Routing->Width** to display the currently defined width rules.
- The right hand side of the dialog displays the settings for that rule, including: the rule's **Where the First Object Matches** in the top section with the rule's **Constraints** below that.
- Since this rule is to target the majority of nets in the design (the signal nets), confirm that the **Where the First Object Matches** setting is set to **All**. An additional rule will be added to target the power nets.
- Edit the **Min Width, Preferred Width & Max Width** values, setting them to 0.25mm.
- Click **Apply** to save it and keep the dialog open.



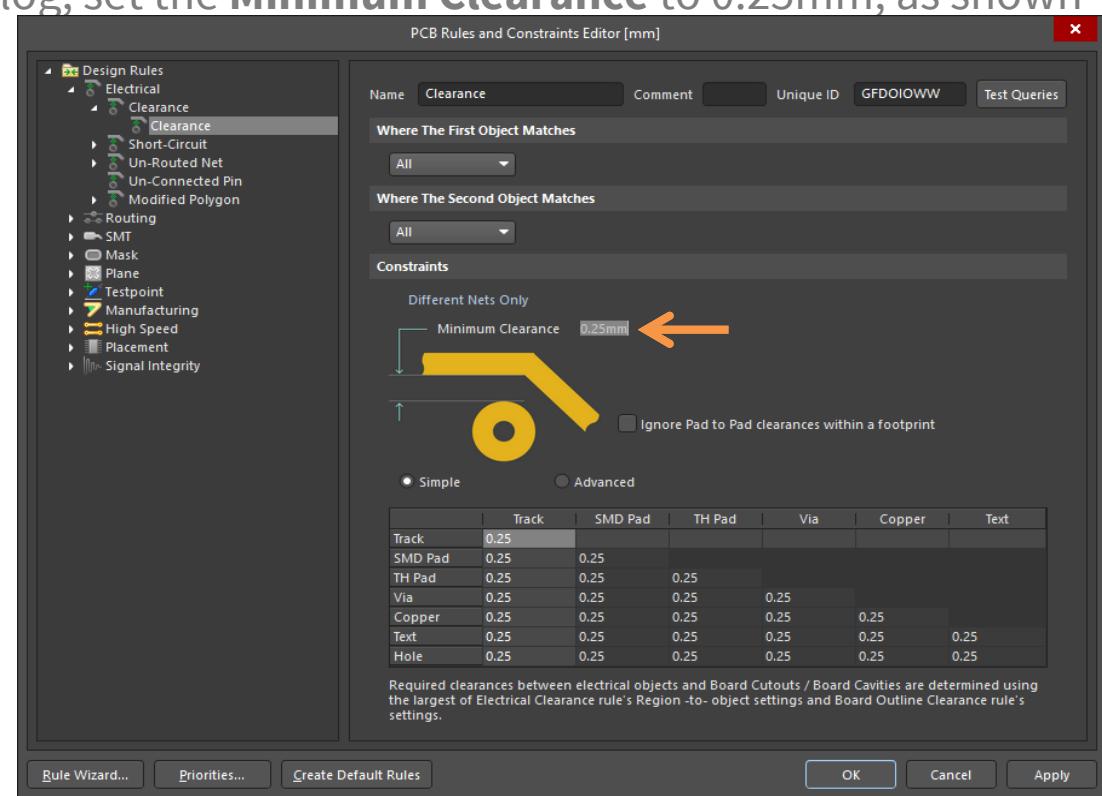
Routing Width Design Rules : Power Nets

- The next step is to add another design rule to specify the routing width for the power nets.
- With the existing **Width** rule selected in the **Design Rules** > right-click and select **New Rule** to add a new **Width** constraint rule.
- A new rule named **Width_1** appears. Click on the new rule and enter the name **Width_Power** in the field.
- Click the **Query Builder** button to open the **Query Builder**, the configure it to target objects: **InNet('12V')** OR **InNet('GND')** and the **OK**.
- The last step is to edit the **Min Width / Preferred Width / Max Width** values 0.25 / 0.5 / 0.5 to allow power net routing widths in the range 0.25mm to 0.5mm, as shown below.
- Click **Apply** to save the rules and keep the dialog open.



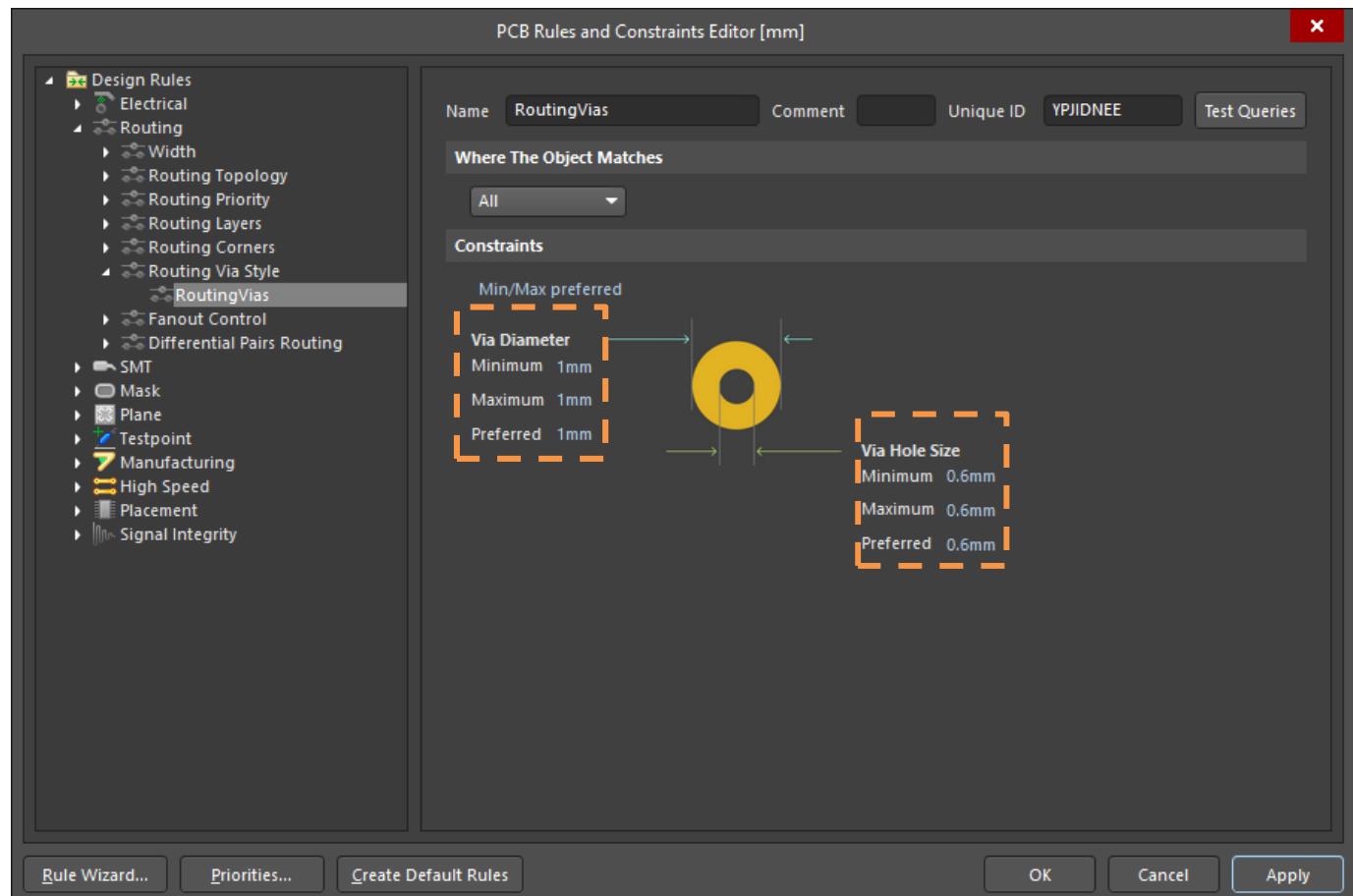
Defining the Electrical Clearance Constraint

- The next step is to define how close electrical objects that belong to different nets, can be to each other.
- Expand the **Electrical** category in the tree of Design Rules, then expand the **Clearance** rule-type.
- This requirement is handled by the **Electrical Clearance Constraint**, for the tutorial a clearance of 0.25mm between all objects is suitable.
- In the Constraints region of the dialog, set the **Minimum Clearance** to 0.25mm, as shown in the image above.
- Click **Apply** to save the rule and keep the dialog open.



Defining the Routing Via Style

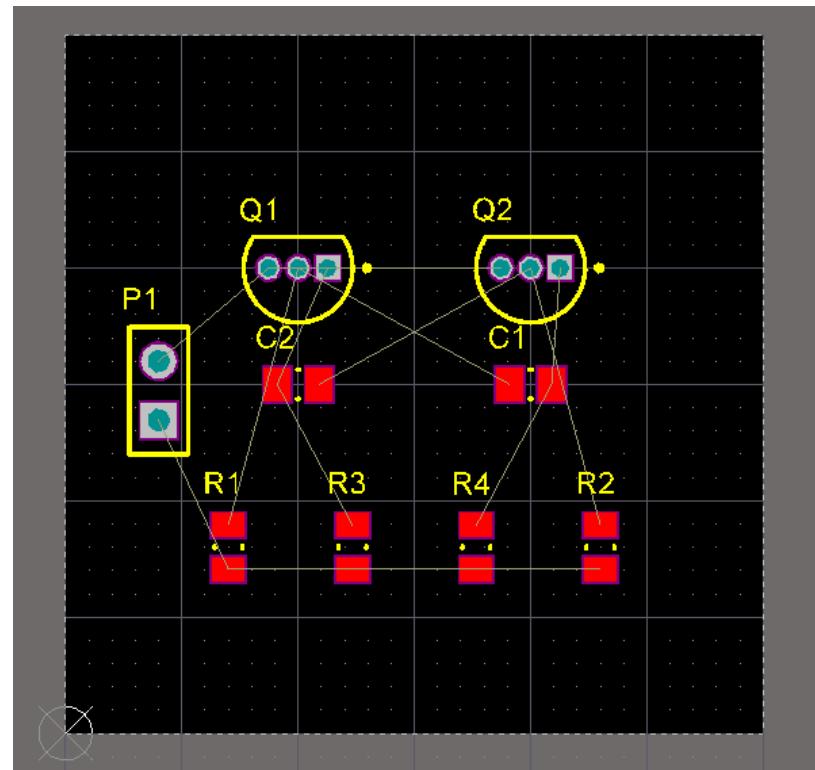
- Expand the Design Rule tree and select the default **RoutingVias** design rule.
- For this tutorial edit the rule settings to **Via Diameter** = 1mm and a **Via Hole Size** = 0.6mm. Set all fields (Min, Max, Preferred) to the same size.
- Click **OK** to close the *PCB Rules and Constraints Editor*.
- Save the PCB file.



Position and Placement

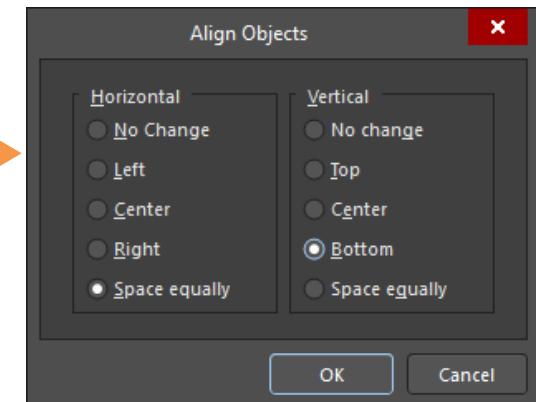
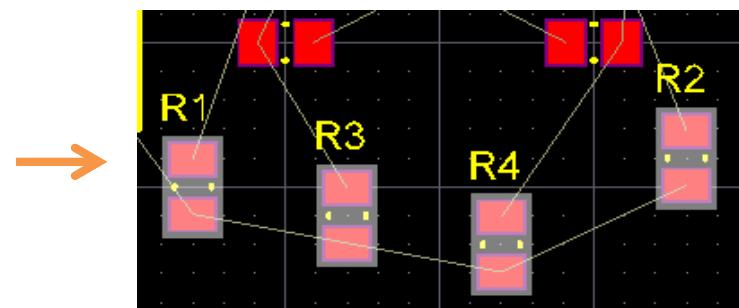
Positioning Components

- Now we can position the components in suitable locations on the board.
- To move a component, either:
- **Click-and-Hold** the left mouse button on the component, move it to the required location, rotate it with the **Spacebar**, then release the mouse button to place it; or
- Run the **Edit » Move » Component** command, then single click to pick up a component, move it to the required location, rotate if required, then click once to place it. When finished, right-click to drop out of the **Move Component** command.

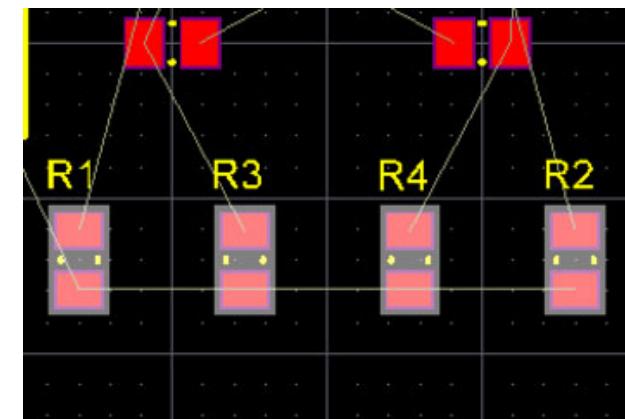


Position and Placement

- It is also possible to **align** the components.
To do this:
- Holding the **Shift** key, click on each of the four resistors to select them.



- Then **Right-click** on any of the selected components and choose **Align » Align** to open the *Align Objects* dialog.

