ALTIUM DESIGN GUIDANCE

BY EREN ALPASLAN

Contents

Altıum Desıgn Guıdance 1

By Eren Alpaslan 1

1. Shortcuts 2

1.1 PCB Design Shortcuts 2

1.2 Schematic Design Shortcuts 3

1.3 PCB Library Shortcuts 4

1.4 Schematic Library Shortcuts 4

2. Creating PCB Library 4

3. Creating Schematic Library 4

4. Design 4

4.1 Adding PCB to project 4

4.2 Transfer schematics to PCB 5

4.3 PCB routing 5

4.4 Via 9

4.4.1 How to add Via to PCB 9

4.5 How to add Pour 10

4.6 Multi-Layer PCB 11

4.7 Design Rules 13

4.7.1 Rules 13

4.8 Exporting PCB 14

4.8.1 Exporting as a PDF file 14

4.8.2 Exporting as a GERBER file 17

1. Shortcuts

1.1 PCB Design Shortcuts

- **2** → 2D view
- **3** → 3D view
- **CTRL + W** or **P + T** → Draw wire
- **CTRL + Shift + B** → Align from bottom
- **CTRL + Shift + T** → Align from top
- **CTRL + Shift + L** → Align from left
- **CTRL + Shift + R** → Align from right
- **CTRL + Shift + D** → Move selected objects to the nearest point on the current snap grid
- **Hold CTRL + R.Click** → Page Up and Page Down
- **Hold L.Shift + Left Click** → To select multiple components
- **Hold Left Click + L** → Rotate component to other side of PCB
- **Hold L.Shift + Rotate with Right Click** → Rotate around PCB
- **Q** → Change grid metric between mm and mils

- **G** → Change grid settings
- **Double press G** → Change grid settings to entered value
- **P** → Place
- **CTRL + M** → Measure the distance (To clear it **Shift + C or Right Click → Clear Filter**)
- **Tools → Cross Select Mode**: When you click component from PCB or Schematic, it shows where component is.
- **Hold Left Click + X** → Mirror
- **Hold Left Click + Space** → Rotate
- **Click one component or path**, **press TAB** (or **double TAB**) → It highlights all the path the component
- **Hold CTRL and click the component or path** → It highlights all the path of the component, To unhighlights **SHIFT + C**
- **Shift + S** → Hide other layers on particular layer
- **J + C →** Find the component (For ex: enter c2 → it will find capacitor C2)
- In 3D, **V + B →** It rotates PCB 180 degree

1.2 Schematic Design Shortcuts

- **CTRL + W** → Draw wire or **P + W**
- **P + P →** Adding component
- **P + T →** Adding text
- **P + L →** Adding label

1.3 PCB Library Shortcuts

1.4 Schematic Library Shortcuts

2. Creating PCB Library

3. Creating Schematic Library

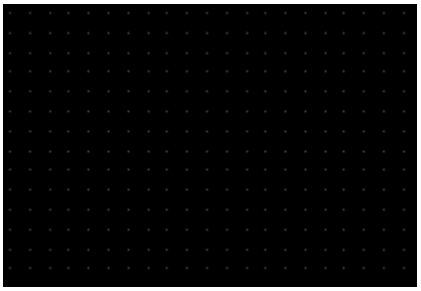
4. Design

4.1 Adding PCB to project

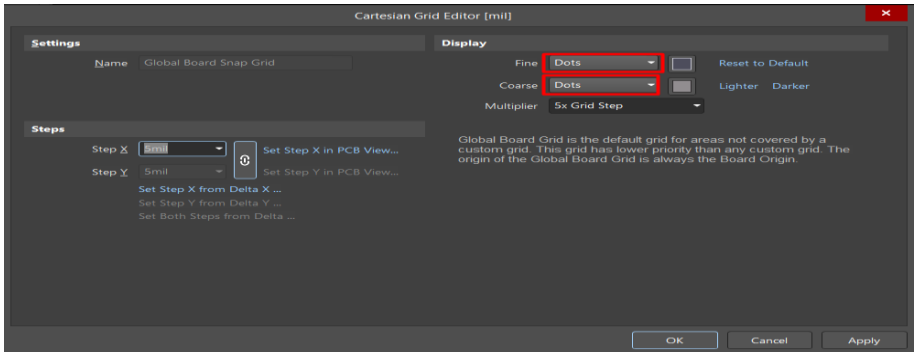
When you complete schematic and PCB libraries, and complete schematic design in *.SchDoc*, it is time to create PCB layout.

If you did not add *.PcbDoc*: **Right Click to project → Add New to Project → PCB**

When you go to *.PcbDoc*, you probably see dark screen with dots



To change it: Press **G → Grid Properties →**



Change Dots → Lines

4.2 Transfer schematics to PCB

First you need to annotate components. You need to on *.SchDoc*→ **Tools** → **Annotation** → **Annotate Schematics Quietly**. You can add or remove schematics on .SchDoc whenever you need to change but you need to follow same process to update.

To update Schematics to PCB layout: **Design** → **Update Schematics in blabla.PrjPcb** → **Execute Changes** (To see if there is a mistake or not) → **OK** (Now you see your components on .PcbDoc)

Now, you need to specify PCB dimensions. To measure the dimensions **CTRL + M** to add measurement.

Then Press **P → Line**, draw PCB however you want

Select one line **Press TAB →** to select all lines you draw**. Design** → **Board Shape** → **Define from selected objects** (Now you create PCB layout with specified dimensions)

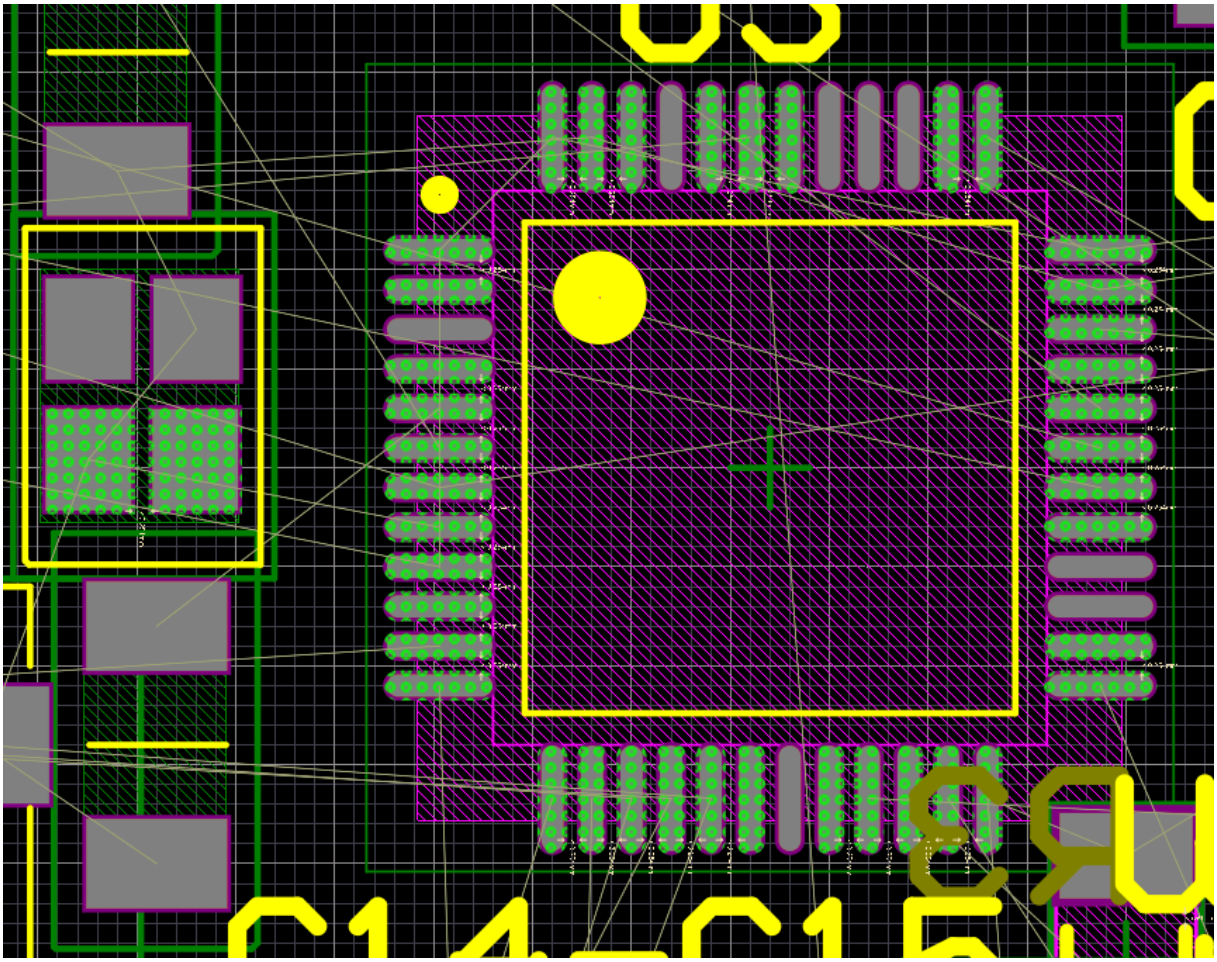
4.3 PCB routing

First, you need to specify rules: **Design → Rules**

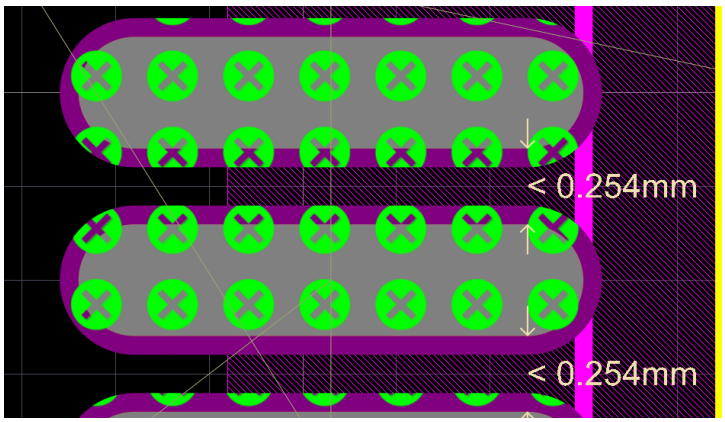
For example, from **Rules** tab → **Routing → Width → Width,** Min width is specified as 0.254mm (If it is not specified in mm, but mills, close Rules tab and press **Q** to change metric)

It is an ideal option to use min width: 0.254mm, max width: 0.3mm, preferred width: 0.3mm

In the PCB design (2D view), there might be errors like in the picture below that is green cross signs.

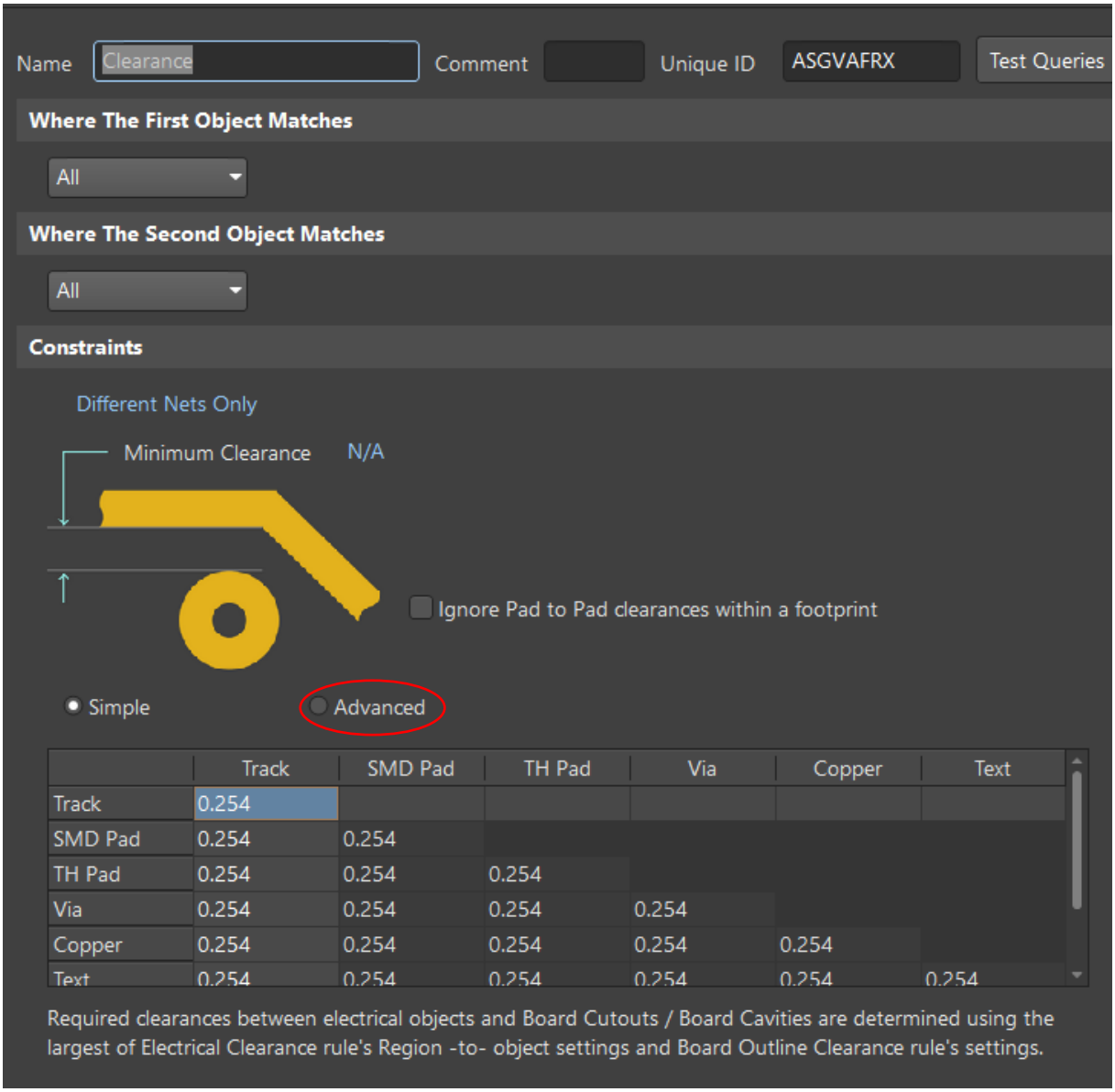


When we zoom in a little, the error is shown below:



The range between two legs is less than 0.254mm is told by Altium.