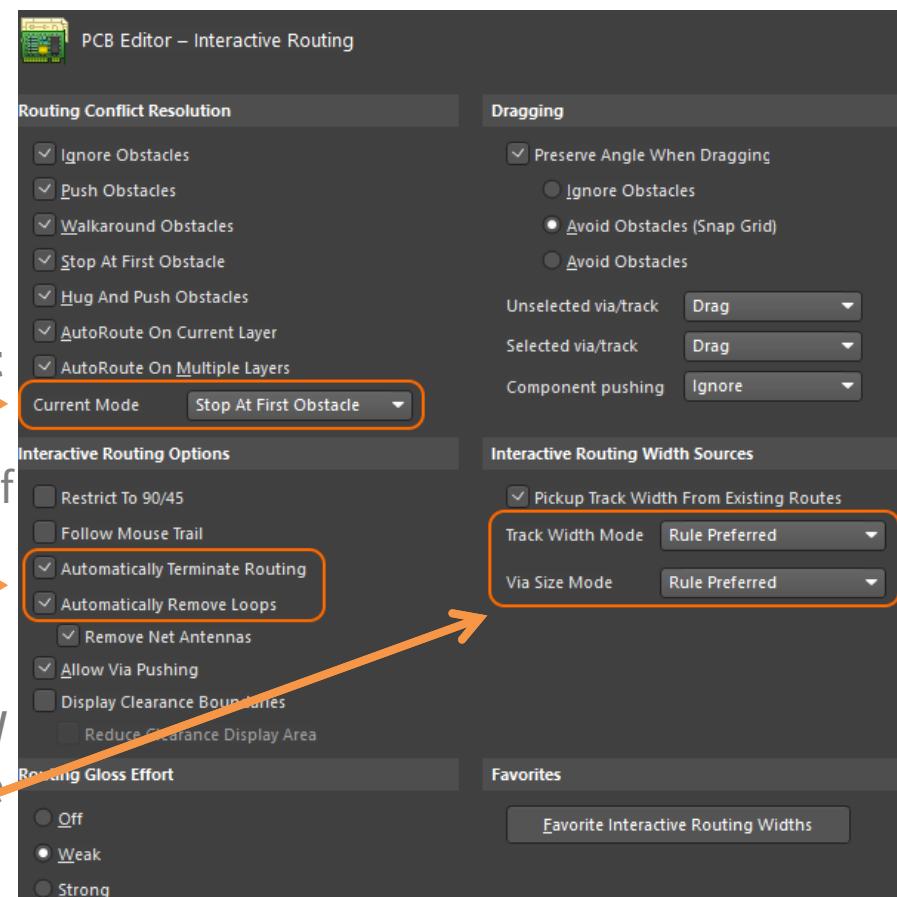


- Select **Space Equally** in the **Horizontal** section and **Bottom** in the **Vertical** section, then click **OK** to apply these changes. The four resistors are now aligned and equally spaced.

Interactively Routing the Board

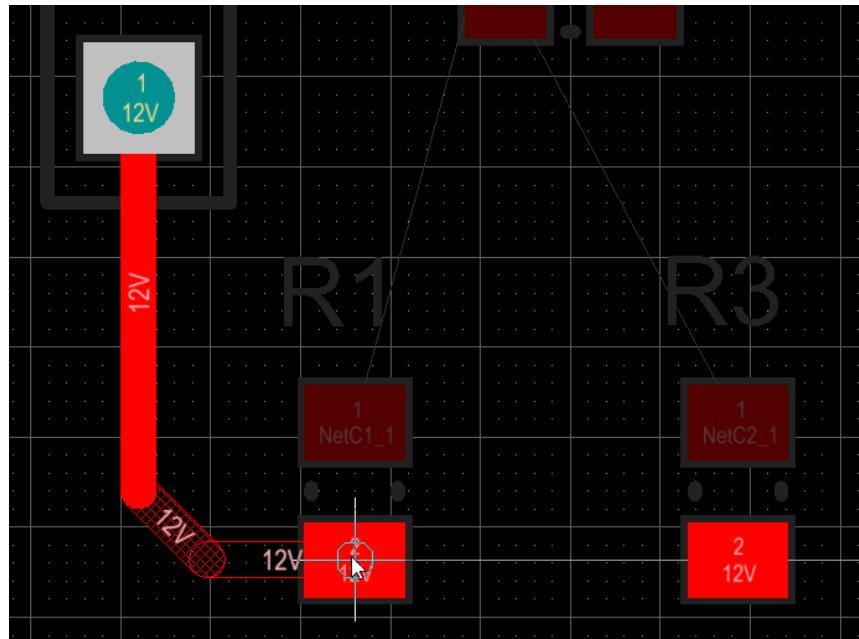
Preparing for Interactive Routing

- Before starting to route, it is important to configure the Interactive Routing options found in the **PCB Editor - Interactive Routing** page of the *Preferences* dialog.
- Set the **Routing Conflict Resolution Current Mode** to Stop at First Obstacle.
- In the **Interactive Routing Options** section of the page, confirm that the **Automatically Terminate Routing** and the **Automatically Remove Loops** options are enabled.
- Confirm that the **Interactive Routing Width / Via Size Sources** options are both set to **Rule Preferred**.



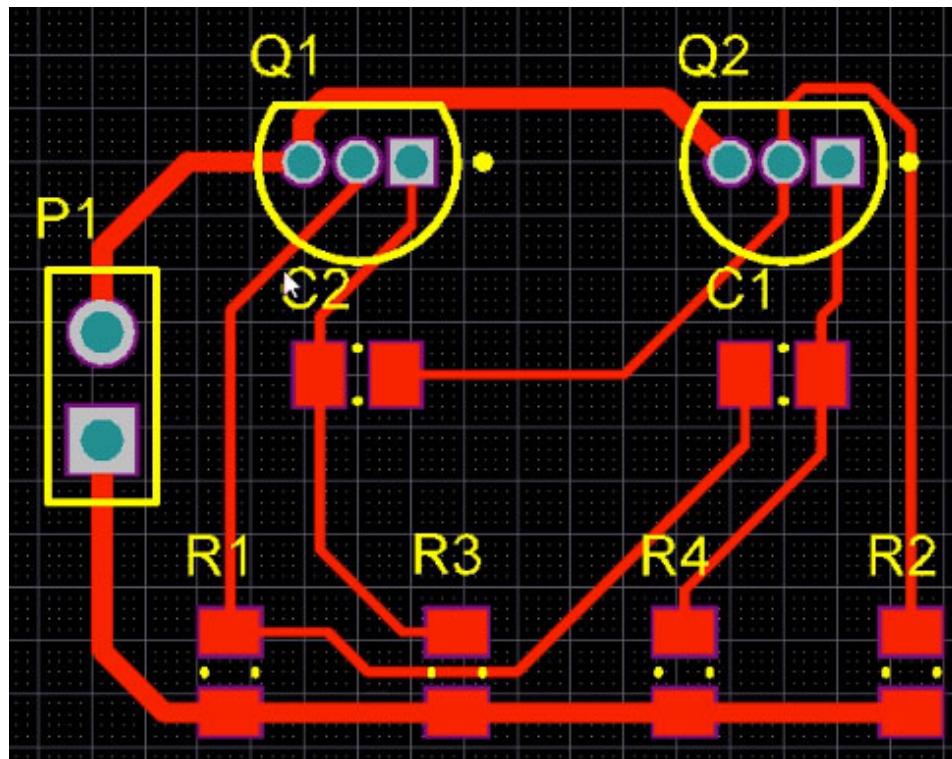
Time to Route

- Check which layers are currently visible by looking at the Layer Tabs at the bottom of the workspace.
- press **Shift+S** to toggle to in and out of single layer mode.
- Interactive routing is launched by clicking the **Route** button , or selecting the routing command, **Route » Interactive Routing** (shortcut: **Ctrl+W**).
- **Left-Click** or press **Enter** to anchor the first point of the track.
- Manually route by **Left-Clicking** to commit track segments, press **Backspace** to cancel the last-placed segment.



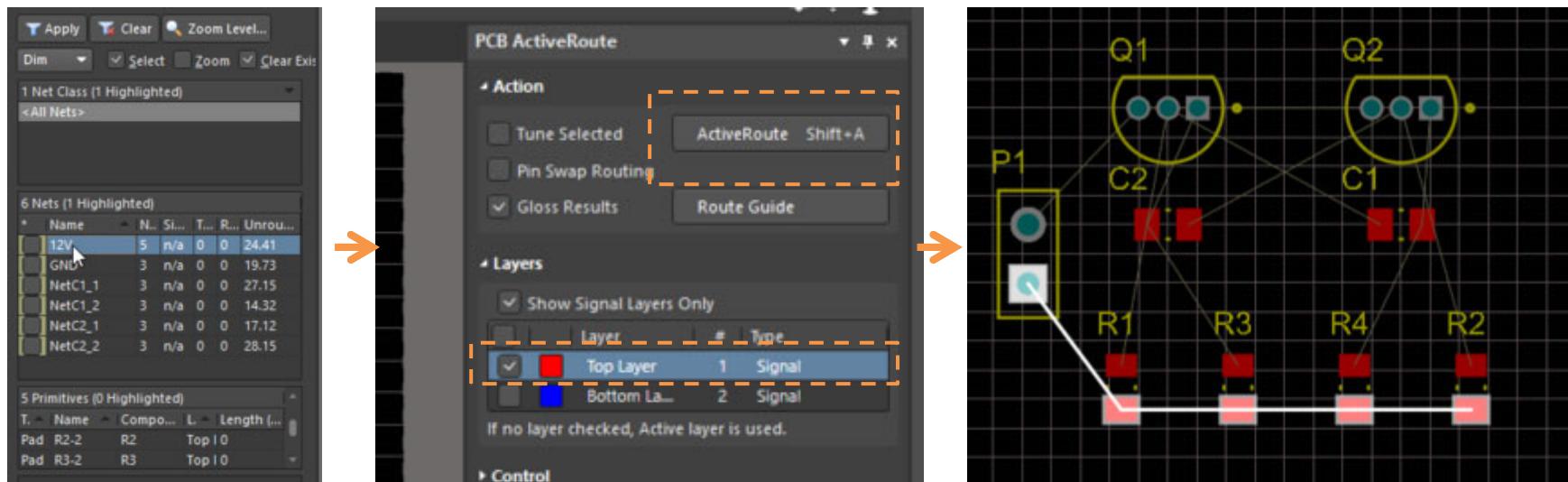
Time to Route

- Rather than routing all the way to the target pad, you can also press **Ctrl+Left Click** to use the *Auto-Complete* function and immediately route the entire connection.
- **Save** the design when you are finished routing.



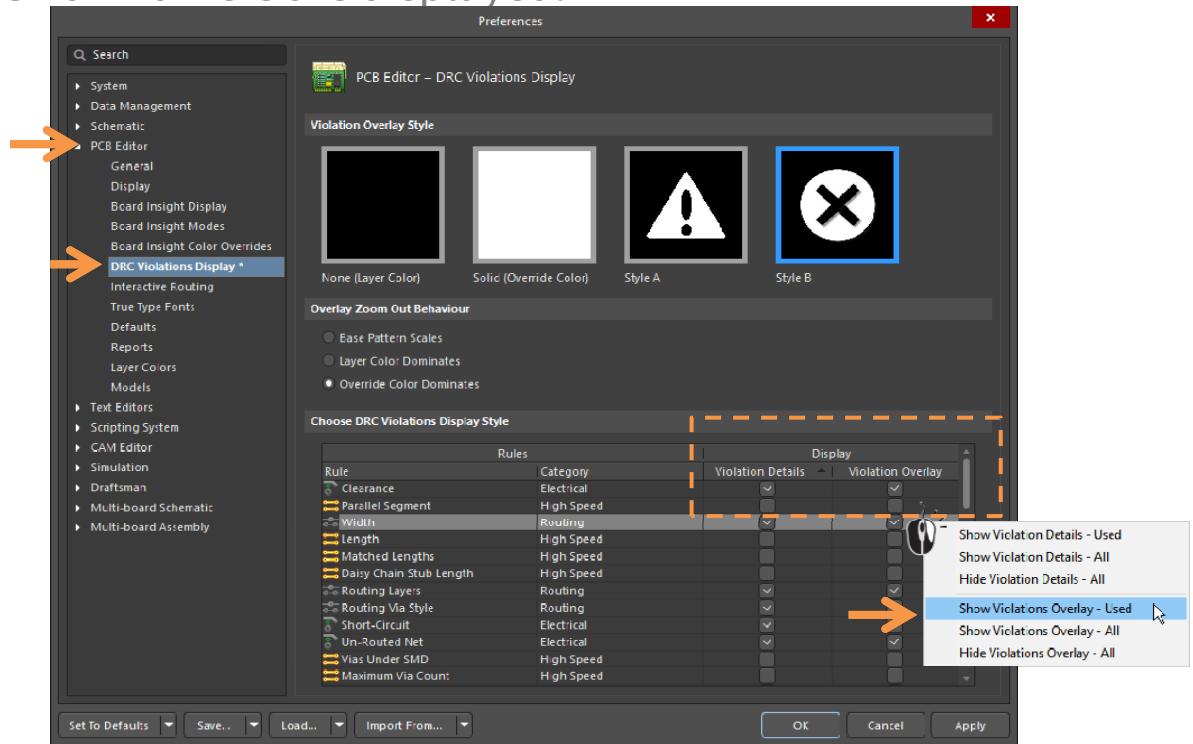
Active Route: Automated Interactive Router

- Another approach to routing the nets on your board is to use **ActiveRoute**, Altium's automated interactive router.
- What does that mean? It means you select the connection or connections to route, choose the layer, and run ActiveRoute. ActiveRoute has efficient multi-net routing algorithms, these are applied to the specific nets or connections that you have selected.
- ActiveRoute has been developed for dense boards using high pin count components, to help accelerate what can be a difficult and time-consuming routing process.
- For example:
- In the list of nets in the panel, click on the **12V net** name.
- In the PCB ActiveRoute panel, enable the **Top Layer**.
- Click the **ActiveRoute** button at the top of the panel, the 12V net will be routed.



Design Rule Check (DRC)

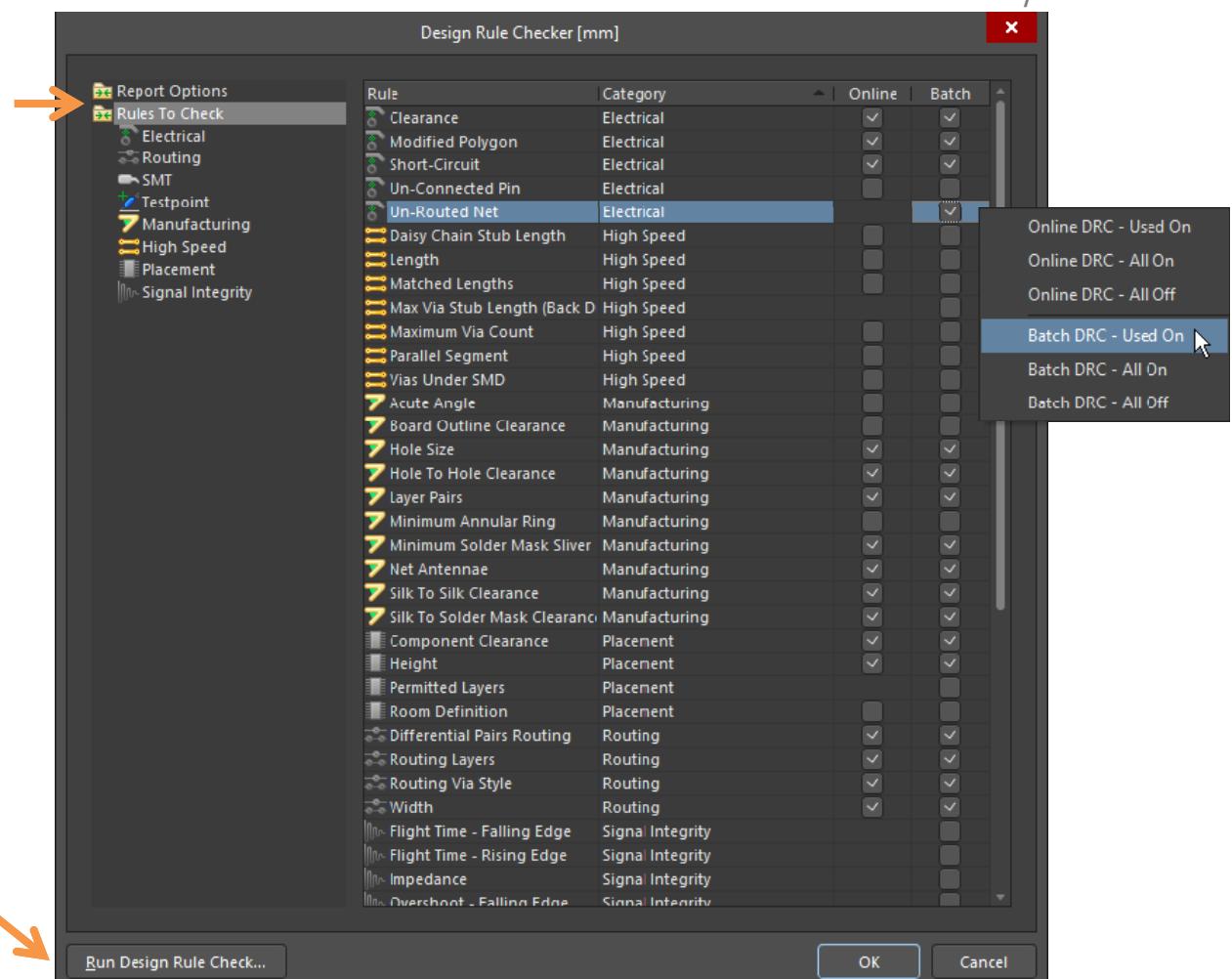
- Select »View Configuration« (shortcut: L) and confirm that the **DRC Error** visibility option is enabled, so that DRC error markers are displayed.
- Confirm that the **Online DRC** (Design Rule Checking) system is enabled.



- **PCB Editor – General ->Preferences->PCB Editor -> DRC Violations Display.**
- Right-click in the **DRC Violations Display** area of the **PCB Editor - DRC Violations Display** page and select **Show Violation Details - Used**, then right-click again and select **Show Violation Overlay – Used**.
- You are now ready to check the design for errors.

Configuring the Rule Checker

- Run the **Tools » Design Rule Check**
- Specific rules -> **Rules to Check**
- For most rule types there are checkboxes for **Online** and **Batch**: Click to enable/disable the rules as required.
- The **online DRC** feature will monitor the enabled rules as you work and immediately highlight any detected design violations.
- To run DRC click on **Run Design Rule Check**



Running a Design Rule Check (DRC)

- The *Messages* panel will appear, listing all detected errors.
- Example :

Design Rule Verification Report



Design Rule Verification Report

Date: 18-Jan-18
Time: 9:11:02 PM
Elapsed Time: 00:00:00
Filename: D:\Design\Multivibrator\Multivibrator.PcbDoc

Warnings: 0
Rule Violations: 18

Summary

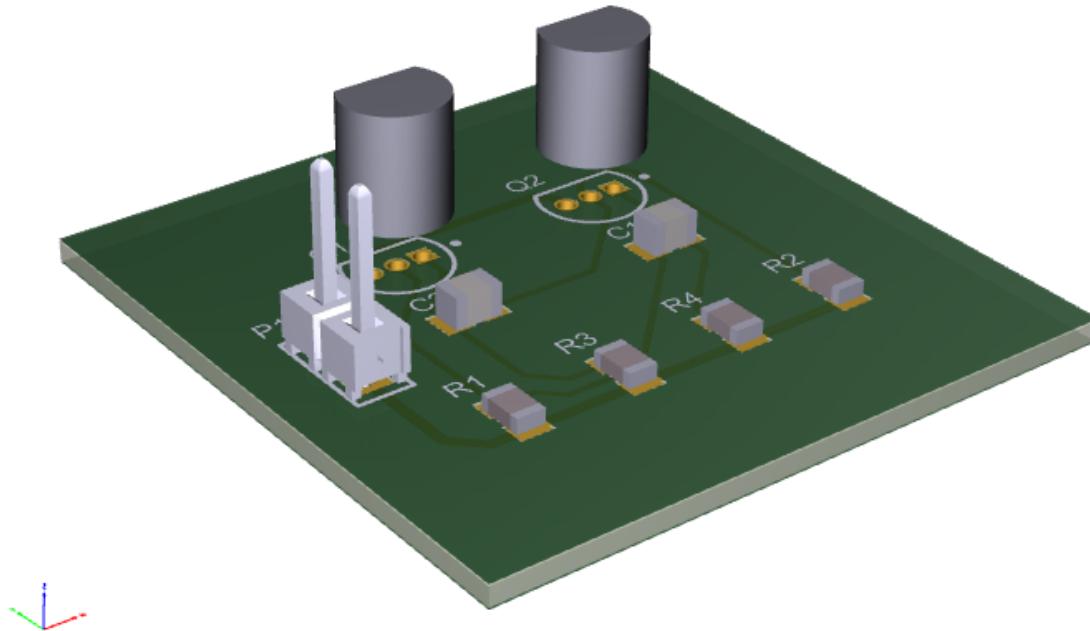
Warnings	Count
Total	0

Rule Violations

Rule Violations	Count
Clearance Constraint (Gap=0.25mm) (All), (All)	4
Short-Circuit Constraint (Allowed=No) (All), (All)	0
Un-Routed Net Constraint ((All))	0
Modified Polygon (Allow modified: No), (Allow shelved: No)	0
Width Constraint (Min=0.25mm) (Max=0.5mm) (Preferred=0.5mm) ((InNet('12V') OR InNet('GND')))	0
Width Constraint (Min=0.25mm) (Max=0.25mm) (Preferred=0.25mm) (All)	0
Routing Layers(All)	0
Routing Via (MinHoleWidth=0.6mm) (MaxHoleWidth=0.6mm) (PreferredHoleWidth=0.6mm) (MinWidth=1mm) (MaxWidth=1mm) (PreferredWidth=1mm) (All)	0
Differential Pairs Uncoupled Length using the Gap Constraints (Min=0.254mm) (Max=0.254mm) (Preferred=0.254mm) and Width Constraints (Min=0.381mm) (Max=0.381mm) (Preferred=0.381mm) (All)	0
Power Plane Connect Rule(Relief Connect)(Expansion=0.5mm) (Conductor Width=0.25mm) (Air Gap=0.25mm) (Entries=4) (All)	0

PCB in 3D

- A powerful feature of Altium Designer is the ability to view your board as a 3D object.
- To switch to 3D, **View » 3D Layout Mode** (shortcut: **3**)
- You can choose to **Show 3D bodies**, or hide them.
- To display the components in 3D, each component needs to have a suitable 3D model included in its footprint.



Output Documentation

Documentation Outputs:

- **PCB Prints** - configure any number or printouts (pages), with any arrangement of layers and display of primitives, use this to create printed outputs such as assembly drawings.
- **PCB 3D Prints** - views of the board from a three-dimensional view perspective.
- **Schematic Prints** - schematic drawings used in the design.

Fabrication Outputs:

- **Composite Drill Drawings** - drill positions and sizes (using symbols) for the board in one drawing.
- **Gerber Files** - creates manufacturing information in Gerber format.
- **NC Drill Files** - creates manufacturing information for use by numerically controlled drilling machines.
- **Power-Plane Prints** - creates internal and split plane drawings.
- **Solder/Paste Mask Prints** - creates solder mask and paste mask drawings.
- **Test Point Report** - creates test point output for the design in a variety of formats.

Output Documentation

Assembly Outputs:

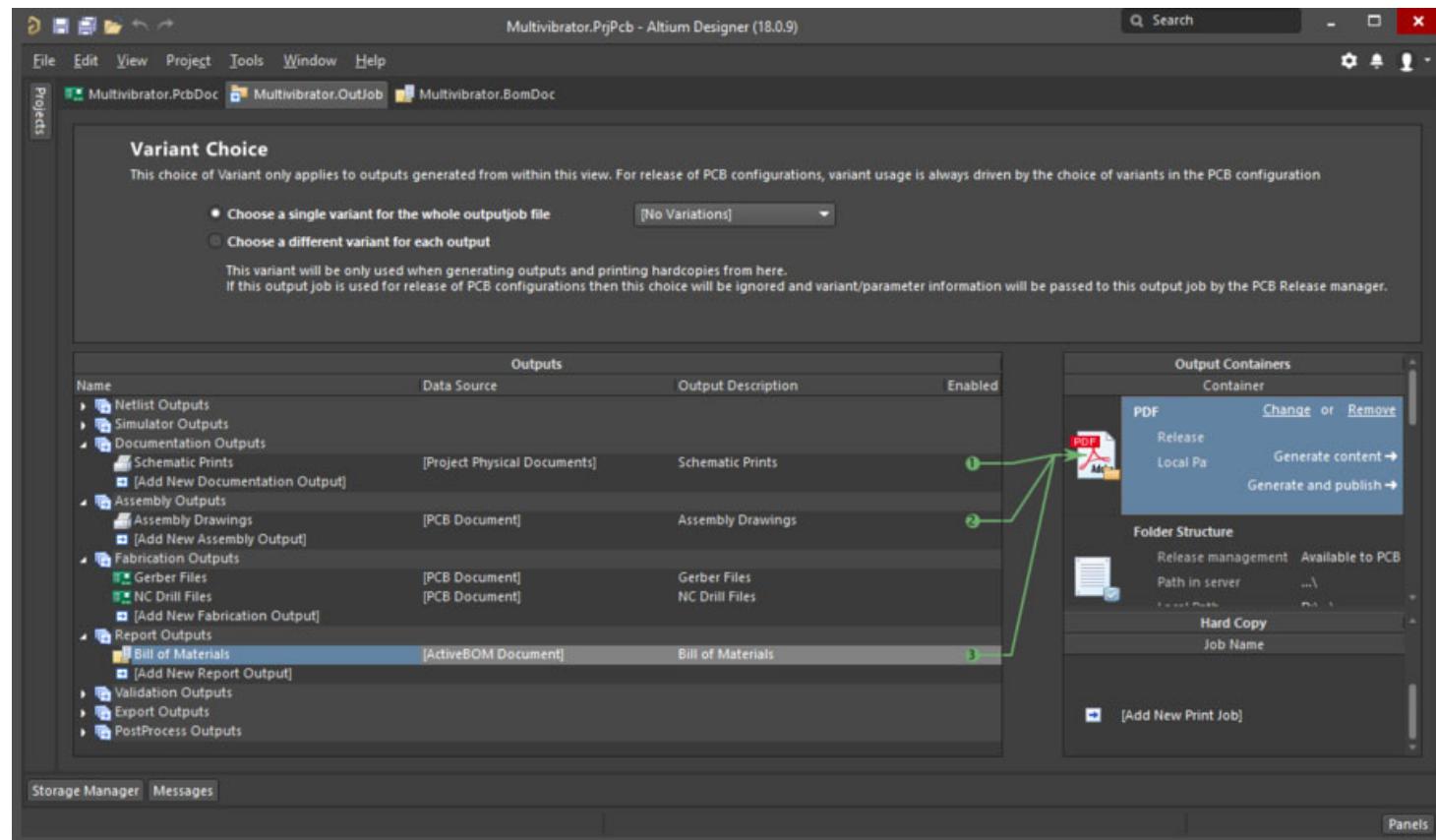
- Assembly Drawings - component positions and orientations for each side of the board.
- Pick and Place Files - used by robotic component placement machinery to place components onto the board.