To solve this problem: **Design → Rules → Electrical → Clearance → Clearange**

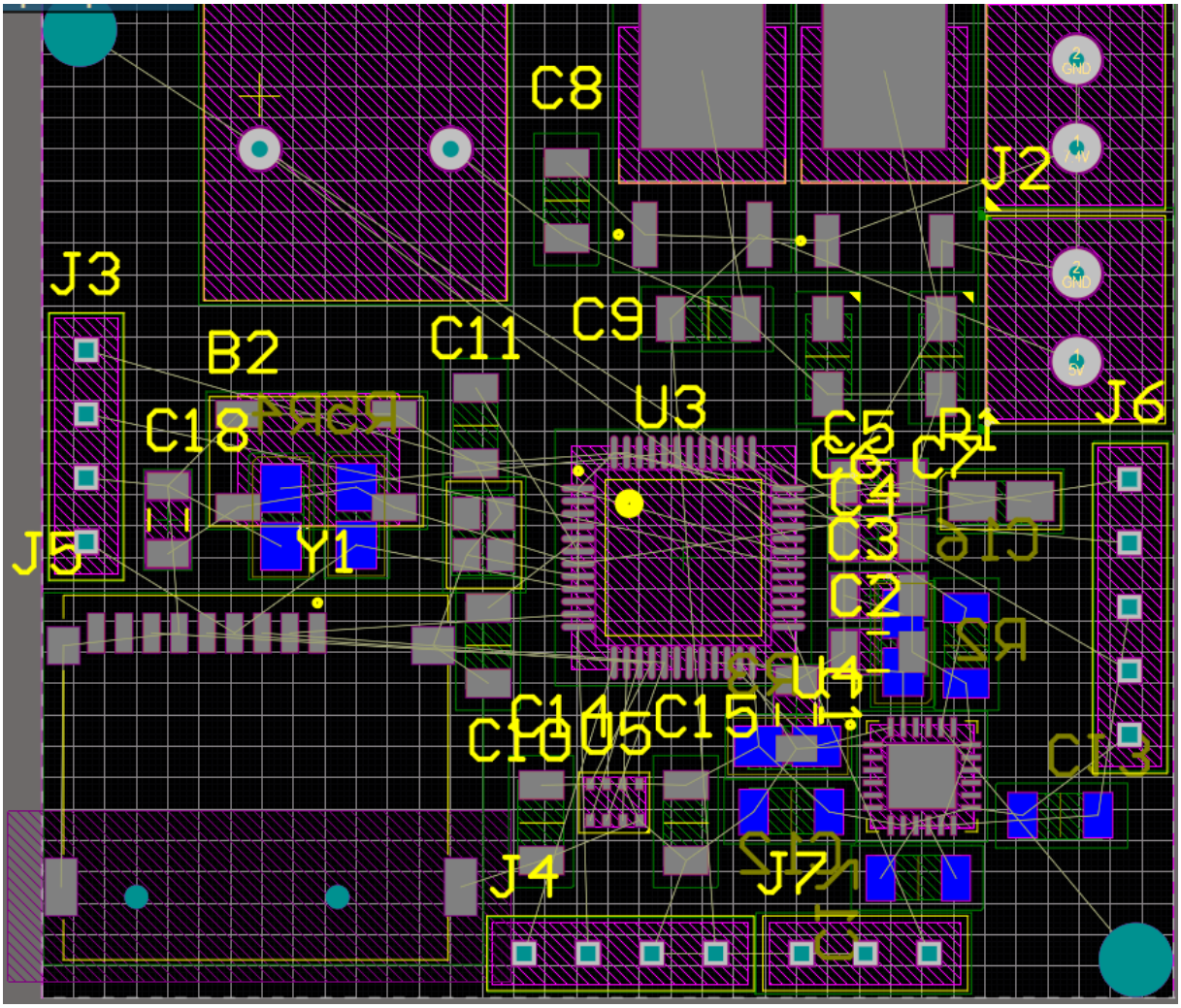


	Track	SMD Pad	TH Pad	Via	Copper	Text
Track	0.254					
SMD Pad	0.254	0.254				
TH Pad	0.254	0.254	0.254			
Via	0.254	0.254	0.254	0.254		
Copper	0.254	0.254	0.254	0.254	0.254	
Text	0.254	0.254	0.254	0.254	0.254	0.254

We can see, there are parameters which indicates ranges between for example, Track vs Track, SMD Pad vs Track, SMD Pad vs SMD.

However, before change dimensions, you need to know your PCB manufacturer production parameters. For example, your manufacturer can produce legs between 0.152mm range of each other.

When you apply proper changes to clearances, it will solve the problem that can be seen on the photo below:



We can add path with **CTRL + W (wire).** When we click any leg with CTRL + W, it shows the path you need to follow.

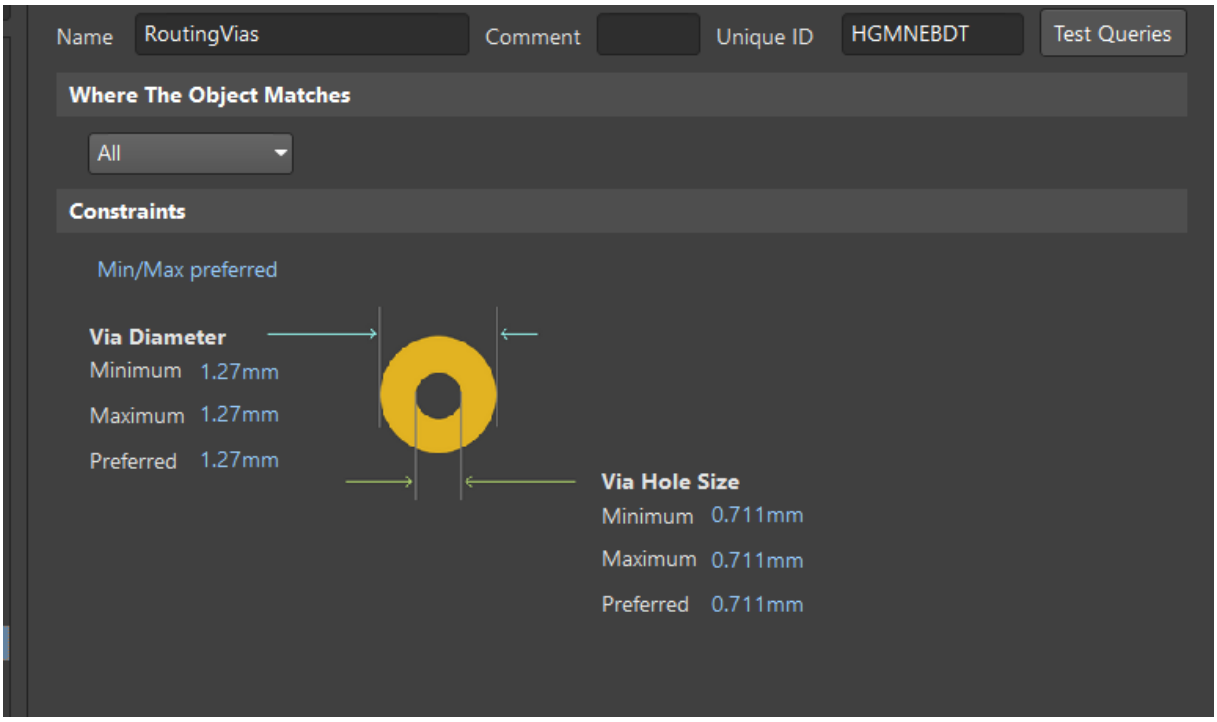
4.4 Via

Via is used when there is no possible path to the path because of crossing two or more wires to avoid crossing each other.

4.4.1 How to add Via to PCB

When we draw a path we press **+**, it moves to other layer, press **TAB** to adjust Via's hole and diameter. If it does not change, Altium does not allow you to change it because of rules.

To change rules: **Design → Rules → Routing → Routing Via Style → RoutingVias**



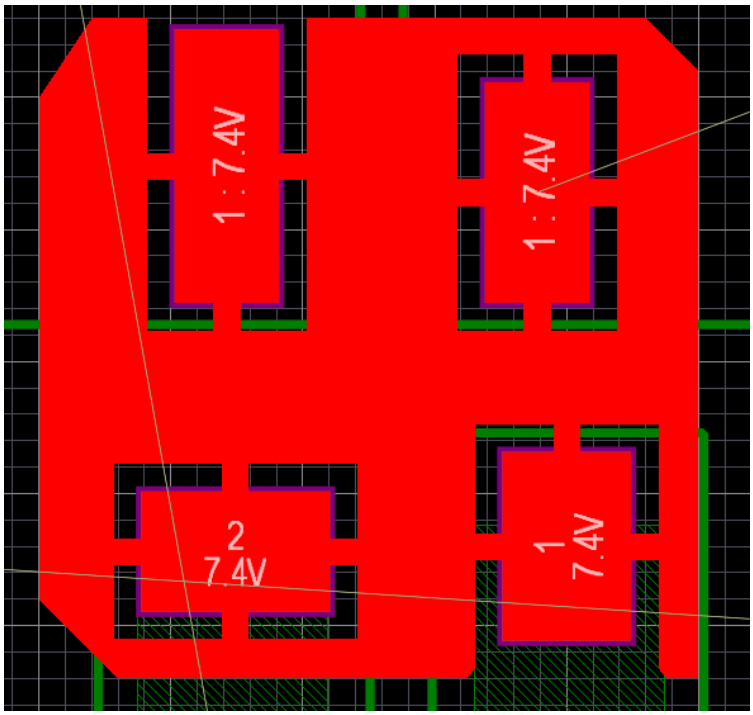
It is again depends on your manufacturer, however to give an example

Via Diameter: Min 0.2mm, Max: 1.27mm Preferred: 0.6mm

Via Hole Size: Min: 0, Max: 0.9mm, preferred: 0.3mm

4.5 How to add Pour

What is pour? Pour is used when there is an area that includes multiple power pins (should be same power level) that required many power lines. Instead, we can use **Pour** to feed those pins and reduce power losses thanks to increasing area.