Netlist Outputs

- Netlists - describe the logical connectivity between components in the design.
- Bill of Materials BOM - creates a list of parts and quantities (BOM), in various formats, required to manufacture the board.
- Component Cross Reference Report - creates a list of components, based on the schematic drawing in the design.
- Electrical Rules Check - formatted report of the results of running an Electrical Rules Check.

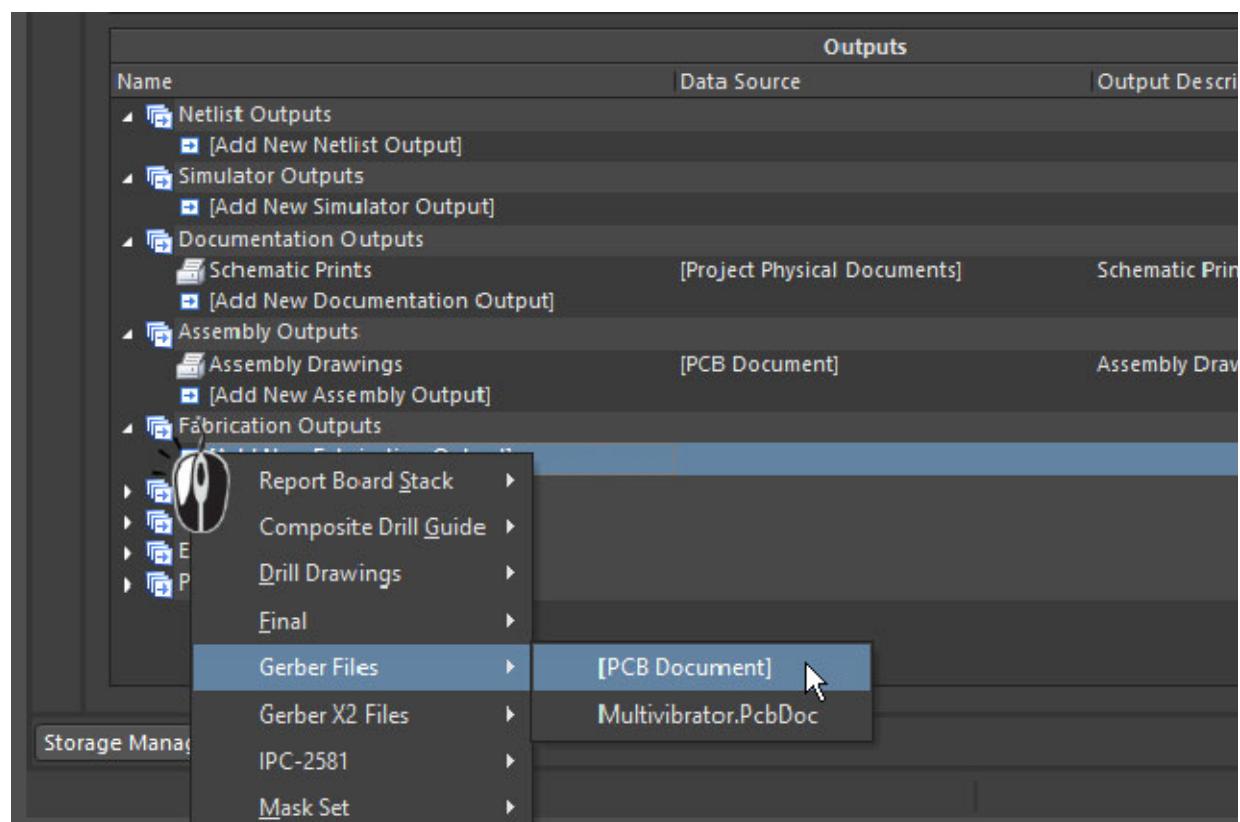
Generating Output

- **Via an Output Job file** - the settings for each output type are stored in an Output Job file, a dedicated output settings document, which supports all possible output types.
- An Output Job file allows you to configure each output type, configure their output naming, format and output location. Output Job files can also be copied from one project to another.



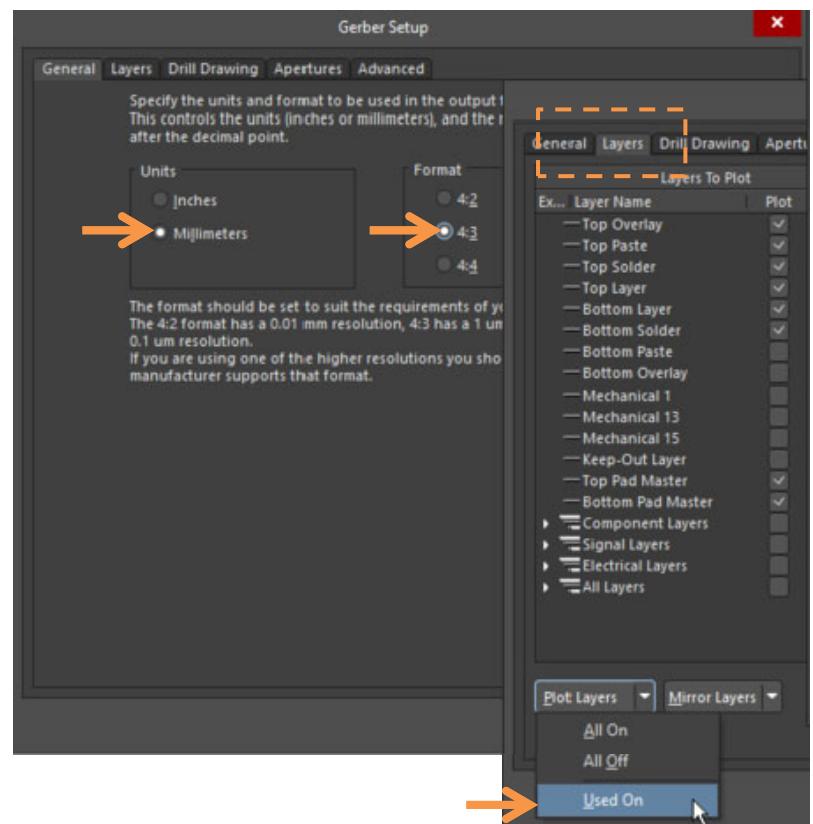
Generating Output: Gerber

- In the **Projects** panel, right click on the project name >> Add New to Project >> **Output Job File**. A new OutJob will be opened and added to the project.
- Save the OutJob and name it Multivibrator.
- To add a new Gerber output, click on **Add New Fabrication Output**, and select **Gerber » PCB Document**. If there are multiple PCBs in the project you will need to select the specific board.
- The Gerber output has been added, it will be configured shortly.



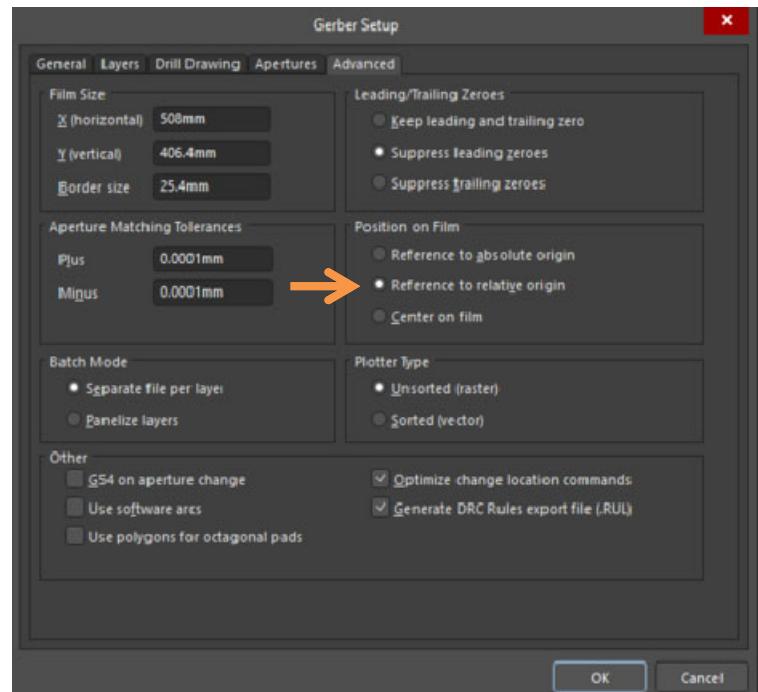
Configuring Gerber generation

- In the OutJob, double-click on the **Gerber Files** output, the *Gerber Setup* dialog will open, as shown in the image above.
- set the **Units** to **Millimeters** in the **General** tab. Set the **Format** to **4:3**. (NC drill file must be configured to use the same **Units** and **Format**).
- Switch to the **Layers** tab, then click the **Plot Layers** button and select **Used On**.
- Click on **Advanced** and Confirm that the **Position on Film** option is set to **Reference to relative origin**.



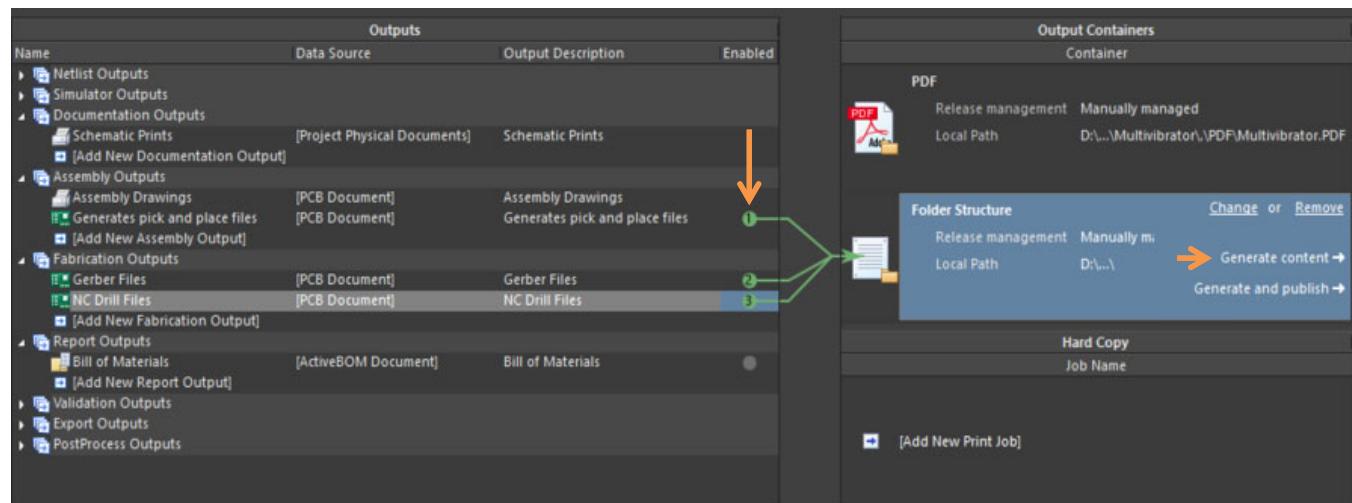
Configuring Gerber generation

- Click on the **Advanced** tab of the dialog. Confirm that the **Position on Film** option is set to **Reference to relative origin**.
- Click **OK** to accept the other default settings and close the *Gerber Setup*.



Configuring Gerber generation

- Once configured the Gerber, the next step is to configure their naming and output location: map them to an **Output Container**. Select **Folder Structure**.
- Click the circle button for the Gerber Files in the **Enabled** column of the Outputs to map this output to the selected container, as shown below.



- Click the **Advanced** button at the bottom of the *Folder Structure settings* dialog and enabled the **Gerber Output** in the list of **CAMtastic Auto-Load Options**. Click **OK** to close the dialog.
- To generate the Gerber files, click the **Generate Content** link in the **Container** region of the OutJob.
- The files will be generated and opened in the integrated CAM editor, which can be used for final checking of CAM files before you release them to manufacture. Close the CAM file without saving it.

PCB Design: Eagle

What's EAGLE?

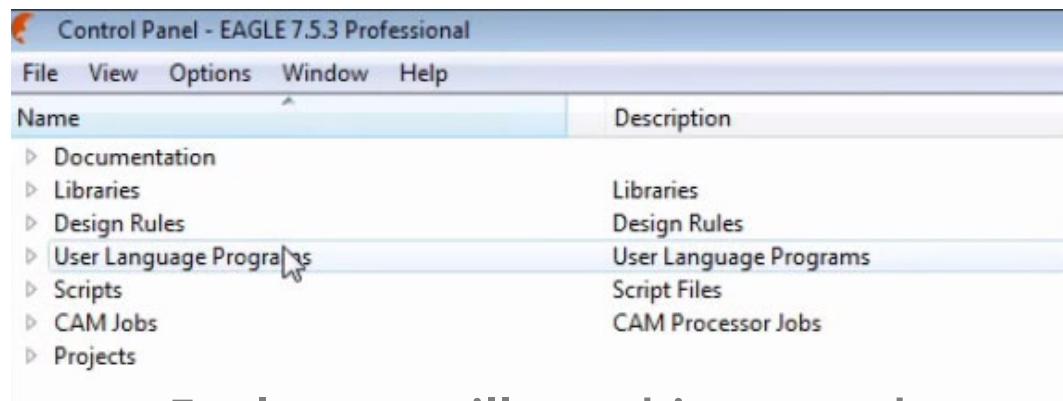
EAGLE is one of many EDA available on the market. It was developed by CadSoft Computer GmbH in 1998 (thirty years ago) but it was acquired by Autodesk Inc. in 2016.

Eagle is available with different license models:

- Premium, which gives you the maximum number of schematic sheets (999), layers (16) and PCB size (4 m^2) for 65\$/month
- Students' and educators' version, which has the same characteristic but for free;
- Standard, which gives you 99 schematic sheets, 4 layers and PCB size (160 cm^2) for 15\$/month
- The free version, which gives you 2 schematic sheets, 2 layers and PCB size (80 cm^2).



Overview – Control panel

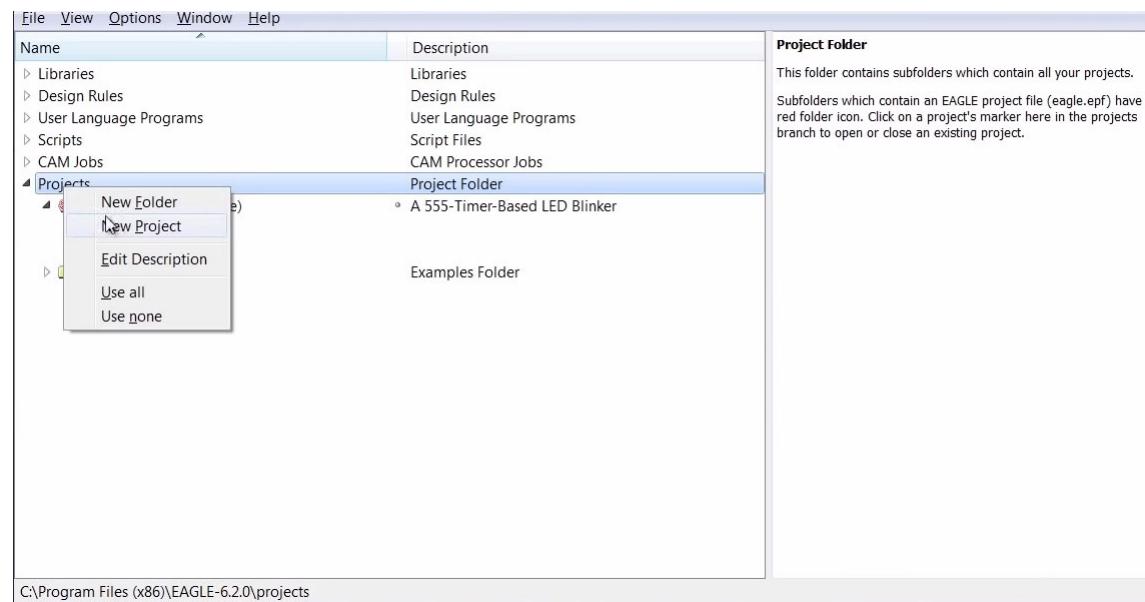


When you first open Eagle, you will see this screen known as the control panel. It consists of various trees:

- Documentation Tree: Here you can find all the manuals and setup information.
- Eagle's Libraries: There are several hundred libraries by default.
- Design Rules: These rules guarantee that your file is manufacturable (usually provided by the board house).
- User Language Program and Scripts: A mix of C with some extensions that allow users to script eagle to perform import and exports of different file formats.
- CAM Jobs: The mechanism Eagle uses to generate manufacturing data
- Projects: Contains the Eagle folder and the Examples folder.

New Project

- To create a new project, you just right click on “Projects” » “New Project.”
- It is possible to add a description to the project by right clicking and then clicking “edit description.”
- The first thing to do in a new project is to create a new schematic by right clicking on the folder and then clicking “Schematic”

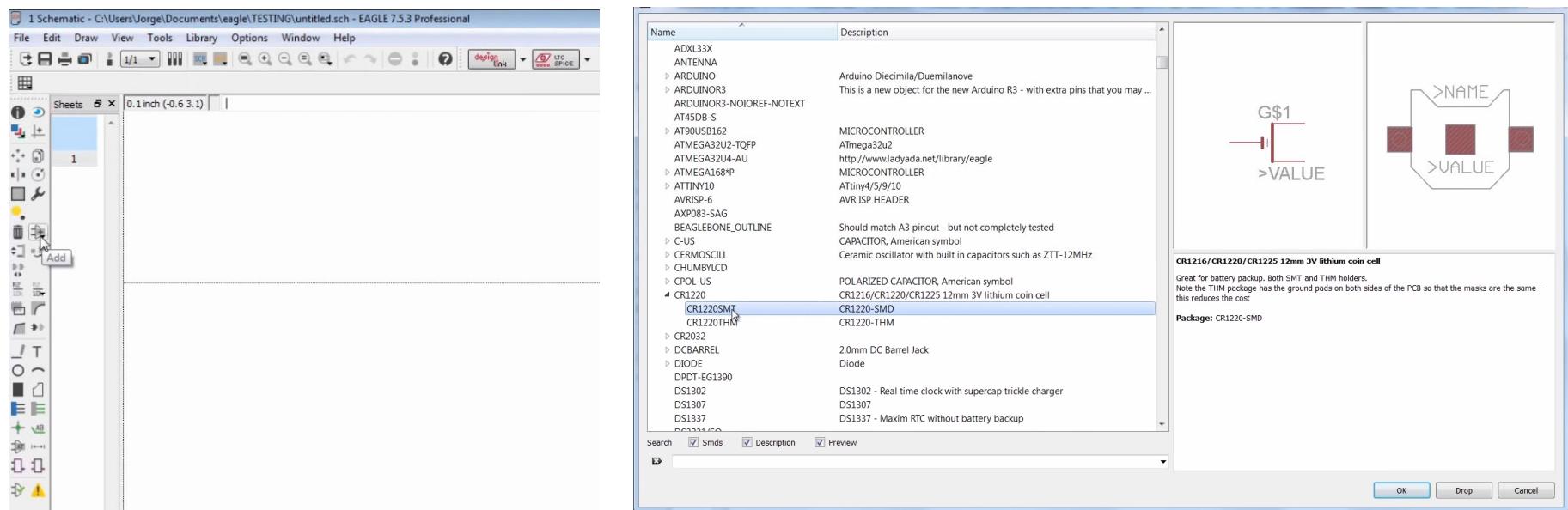


Libraries

The picture shows the basic layout of the schematic editor. Adding components is very easy and it can be done by using the add command.

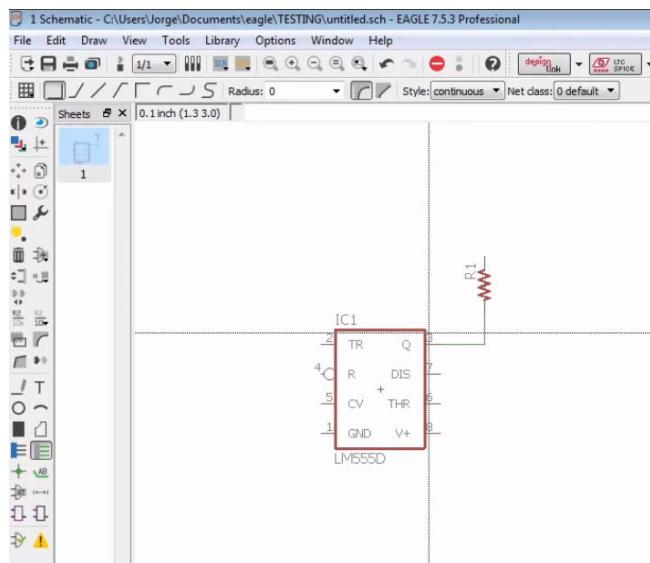
If the component you are looking for is not available, you can just add new libraries by downloading the .lbr file and adding it to the Library folder in the Eagle installation folder.

Then open the tab “Library” » “Use” and add the new .lbr file.



Schematic

- When the component is selected, a drawing of the part floats on the mouse cursor. It can be rotated by right clicking and it can be placed by left clicking.
- The characteristics of the components can be modified by right clicking on the component and pressing “edit properties.”
- With the move button pressed, components can be moved into the correct position.
- You can use single connections or busses. Single connection is easy and only requires clicking on the pin that you want to connect and dragging the wire to the other pin.
- Pay attention to the notes that appear if one wire cross another. The notes indicate that there is a connection between two wires.



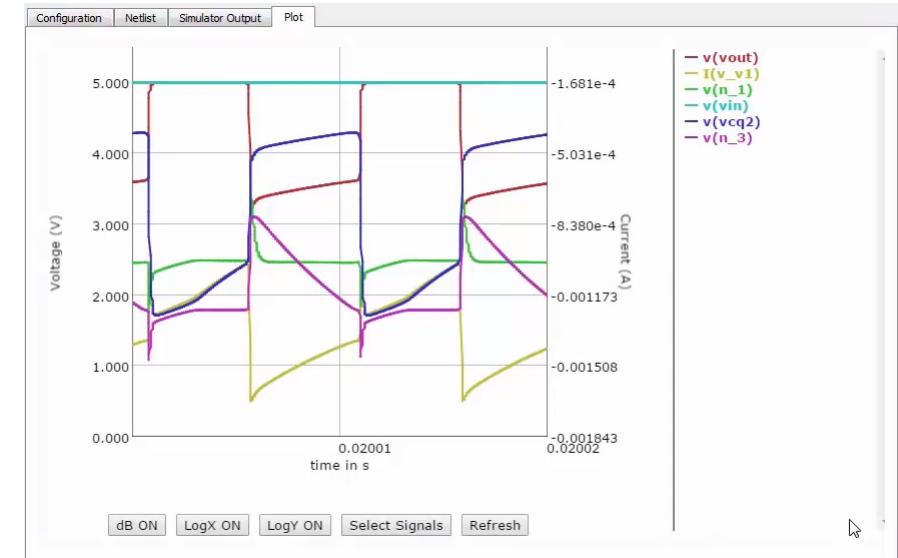
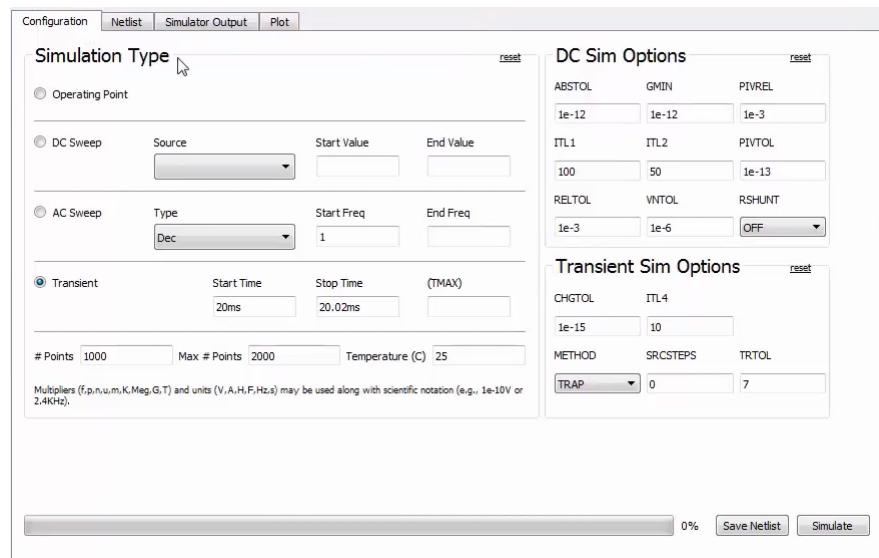
Spice Simulation

A new function just added to Eagle is the Spice Simulation. All the spice functions are available in the toolbar at the top of the screen.

Voltage probes can be added by simply clicking the button and then selecting the point where you want to put it. There are also phase probes.

Through the simulation command, you can manage the simulation setup, as well as view the results. The netlist tab shows a spice netlist for current design.

After clicking the simulate button, it will immediately jump to a plot of the simulation.

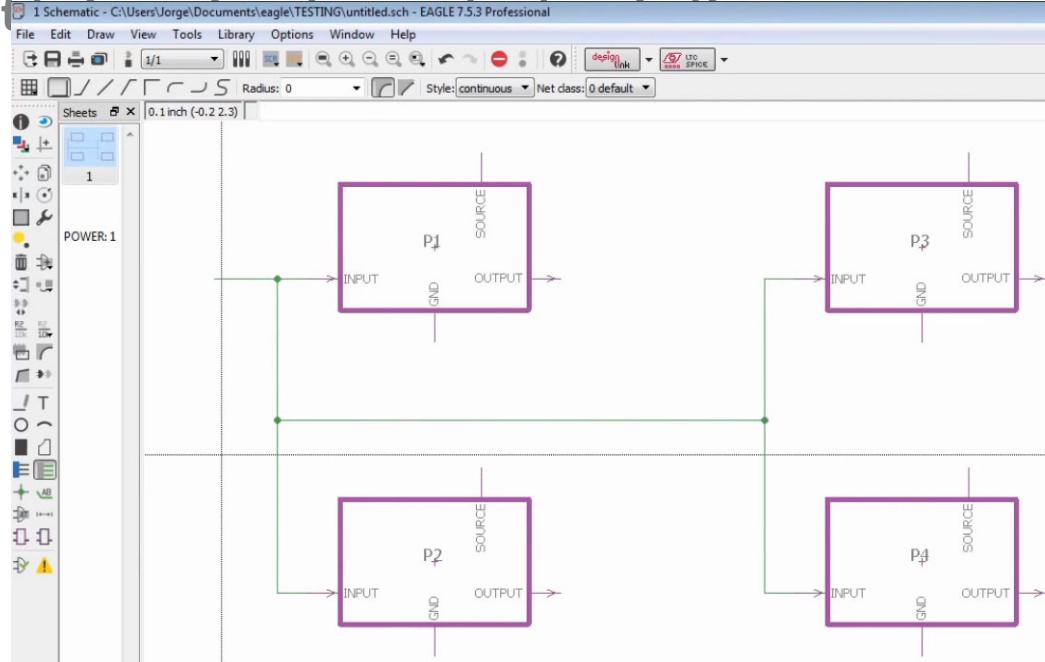


Hierarchy

Implementing hierarchy allows you to subtract from the complexity and shows the design described in simple blocks. This simplifies the division of labor, since blocks can be assigned to different engineers, and makes the design easier to read and to understand.