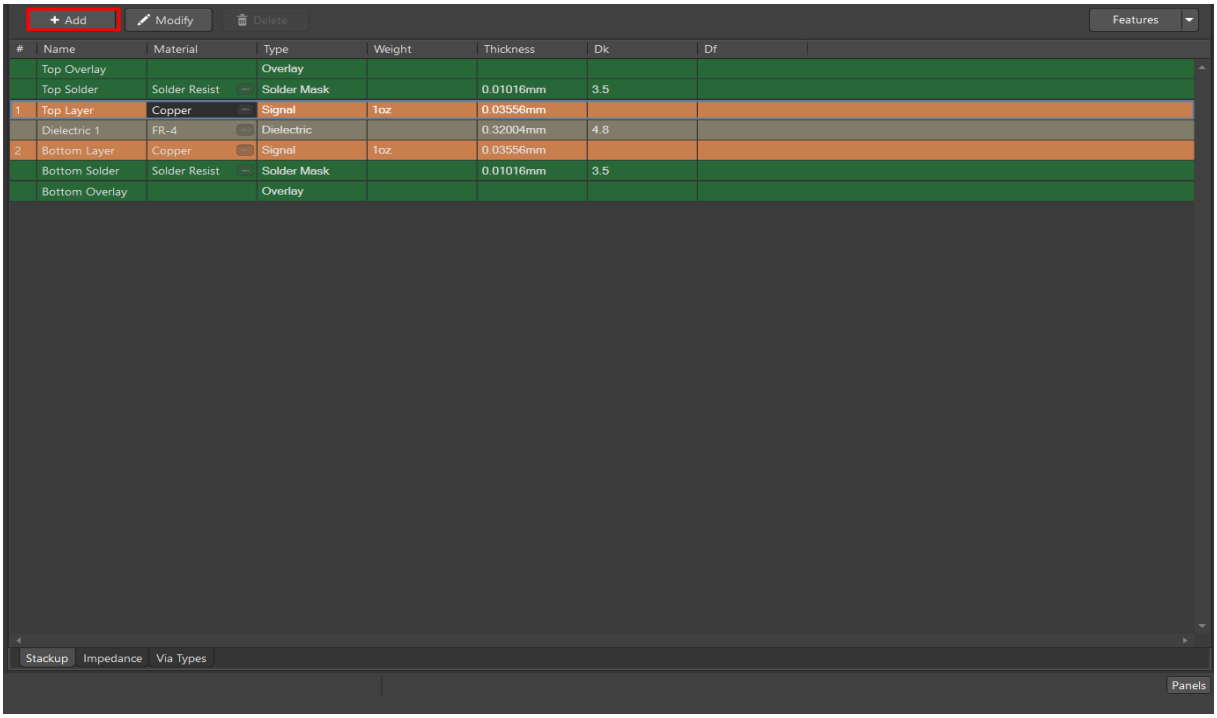


It is a pour example. How to pour specific area: **P → Polygon Pour → Select the area → Right Click at the end → ESC → Click Poured area →** Select **Net → In this example Net: 7.4V**

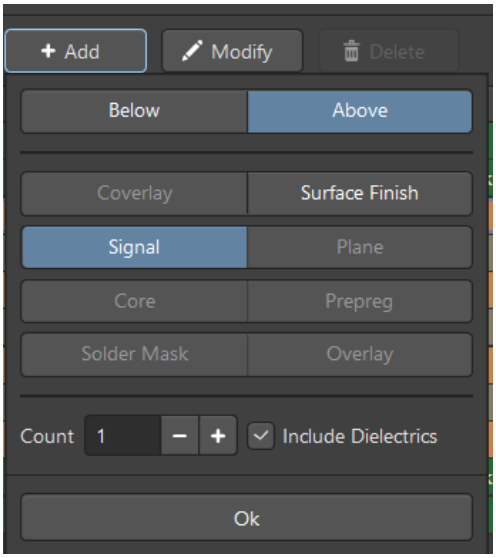
4.6 Multi-Layer PCB

You can draw double-layer PCB in normal configuration; however, if you want to increase layers:

You need to be on .PcbDoc→Design→Layer Stack Manager

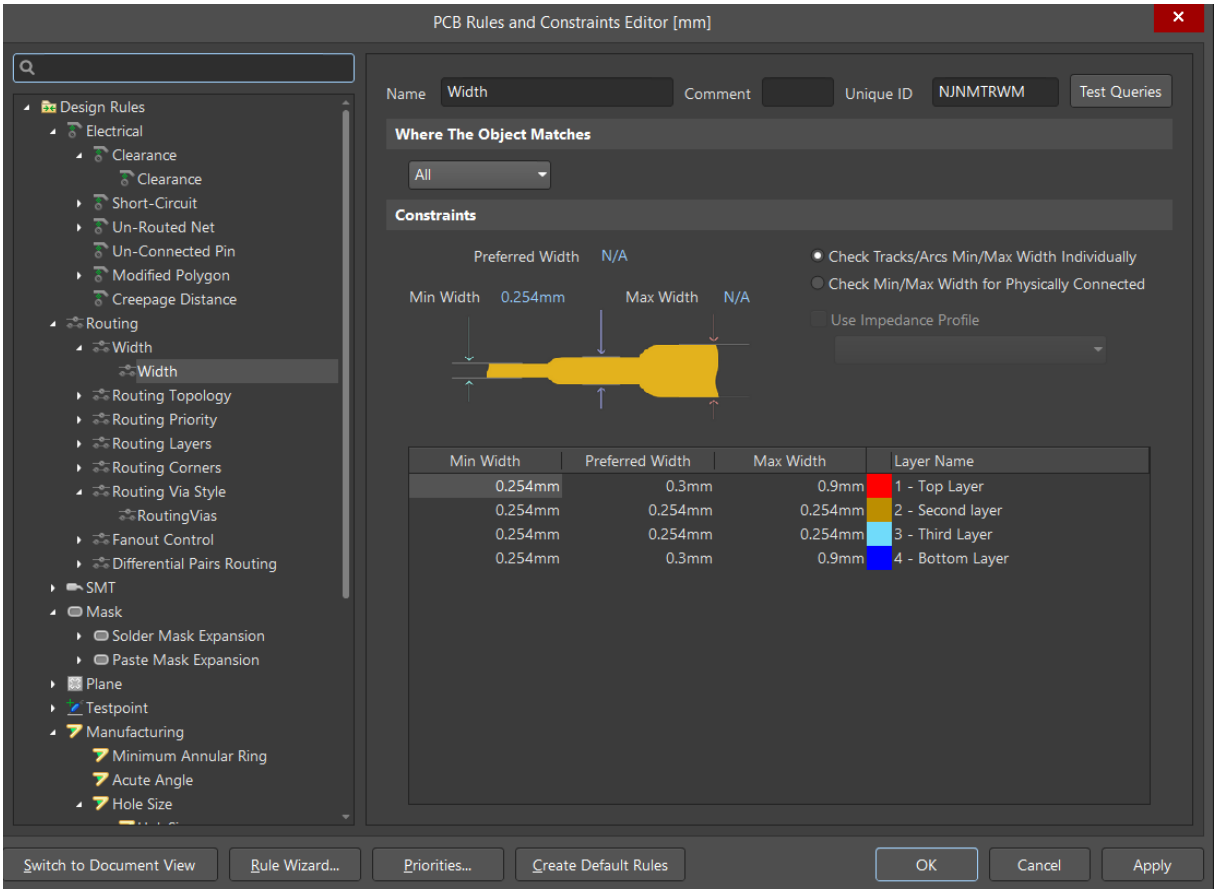


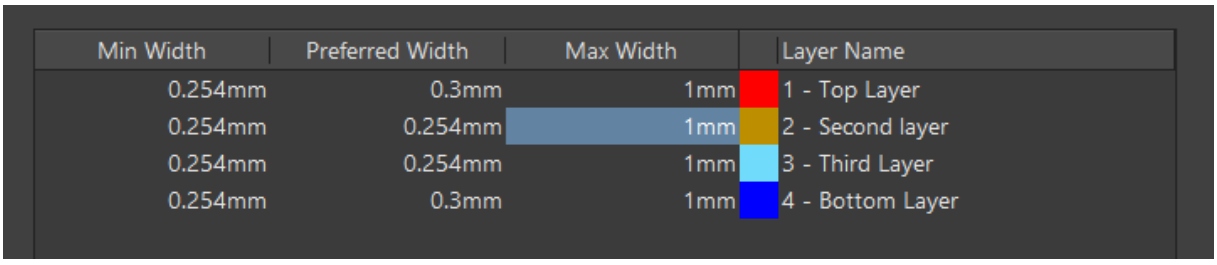
Click Add → Signal → Ok : You can add below or above with multiple counts.