To create a Hierarchy:

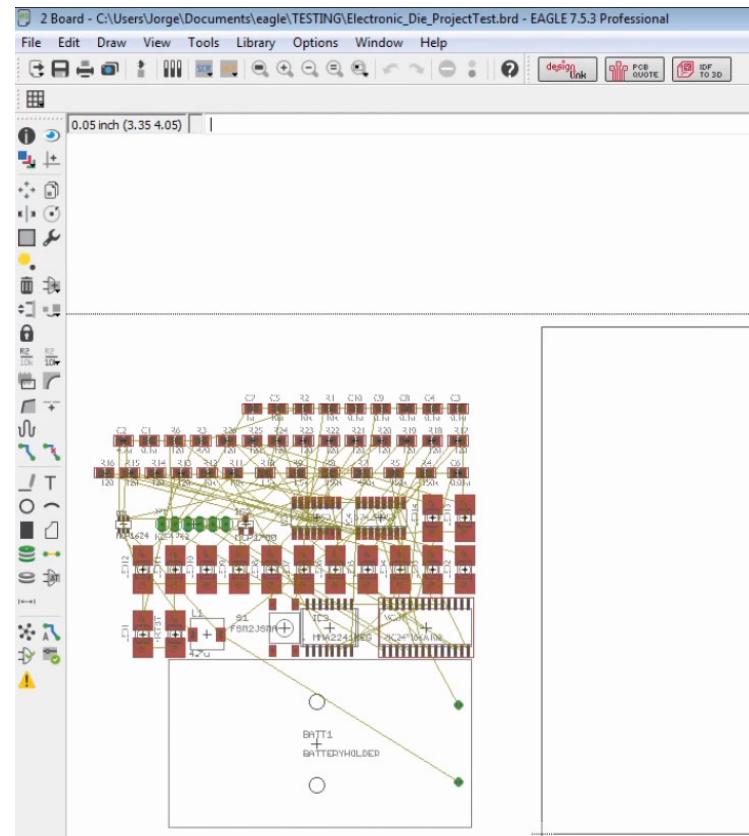
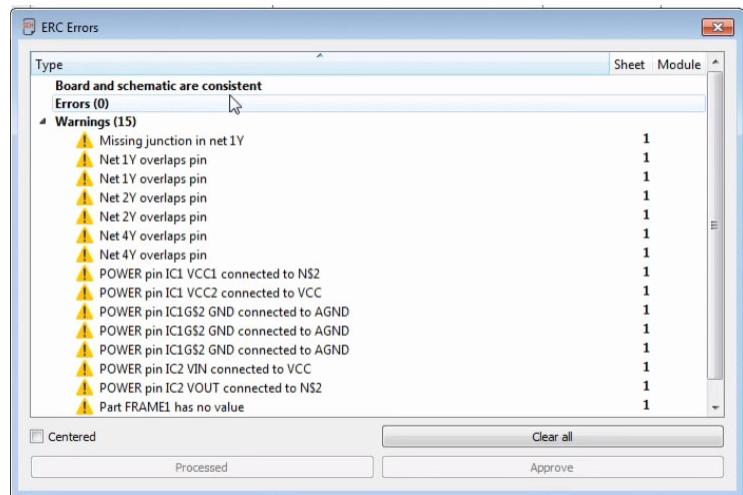
1. Go over the module icon, create a module, and give it a name.
2. Place as many blocks as you want / need.
3. Add IO through the ports command and give them names.
4. A pin can be set



Checking the schematic and creating the board

To check the schematic, simply click the ERC (electric rule check), to view all the errors and warnings related to the design.

Once the schematic design is completed, generating a board is simple and it: just click the generate board icon. If you have not already generated a board, Eagle will ask you to confirm your decision.



Design Link

- Design Link allows you to look parts up
- Click on “Design link” » “Schematic” Eagle will look for the parts in the schematic in the New York database and help you find the components you can use to assemble your circuit board.
- With this feature, you have an idea of how much the PCB costs.

AVX - 08056D226MAT2A - CAPACITOR CERAMIC, 22uF, 6.3V, X5R, 0805

Manufacturer: AVX
Order Code: 96M1423
Manufacturer Part No: 08056D226MAT2A
RoHS: YES

Description

- AVX - 08056D226MAT2A - CAPACITOR CERAMIC, 22uF, 6.3V, X5R, 0805
- Capacitance: 22 μ F
- Capacitance Tolerance: \pm 20%
- Dielectric Characteristic: X5R
- Voltage Rating: 6.3V
- Capacitor Case Style: 0805
- Operating Temperature Range: -55°C to +85°C

Packaging / Drawing / Footprint

Availability
Availability: 0

Price For: 1 TC
Minimum Order Quantity: 1
Order Multiple: 1

Price (USD)	Qty	List Price
1 - 49		1.8
50 - 99		1.44
100 - 499		1.03
500 - 999		0.677
1000 - 1999		0.526
2000 - 4999		0.45
5000+		0.436

Order code	Manufacturer	Manuf. code	Availability	Price (from)	Description
96M1423	AVX	08056D226MAT2A	0	0.436	AVX - 08056D226MAT2A - CAPACITOR CERAMIC, 22uF, 6.3V, X5R, 0805
96M1523	AVX	1210YC226MAT2A	0	0.62	AVX - 1210YC226MAT2A - CAPACITOR CERAMIC, 22uF, 16V, X7R, 1210
96M1536	AVX	12103D226MAT2A	3271	0.484	AVX - 12103D226MAT2A - CAPACITOR CERAMIC, 22uF, 25V, X5R, 1210
96M1564	AVX	1812ZD226KAT2A	198	0.66	AVX - 1812ZD226KAT2A - CAPACITOR CERAMIC, 22uF, 10V, X5R, 1812
98K6375	SANYO	20SVP22M	0	0.799	SANYO - 20SVP22M - CAPACITOR POLYMER ALUM, 22uF, 20V, SMD
99M0583	VISHAY SPRAGUE	199D226X0016D1V1E3	456	0.394	VISHAY SPRAGUE - 199D226X0016D1V1E3 - CAPACITOR TANT, 22uF, 16V, RADIAL

Description EAGLE Part/Device:
Value: 22uF Package: C0805
CAPACITOR, American symbol

22uF Select Skip this Manual Search Help... Directly to order list

Placing and routing

Once the board is generated, all the components are placed on the left side of the screen. The rectangle on the right represents the limits of the work area.

Moving components is done simply by using the move command.

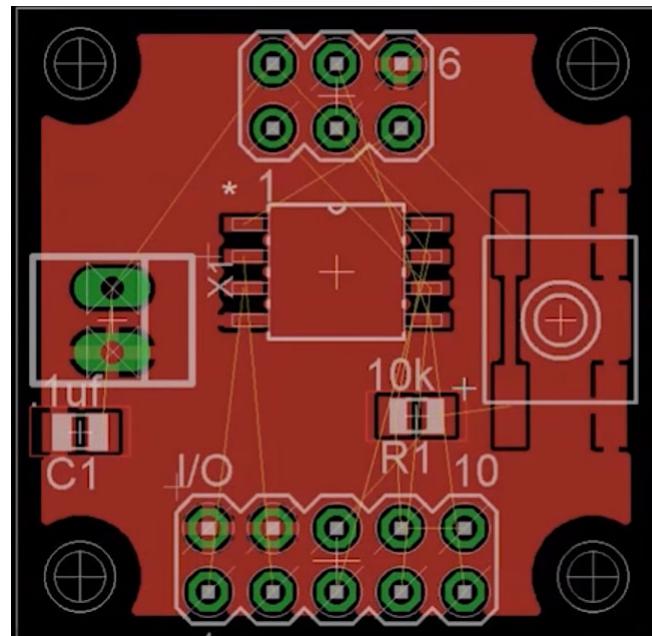
If a component has to be placed on the bottom, it can be done by mirroring it: Just click the scroll wheel.

All the yellow wires are the connections defined in the schematic. The goal is using the route command to convert them into copper wire. Whenever you left click, you create a new segment of wire, pressing the scroll wheel allows to move the wire on the bottom layers using a via. The diameter of the via can be easily set in the option panel on the top of the screen.



Power ang GND planes

By using the polygon tool you just have to draw a rectangle as big as your board. Once you've done that you can give the rectangle a name with the Name tool. If you, for example, renominate it GND it'll be connected with everything else that's ground. To see the result the ratsnets button must be clicked, Eagle will recalculate everything in the board and and the airline labeled GND get's connected to enything that's ground

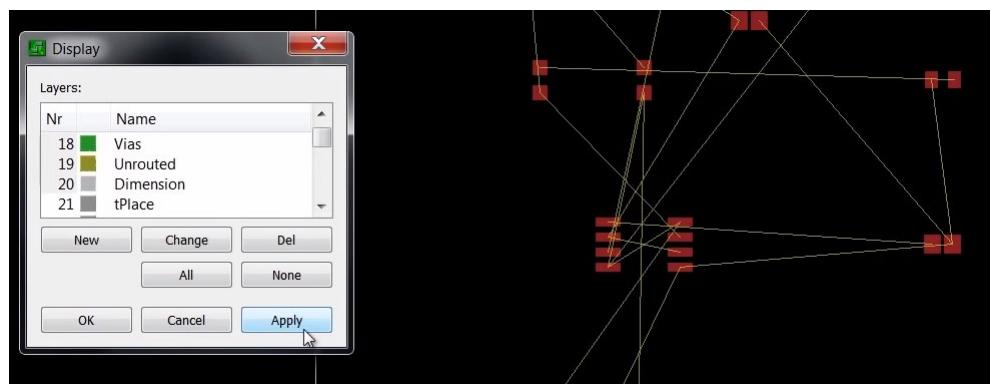


Layers

With the layers button, you can select the layers you want to see on the schematic.

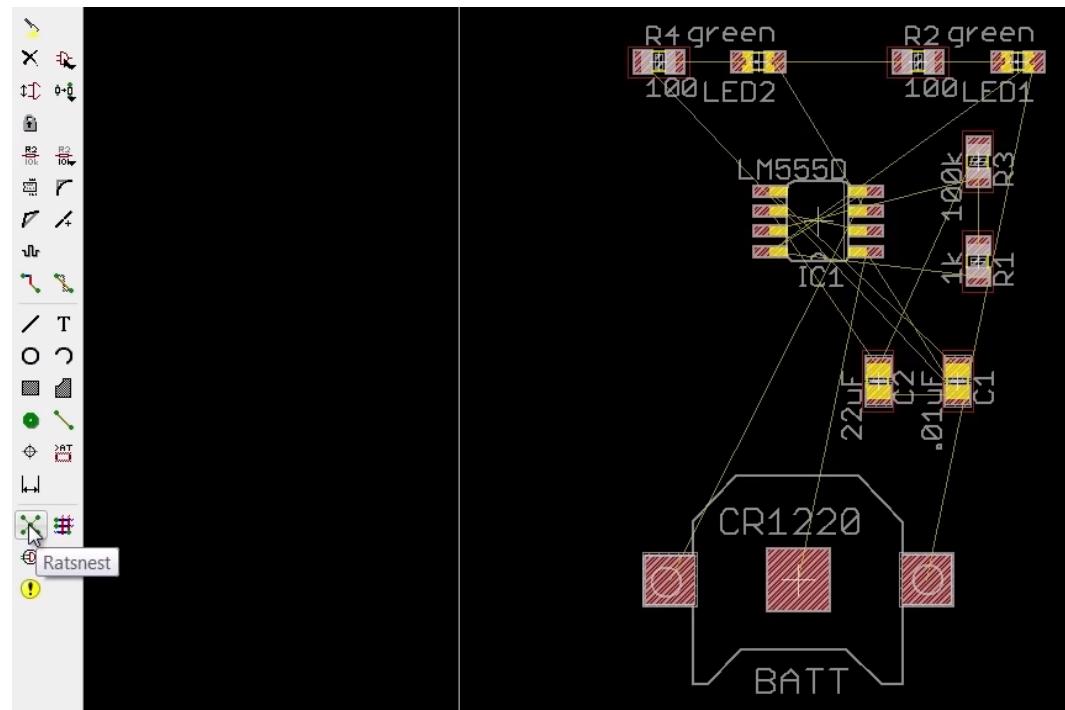
The most important layers are usually:

- **Top:** Defines the top copper layers. This is where you want to have connections among various components and where each of the components' pads are.
- **Bottom:** Defines all the copper on the bottom.
- **Pads and Vias:** Shows connections between layers.
- **Unrouted:** Shows the yellow wire that represents the connection that you have to make.
- **Dimension:** Indicates the size of the board.



Autorouting

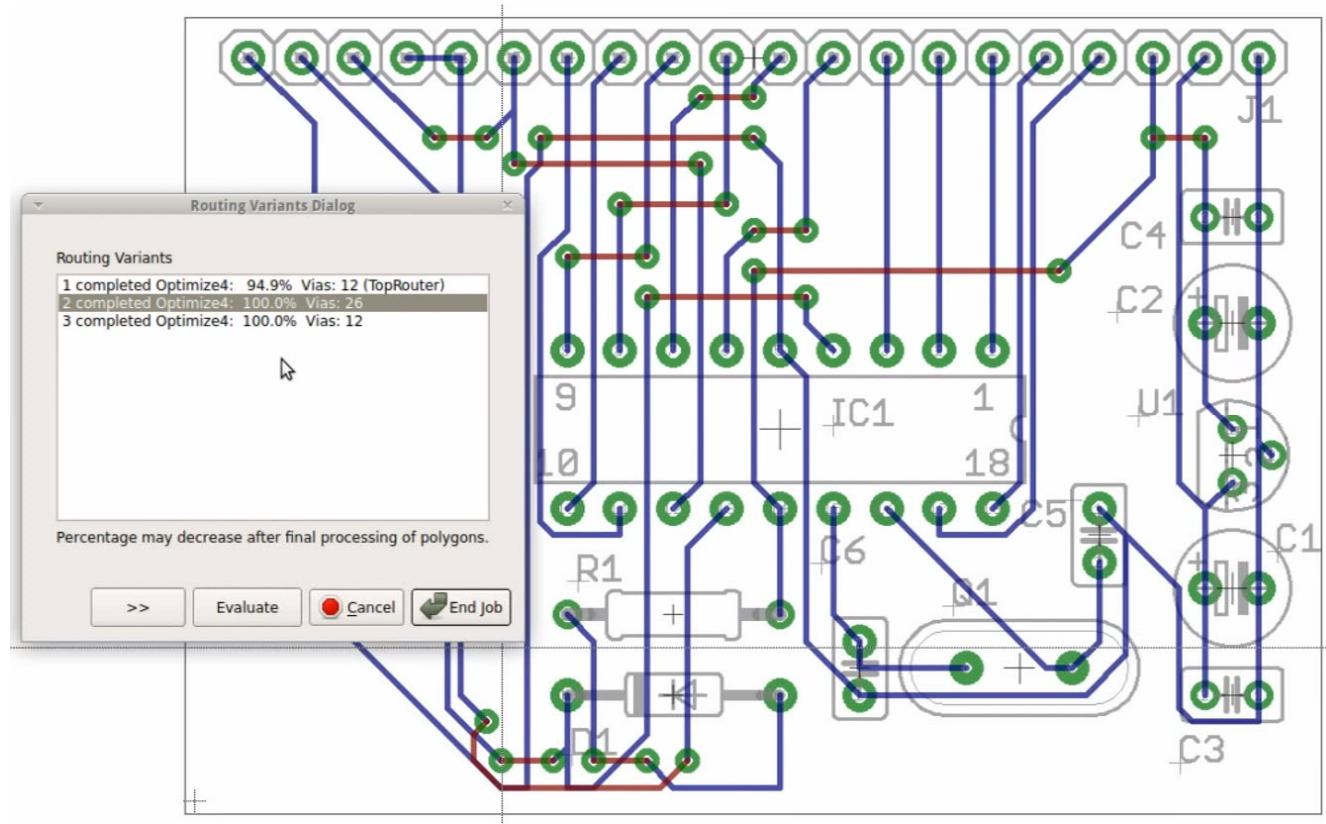
EAGLE comes with an Autorouter command. It can calculate a number of routing jobs with different settings for the board. Once you have the parts where you want them click the ratsnest button and it will draw new lines defining where everything is connected and that should make it easier to start drawing a connection on your own. After processing the routing variants the results can be evaluated in order to choose the best one.



Autorouting

There are different ways that Eagle* can be used to make connections between components. You can use Eagle autorouter or you can do the routing manually.

By clicking “Start,” the autorouter will bring up a few options, defining the layers it’s allowed to use and how it will prioritize each one. The other tabs list the optimizations you can work with to get the best writing (wiring?)* possible. In the end, different solutions will be available and described by percentage of optimization and number of vias.

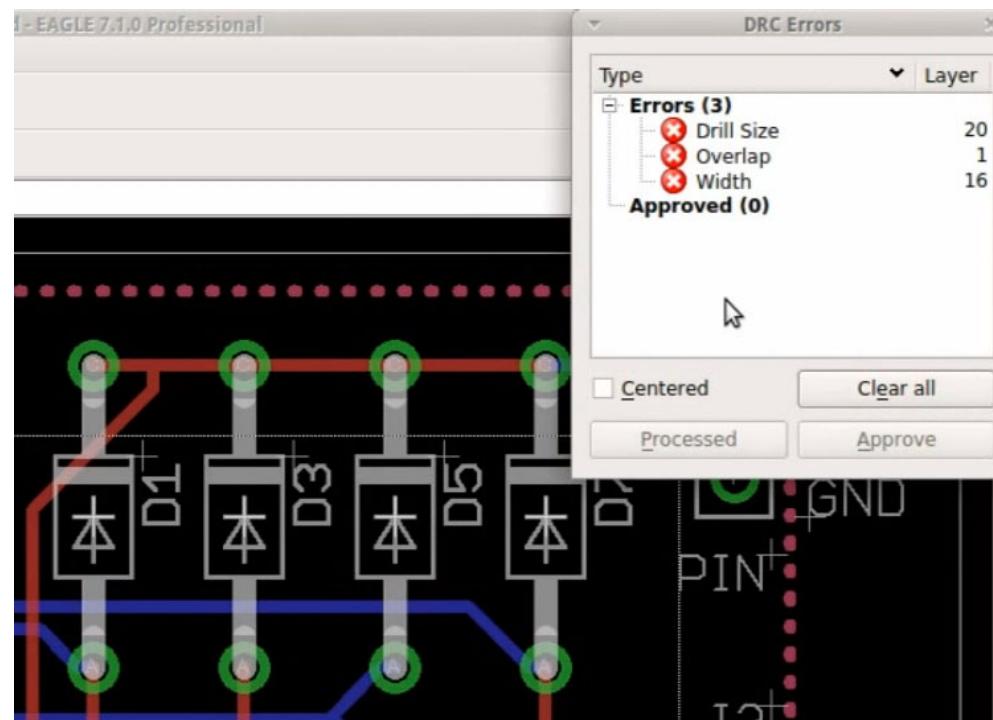


Design Rule Check (DRC)

Design Rule Check refers to the capabilities of the PCB manufacturing house and checks if the board fits its parameters.

By clicking the DRC button and pressing «load» , Eagle allows you to upload* DRC files that can be downloaded from the PCB supplier website.

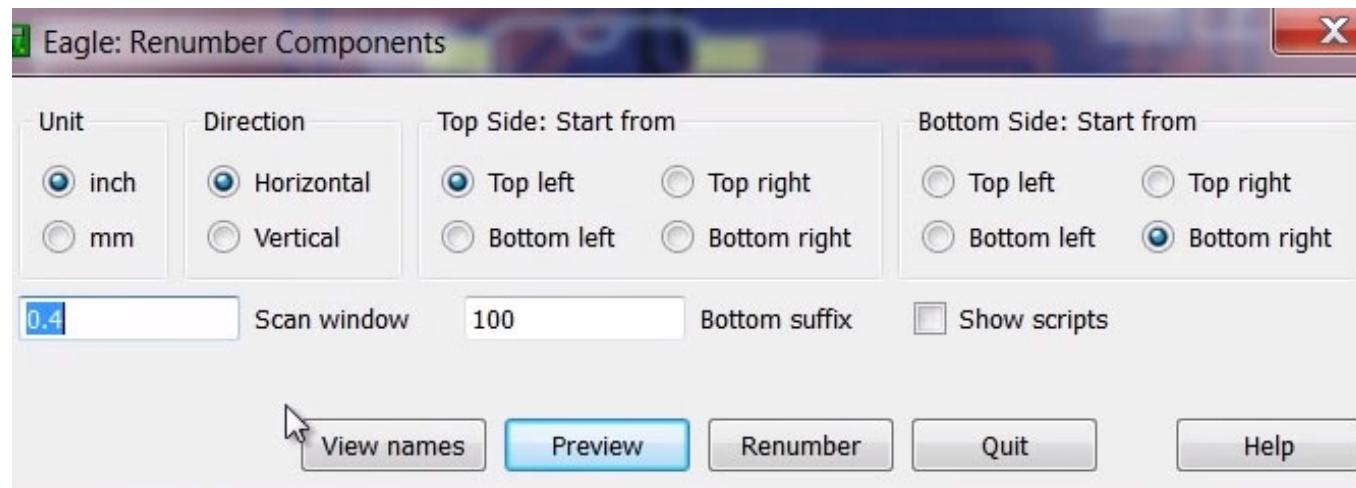
If you dont have a DRC file, you can select the parameters manually, navigating through the tabs and configuring the specifics the project must have.



Adjust Board Design

By using a user language program, accessible by clicking the ULP button, you can automatically adjust your board design.

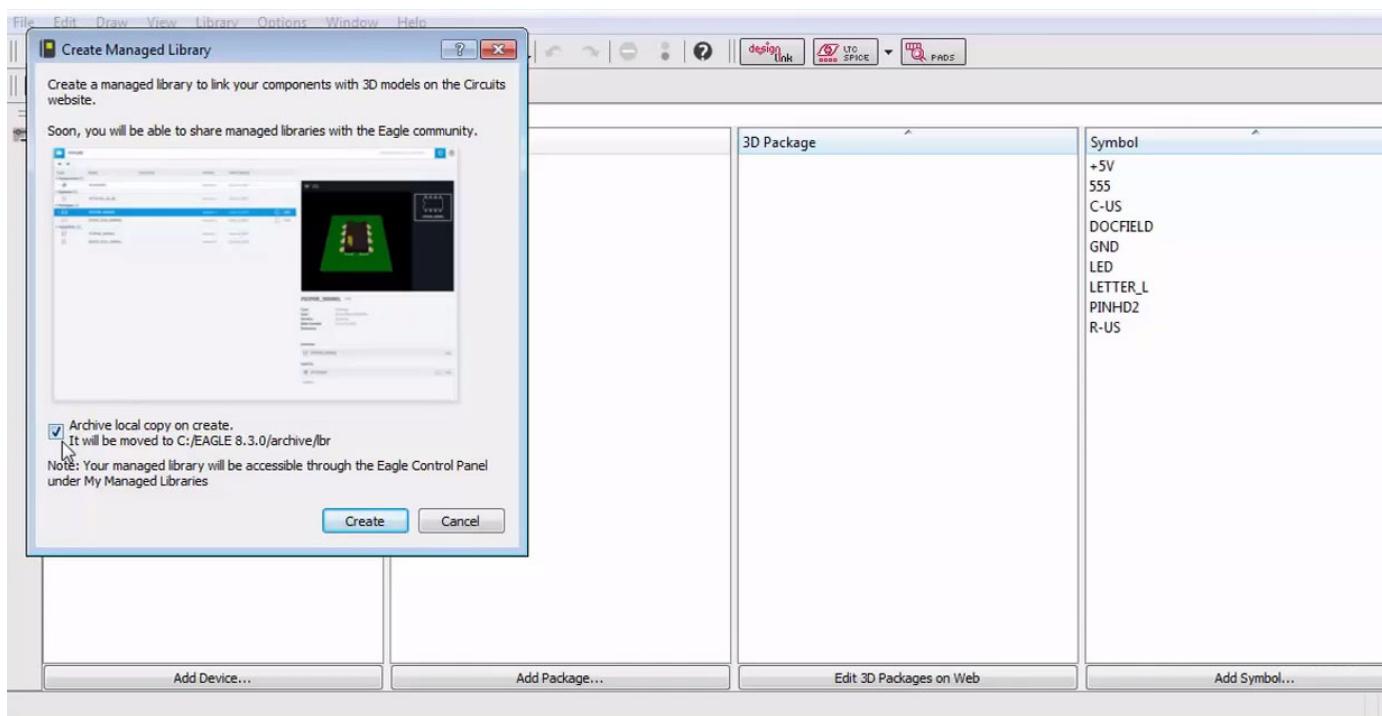
For example, with the command “Renumber” you can renumber things in the order of the more* (You can reorder things...?) making it easier to find the parts that you want in large boards. After pressing “Renumber” all the components will be renumbered by going down from top left.



Fusion 360

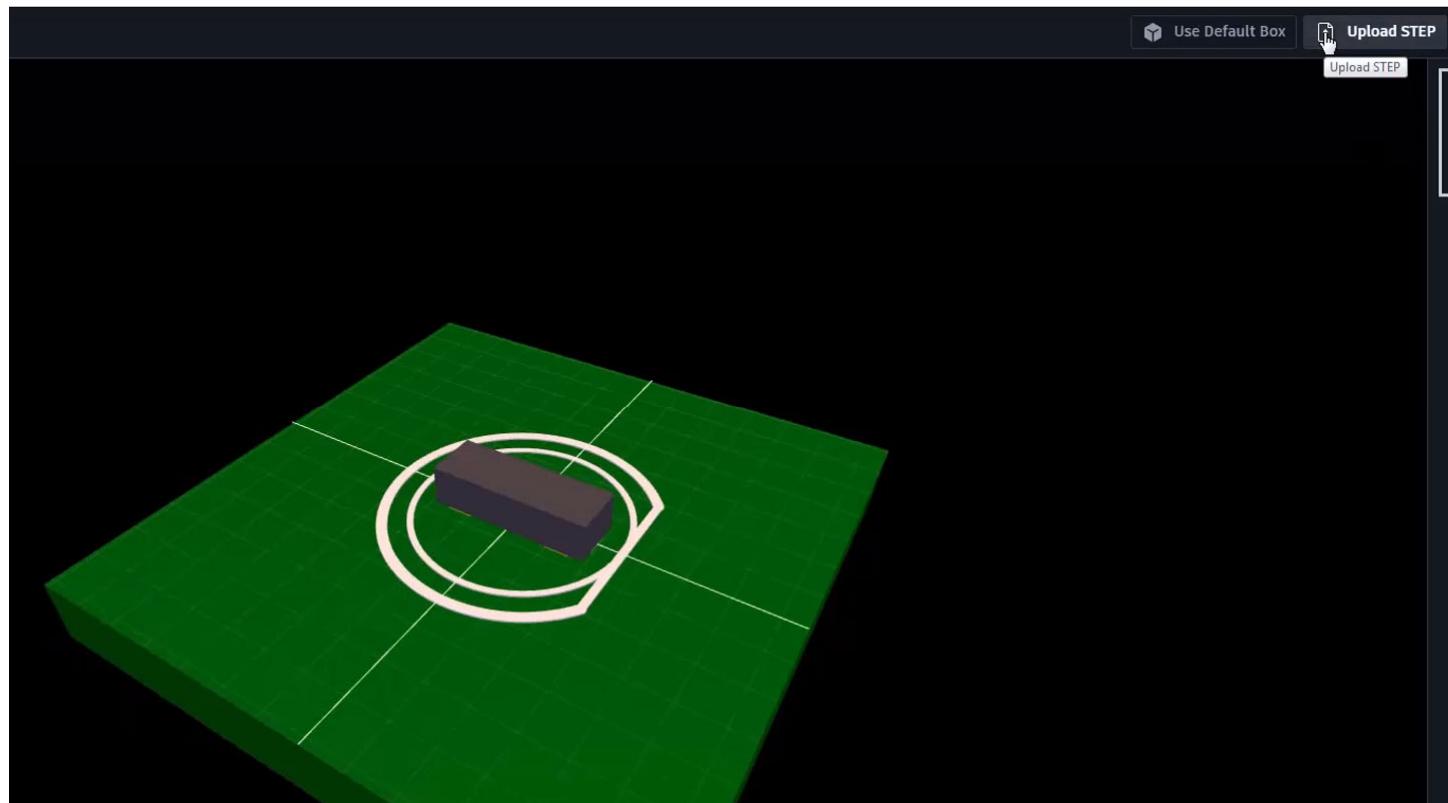
Autodesk made it possible to integrate Eagle and Fusion 360. To do so, it is necessary to map 3D models of the components to achieve a more realistic model in fusion. The library used for the project must be turned in a managed library.

Managed libraries are the way that Eagle and Fusion use to communicate with each other. To do this, you must select the library of your project, click on the tab «Library» -> «Create a Managed Library» -> «Create».



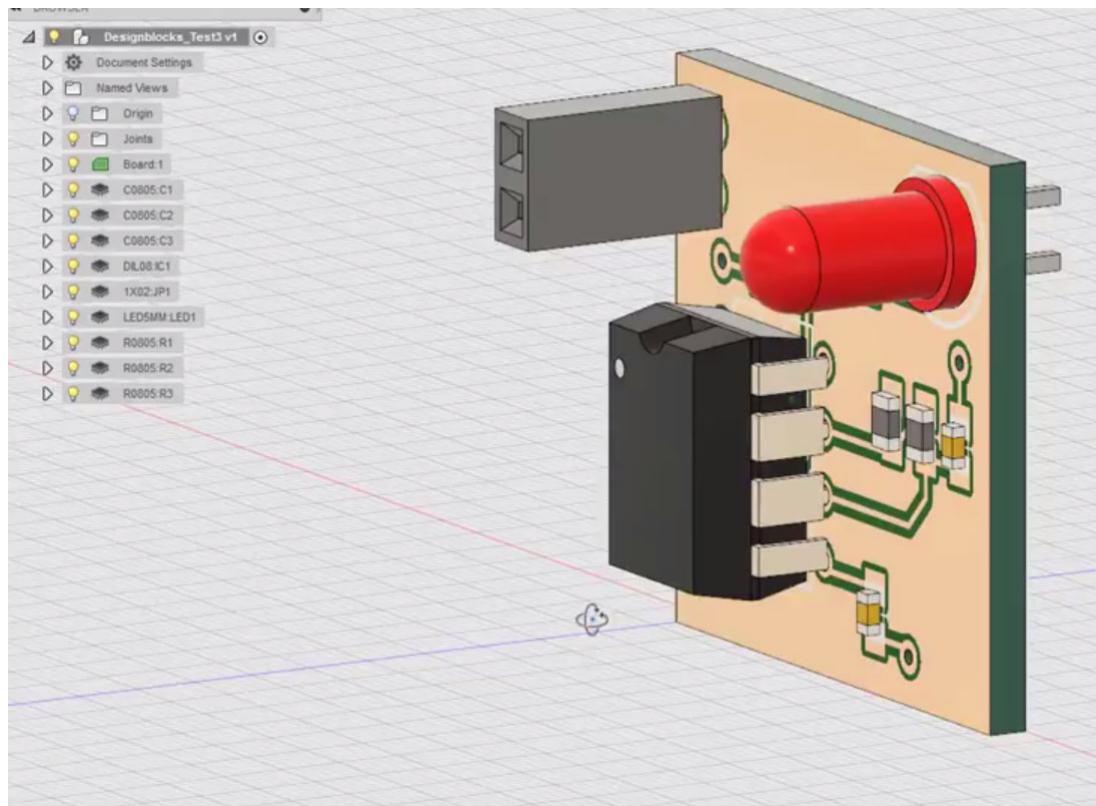
Fusion 360

Then, you have to go into the schematic of the board to do a library update and select the new library. You will have a distinction between packages and footprints in the components information. If the 3D model is not available, the program will give the component a block representation. You can update it, if you have a step file of the package, by clicking “Upload STEP.”



Fusion 360

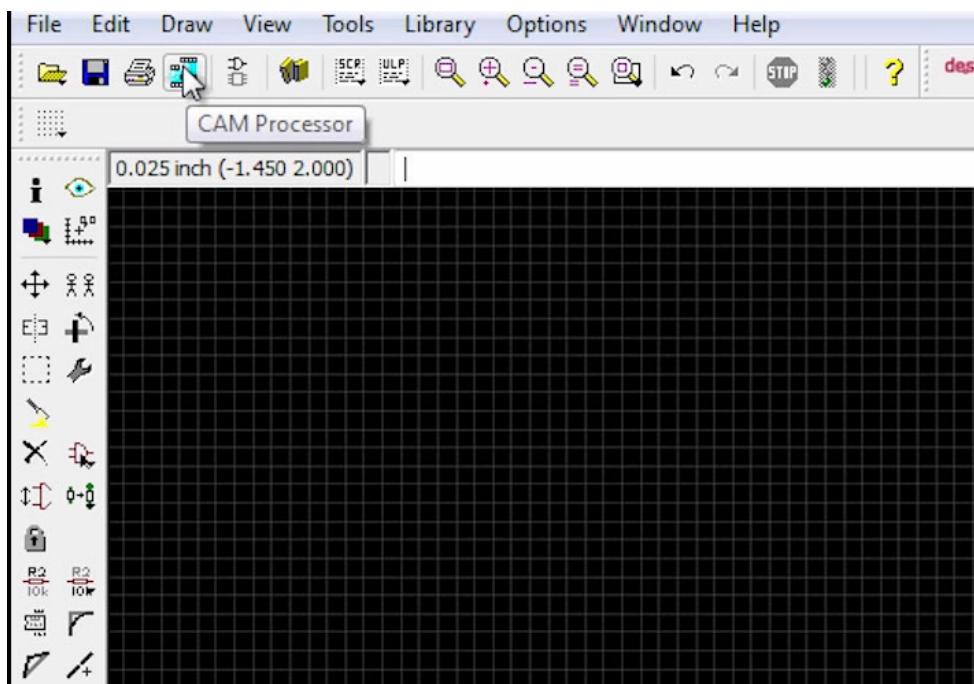
Once you have updated the library with all the mapped components for the 3D packages, you just have to click the button «Fusion sync» and «Create a new Fusion design», then select your project name and click «OK». When the pushing operation is done, you can switch over to fusion and update the project (or open it, if it wasnt opened yet) and see the fully mapped design with all the components.



CAM Processor

The CAM Processor allows you to create the data you need to manufacture your PCB. It can produce data for drill stations, photo plotters. You can find the button in the top left part of the screen. In this window, you can select all the layers from within Eagle and how those layers from Eagle translate into layers for Gerber files.

Every tab represents a different Gerber file that to be generated.



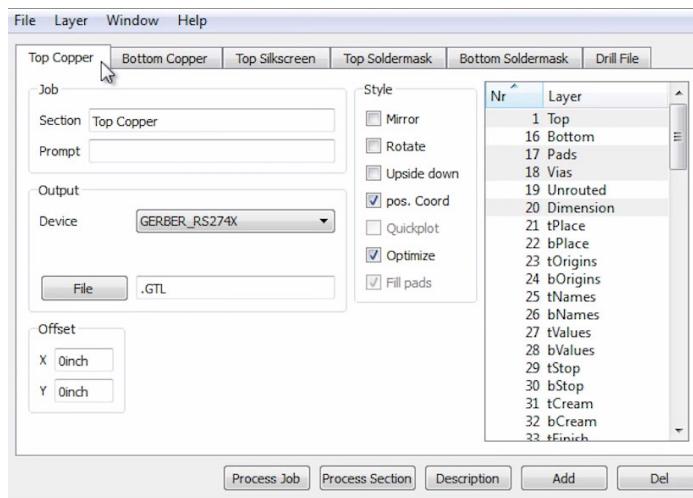
CAM Processor

There is also a tab for the drill file which specifies where the board fabrication house has to drill holes for your vias or through-hole components or for actually installing the board.

Each tab has a number of actual layers from Eagle that have been specified to be added to the copper Gerber file. For example, the top copper Gerber file will have the top layer, pads, vias and dimensions.

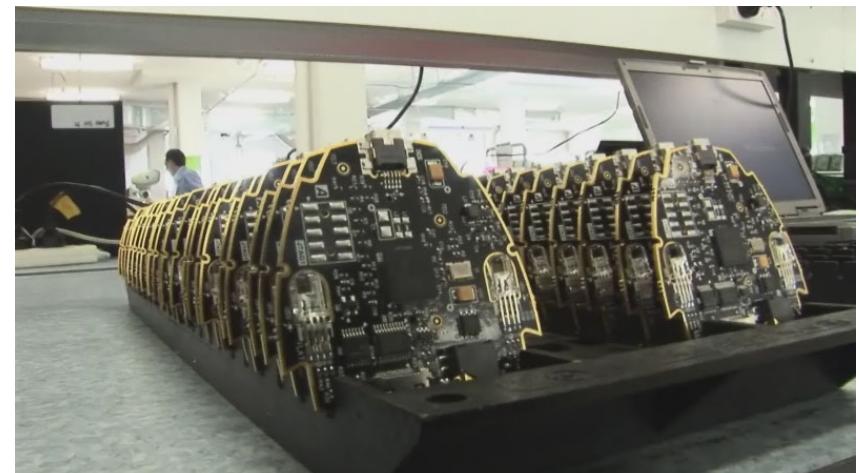
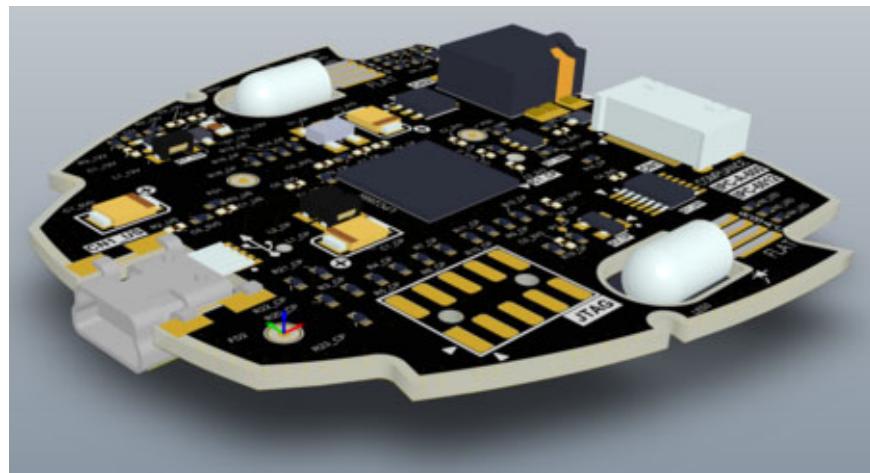
You can easily adapt the CAM Processor to your specific output device from the Output Device button. Then, the extension of the file will be modified based on your choice.

Once you checked everything, clicking “Process Job” will print all the apertures and generate output Gerber files in the folder of your project.



Design Completed

- Once completed the design, PCB layout and the output documentation, after all these output documentation is provided to a PCB house that fabricate and some, assemble the PCB boards too.



Conclusions: Altium vs Eagle

1. A high-quality 3D visualization that allows for routing using 3D.
 2. The automatic appearance of net names on the pads of the printed circuit board editor.
 3. An outstandingly advanced filtering which simplifies selection and modification of what a user wants.
 4. A simplified design constraints creation procedure that has an in built helper.
 5. Manufacturing and Generation of the output file are superior.
 6. A good number of hot keys and shortcuts that allow efficiency in PCB design.
 7. Its design system operates offline and can still function without a license.
 8. The adds-on for export and import used in other programs and formats are quite decent.
 9. Its project tree is simple and allows user drag-n-drop files to desired locations.
-
1. An auto-router
 2. Copper pouring
 3. Different types of outputs for used to manufacture data.
 4. Features and techniques that allow for routing at advanced levels.
 5. Backward and forward annotations between printed circuit boards and schematic.
 6. Users have the option of defining certain things such as the clearance, nets and width of wires among others.

Web-based references

- <https://www.altium.com/documentation/18.0/display/ADES/From+Idea+to+Manufacture+-+Driving+a+PCB+Design+through+Altium+Designer>
- <https://autodesk.com/products/eagle/overview>
- [https://en.wikipedia.org/wiki/EAGLE_\(program\)](https://en.wikipedia.org/wiki/EAGLE_(program))
- <https://learn.sparkfun.com/tutorials/how-to-install-and-setup-eagle>
- <https://youtube.com/watch?v=1AXwjZoyNno>
- <https://youtube.com/watch?v=CCTs0mNXY24>
- <https://youtube.com/watch?v=old-h6AeXXE>
- <https://medium.com/@warrenC2017/eagle-vs-altium-what-is-the-features-and-differences-961793629145>