We can adjust wire thickness of new added layers:

Design → Rules → Width





4.7 Design Rules

Design rules are used to check if there is a design errors or not.

To check rules: **Tools → Design Rule Check – Run Design Rule Check**

4.7.1 Rules

If there is any error with the rules, it might be need to change/adjust rules to remove/eliminate errors. To see all rules:

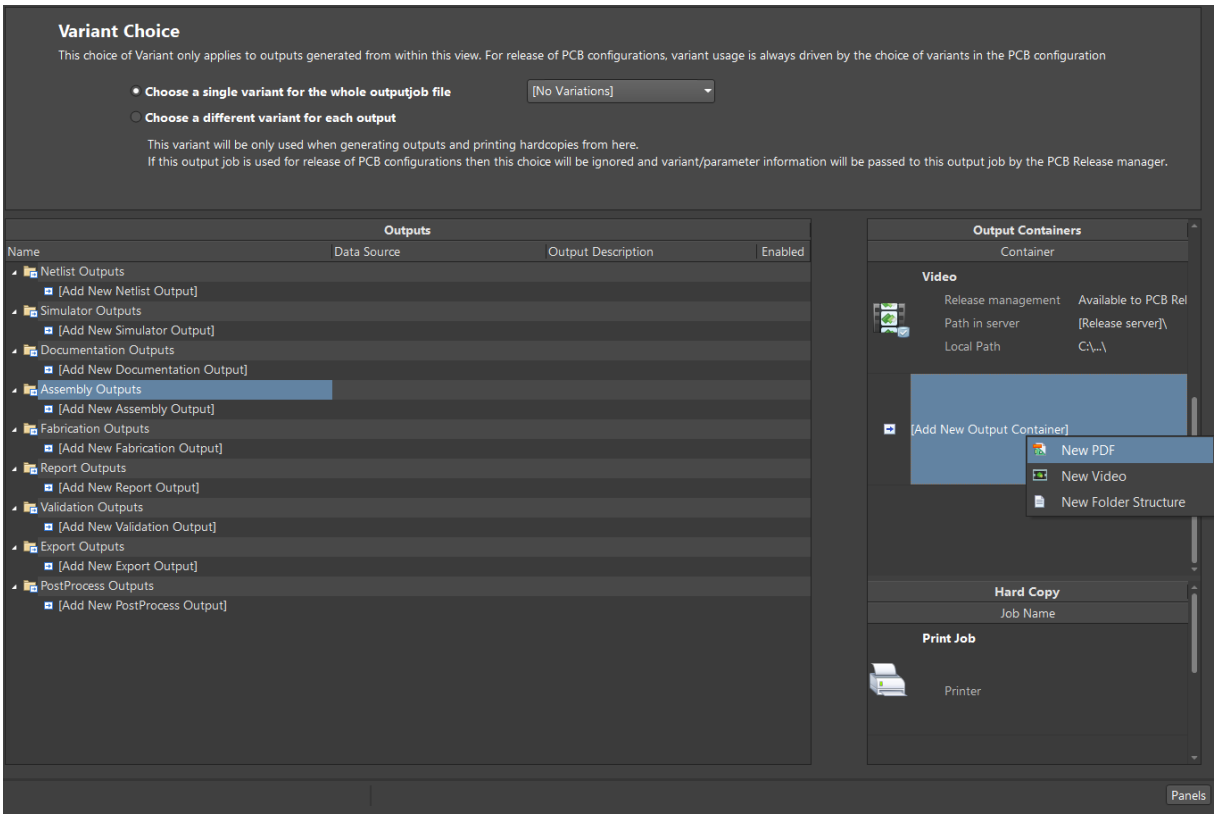
You need to be on **.PcbDoc page, Design → Rules**

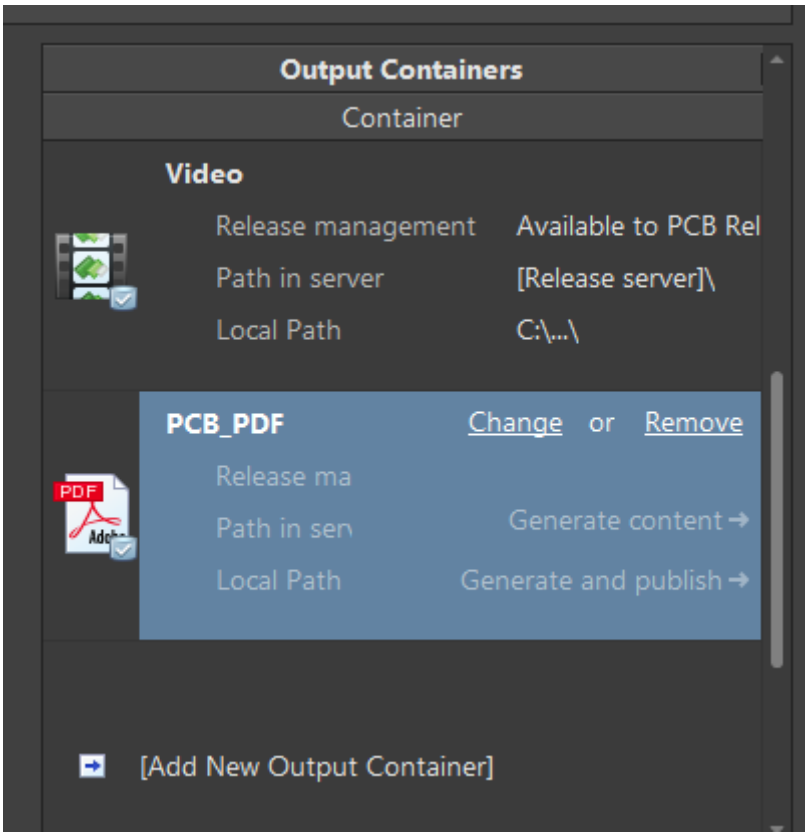
4.8 Exporting PCB

We can export PCB file in multiple forms. One of them for export as a PDF file:

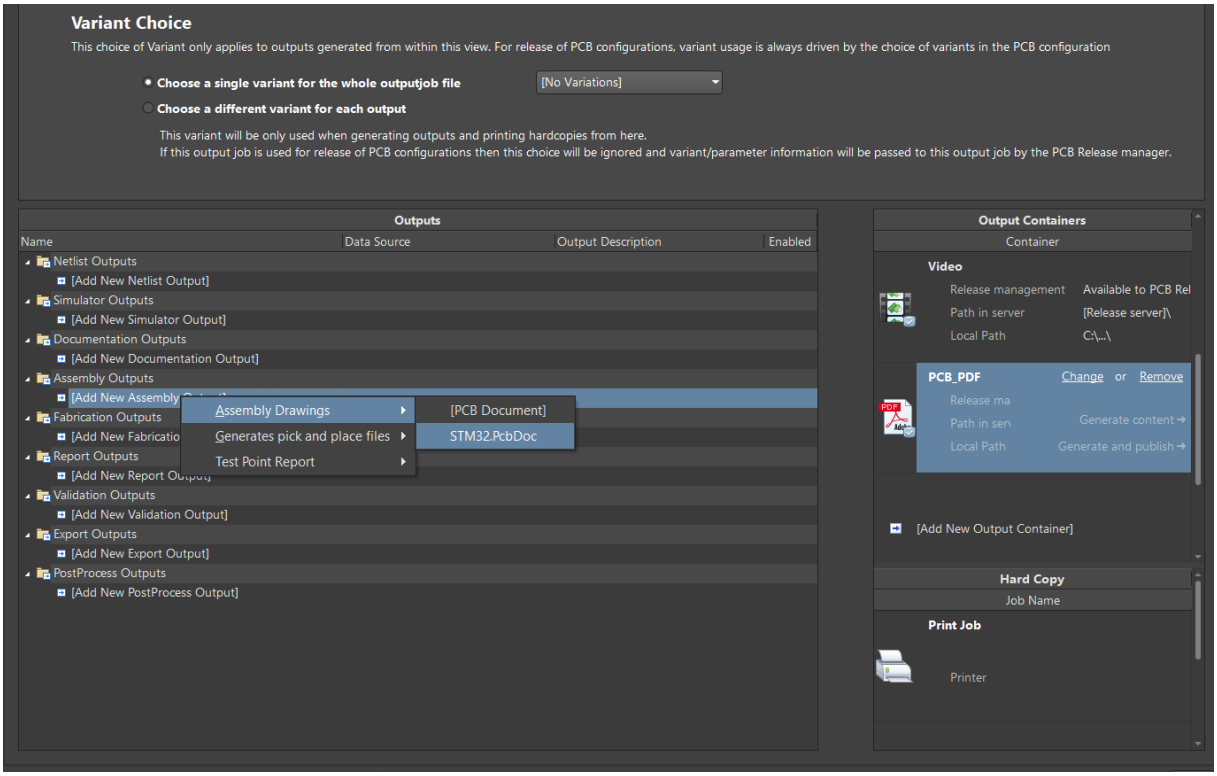
4.8.1 Exporting as a PDF file

Click right on the current project → Add New to Project → Output Job File

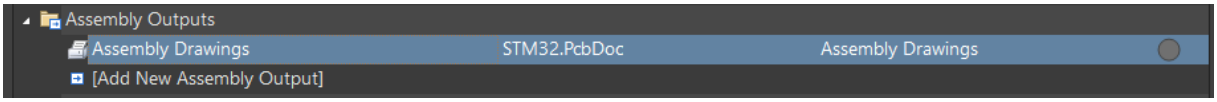




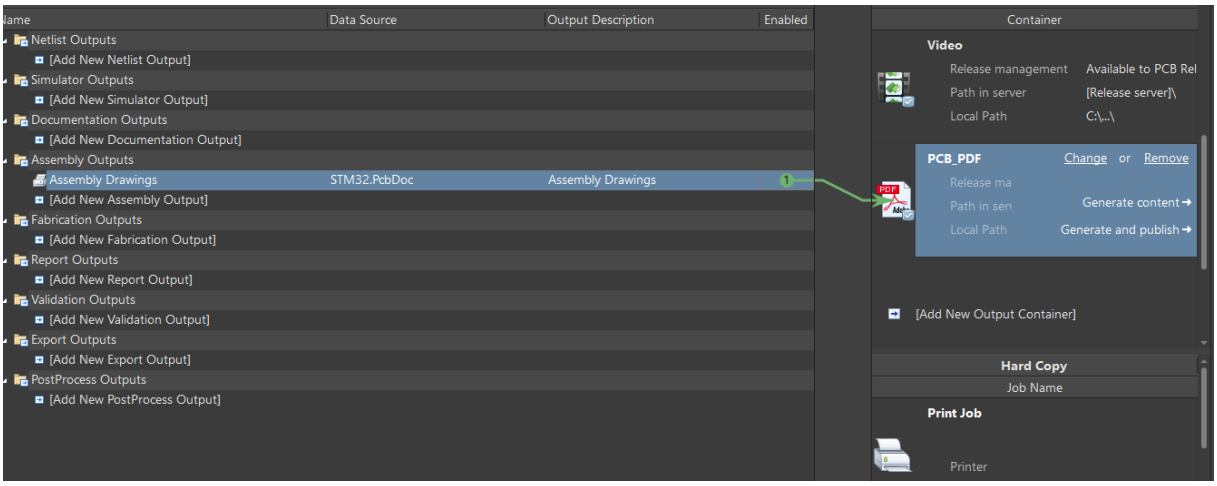
Then, you need to add assembly drawings:



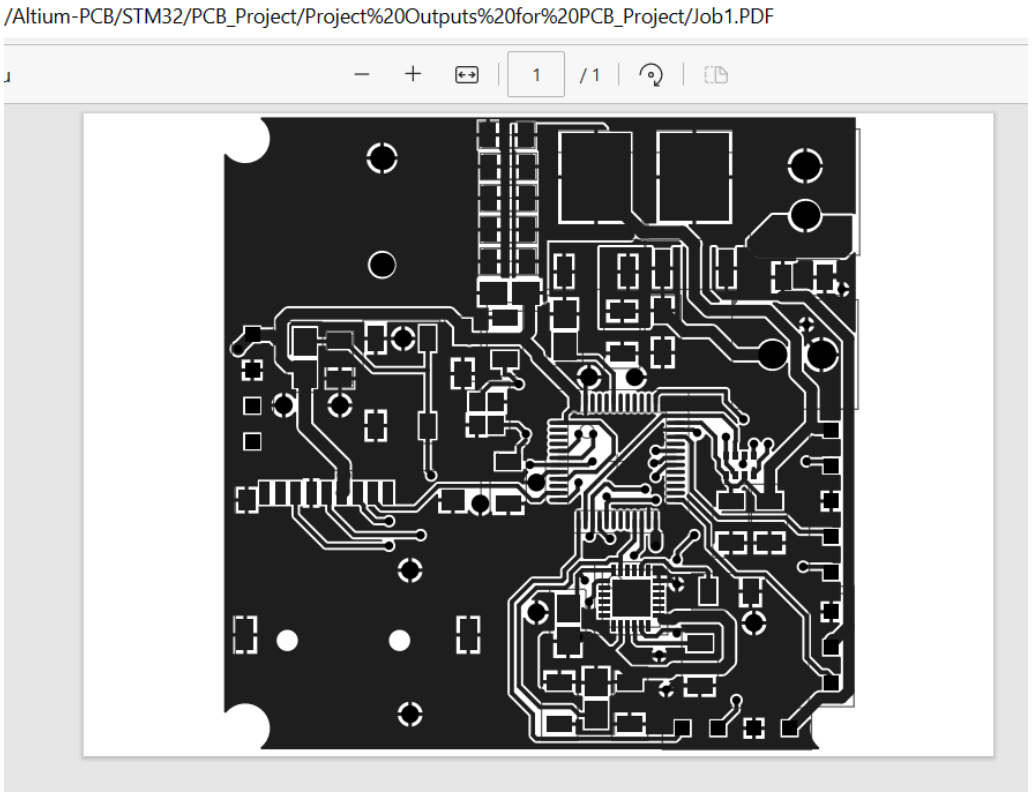
After added assembly drawings, click double. It will open **Preview PCB** page



After set-up the preview PCB page click **''OK''** and **Enabled**

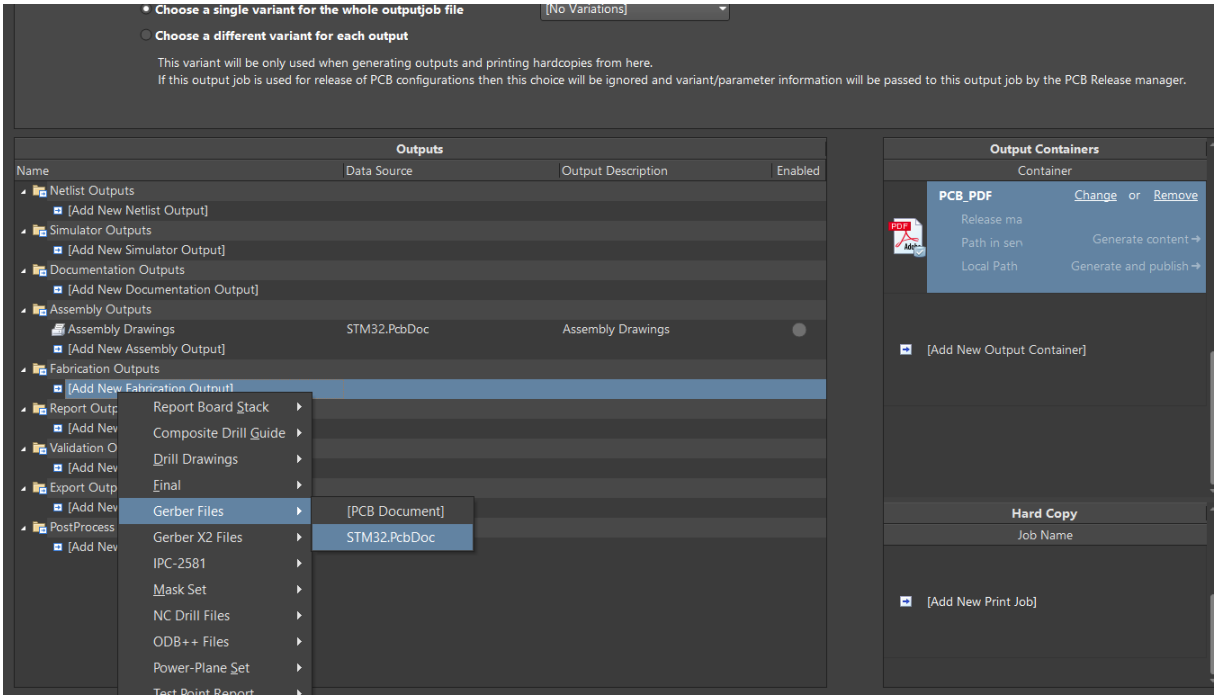


Click **''Generate content''** and it will generate PDF file of PCB to do hand-print circuit:



4.8.2 Exporting as a GERBER file

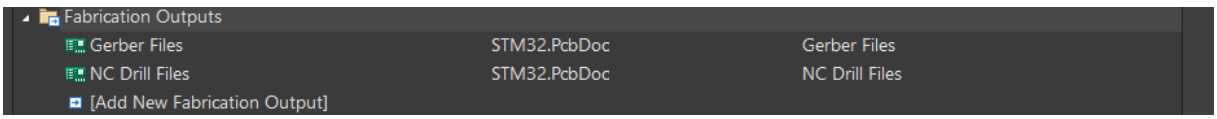
Click right on the current project → Add New to Project → Output Job File → Fabrication Outputs → Click Right → Gerber Files → projectname.PcbDoc



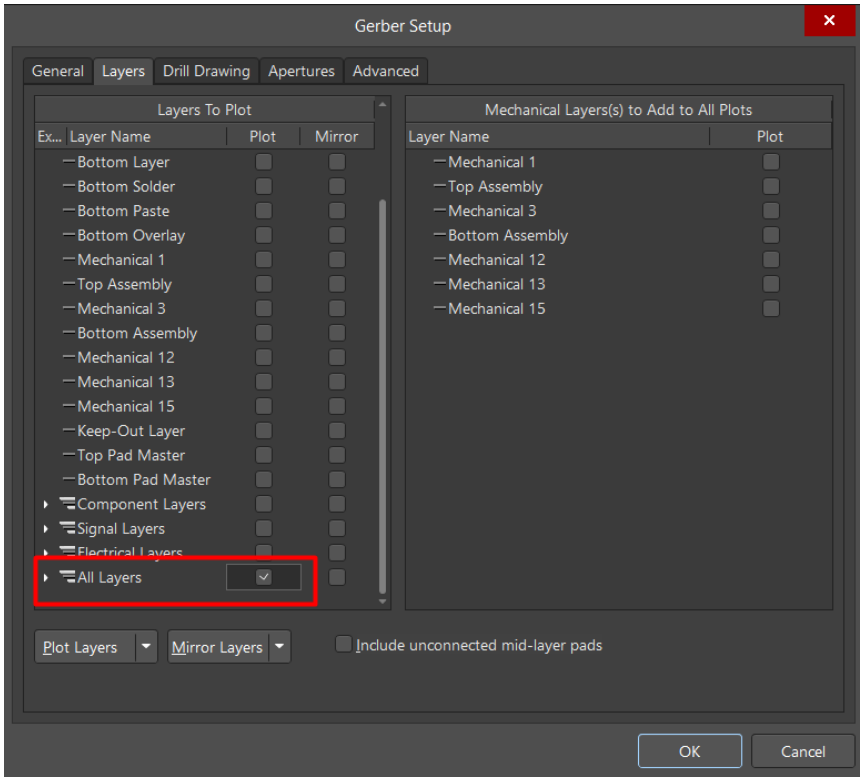
To add NC Drill Files again:

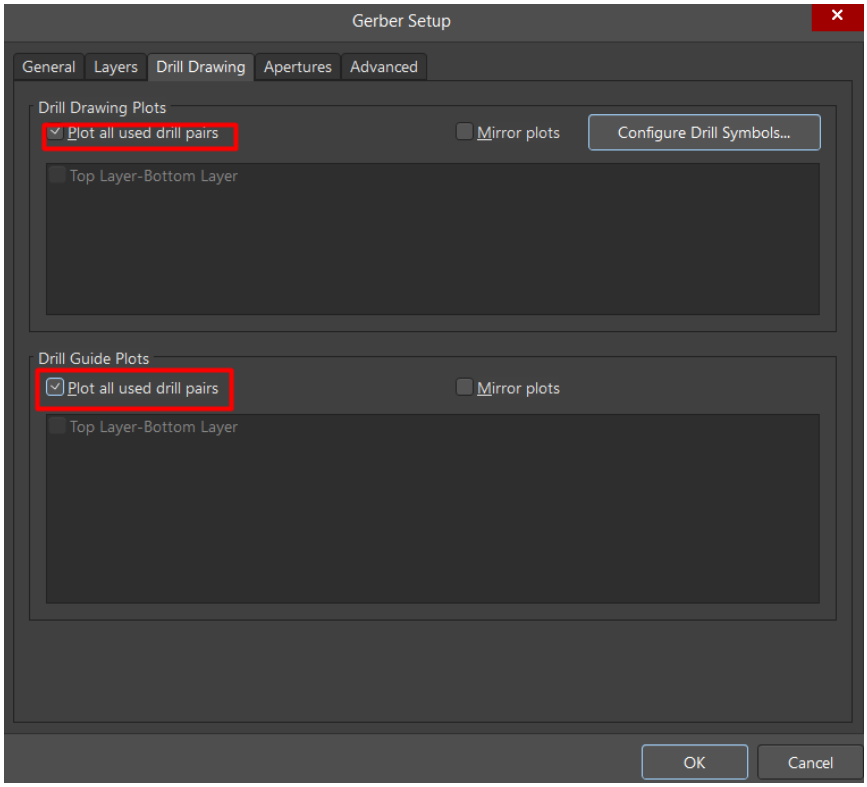
Fabrication Outputs → Click Right → NC Drill Files → projectname.PcbDoc

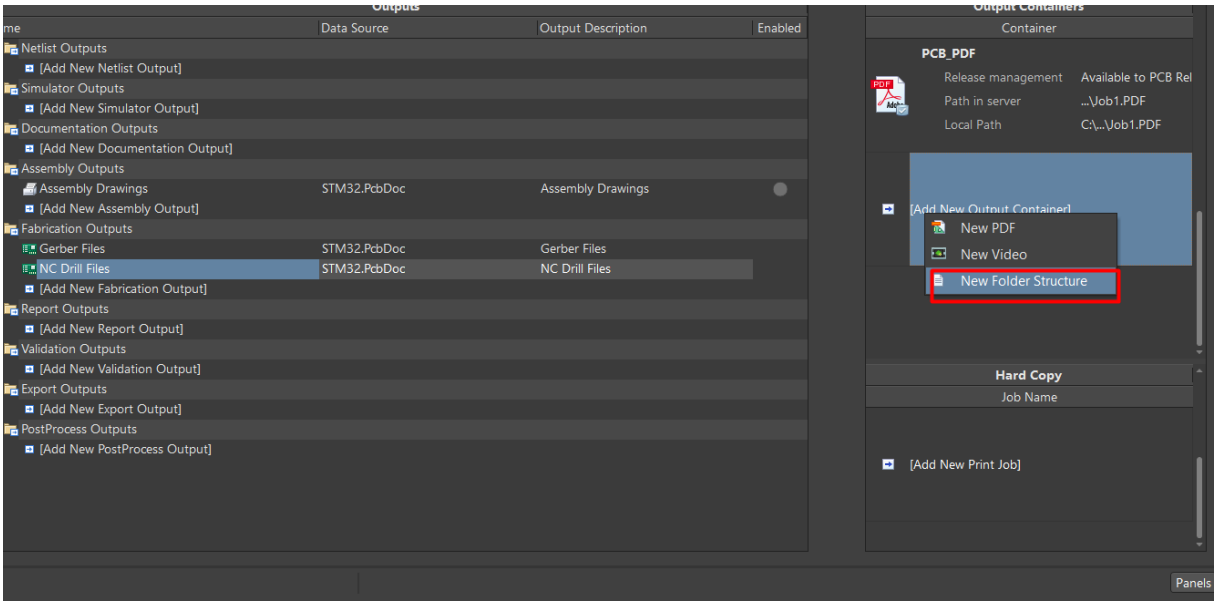
Gerber files and NC drill files are added

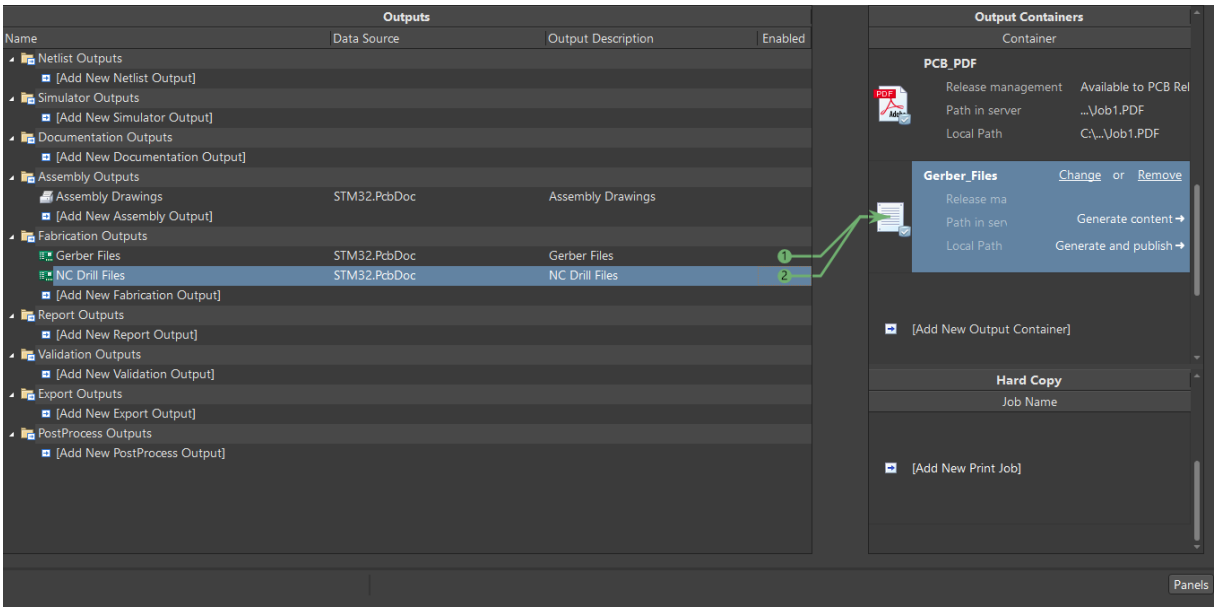


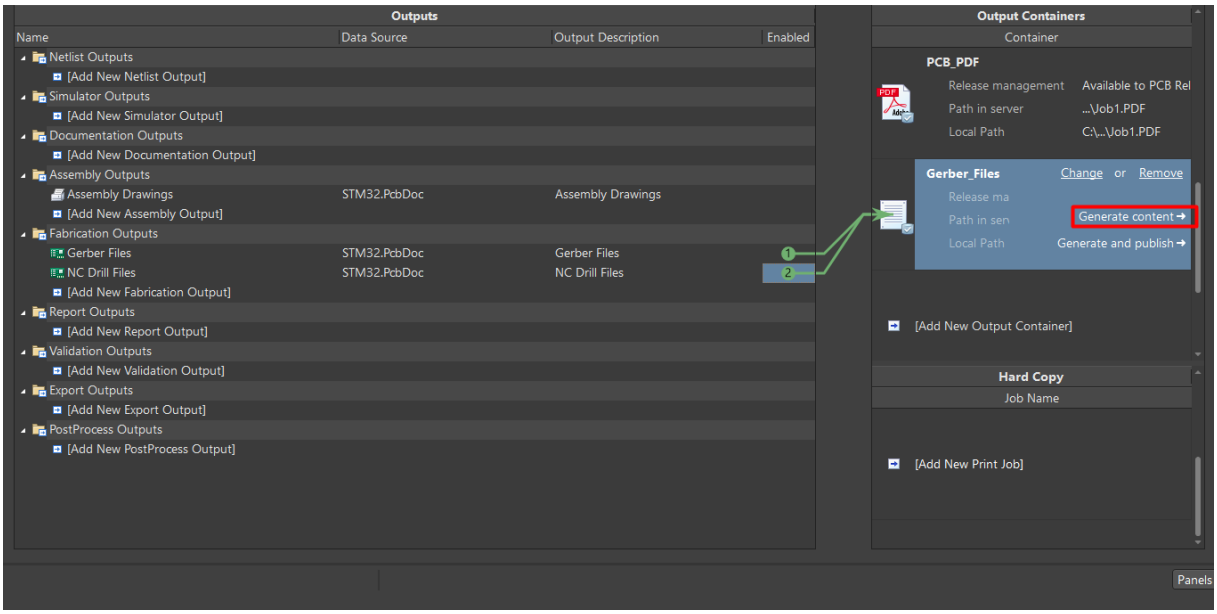
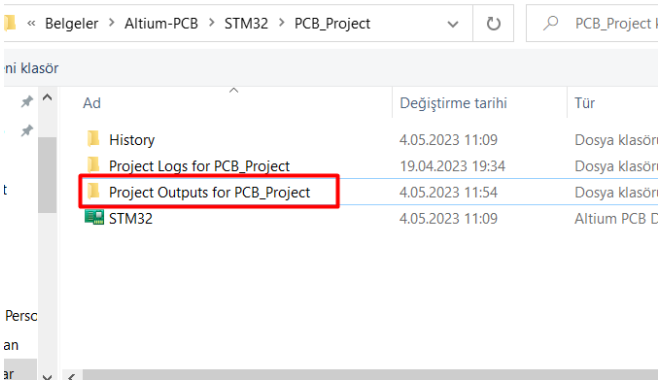
Click Gerber files, dont forget to select all layers and plot drills






me
Data Source
Output Description
Enabled
Netlist Outputs
[Add New Netlist Output]
Simulator Outputs
[Add New Simulator Output]
Documentation Outputs
[Add New Documentation Output]
Assembly Outputs
Assembly Drawings
STM32.PcbDoc
Assembly Drawings
[Add New Assembly Output]
Fabrication Outputs
Gerber Files
STM32.PcbDoc
Gerber Files
NC Drill Files
STM32.PcbDoc
NC Drill Files
[Add New Fabrication Output]
Report Outputs
[Add New Report Output]
Validation Outputs
[Add New Validation Output]
Export Outputs
[Add New Export Output]
PostProcess Outputs
[Add New PostProcess Output]
Container
PCB_PDF
Release management
Available to PCB Rel
Path in server
...\Job1.PDF
Local Path
C:\...\Job1.PDF
[Add New Output Container]
New PDF
New Video
New Folder Structure
Hard Copy
Job Name
[Add New Print Job]
Panels


Outputs
Name
Data Source
Output Description
Enabled
Netlist Outputs
[Add New Netlist Output]
Simulator Outputs
[Add New Simulator Output]
Documentation Outputs
[Add New Documentation Output]
Assembly Outputs
Assembly Drawings
STM32.PcbDoc
Assembly Drawings
[Add New Assembly Output]
Fabrication Outputs
Gerber Files
STM32.PcbDoc
Gerber Files
NC Drill Files
STM32.PcbDoc
NC Drill Files
[Add New Fabrication Output]
Report Outputs
[Add New Report Output]
Validation Outputs
[Add New Validation Output]
Export Outputs
[Add New Export Output]
PostProcess Outputs
[Add New PostProcess Output]
Output Containers
Container
PCB_PDF
Release management
Available to PCB Rel
Path in server
...\Job1.PDF
Local Path
C:\...\Job1.PDF
Gerber_Files
Change
or
Remove
Release ma
Path in ser
Generate content
Local Path
Generate and publish
[Add New Output Container]
Hard Copy
Job Name
[Add New Print Job]
Panels



After click generate content→ it will create GERBER files on your project file like this: