Manufacturing Files Generation

ELEN90053 - Electronic System Design

This document will guide you through the process of generating the necessary files for the manufacturing of your PCB.
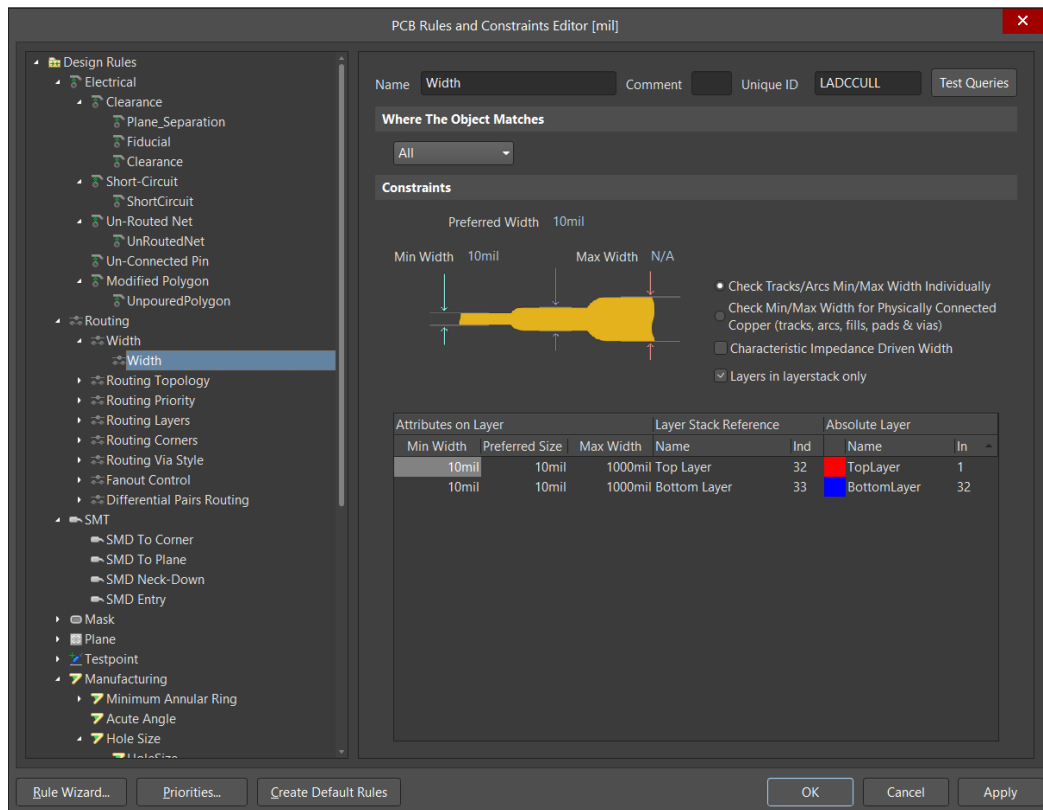
Checking your design

Before you start, you want to check if your design is correct and meets manufacturing specifications.

Step 1: Design Rules

To make sure the rules are properly set, go to the menu **Design** and then **Rules**. Check that the minimum clearance and minimum width are set to **10mil**.
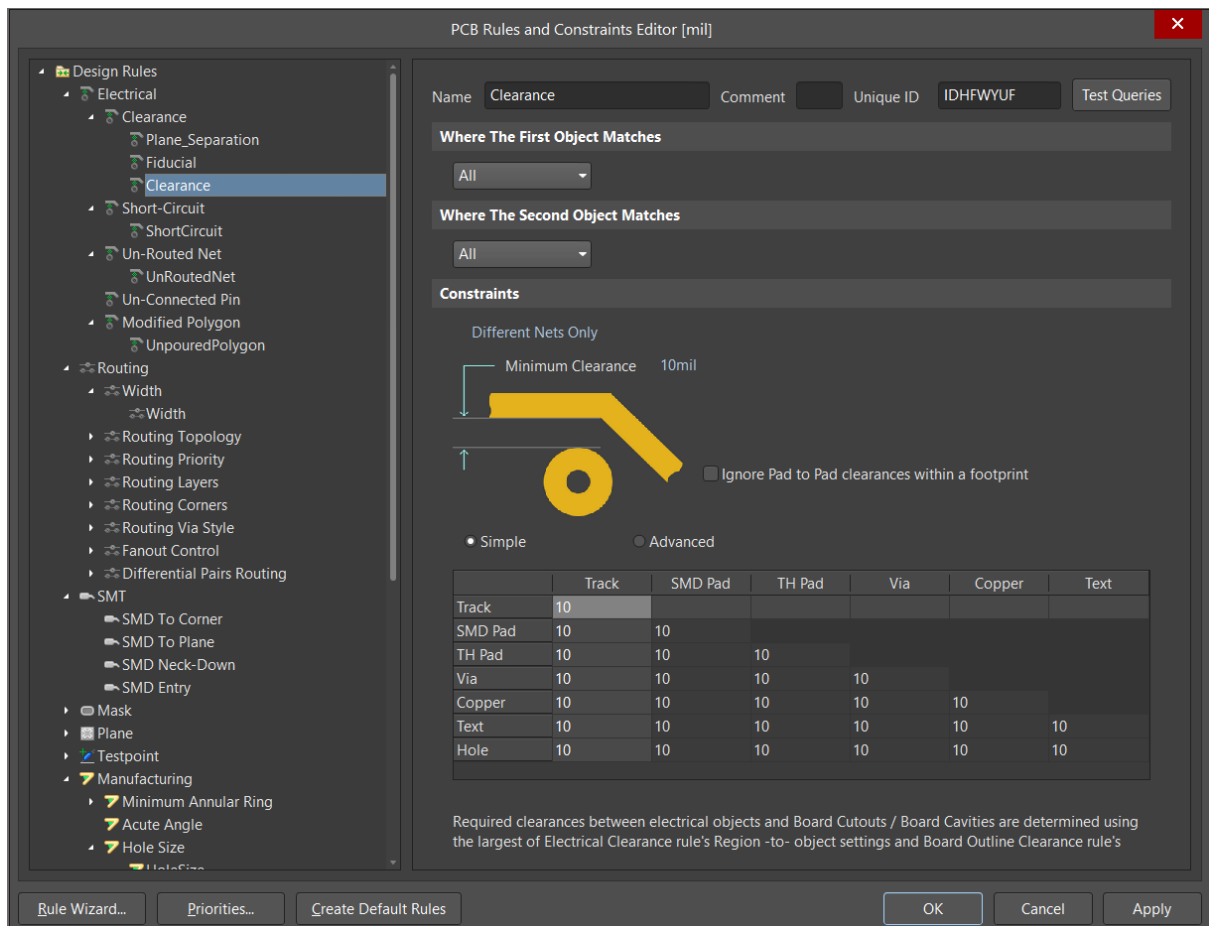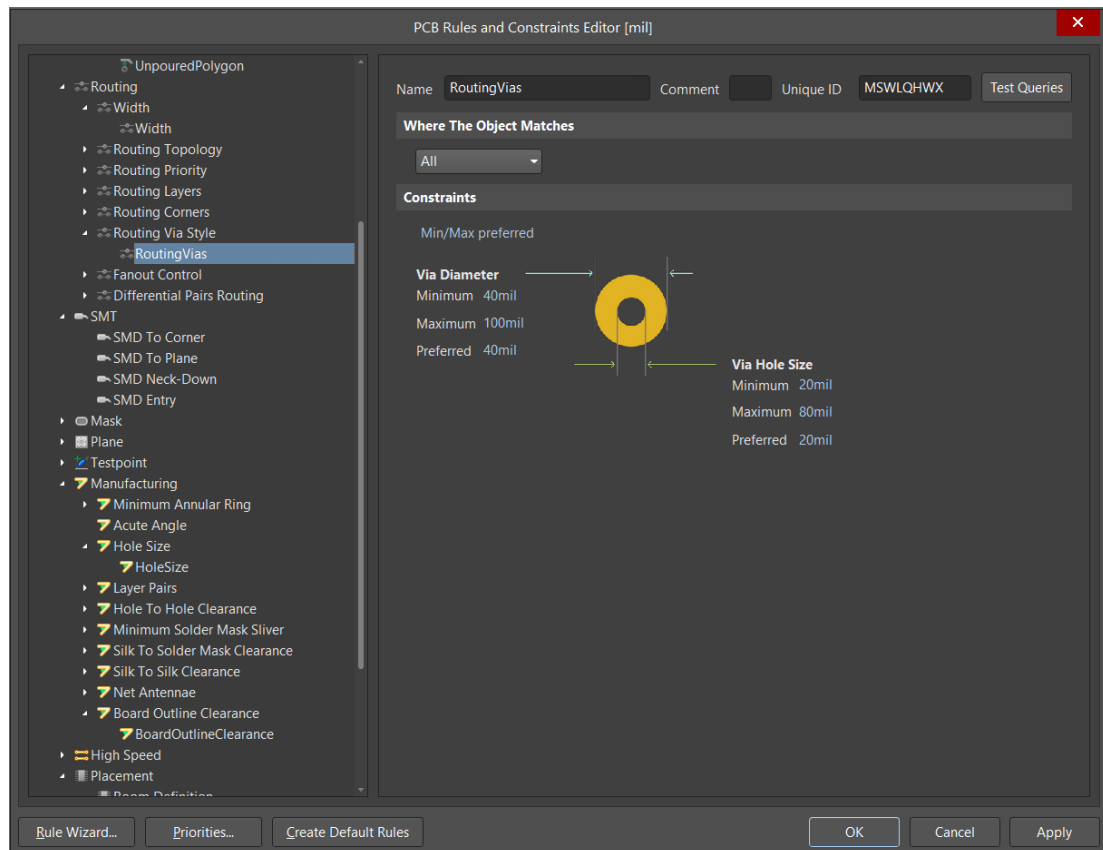
If the units are in **mm** instead **mil**, remember that you can switch them by using the shortcut **Ctrl+Q**.

Then check that the Board Outline Clearance is at least **40mil**. That will prevent copper to be too close to the edge of the board and cut accidentally.
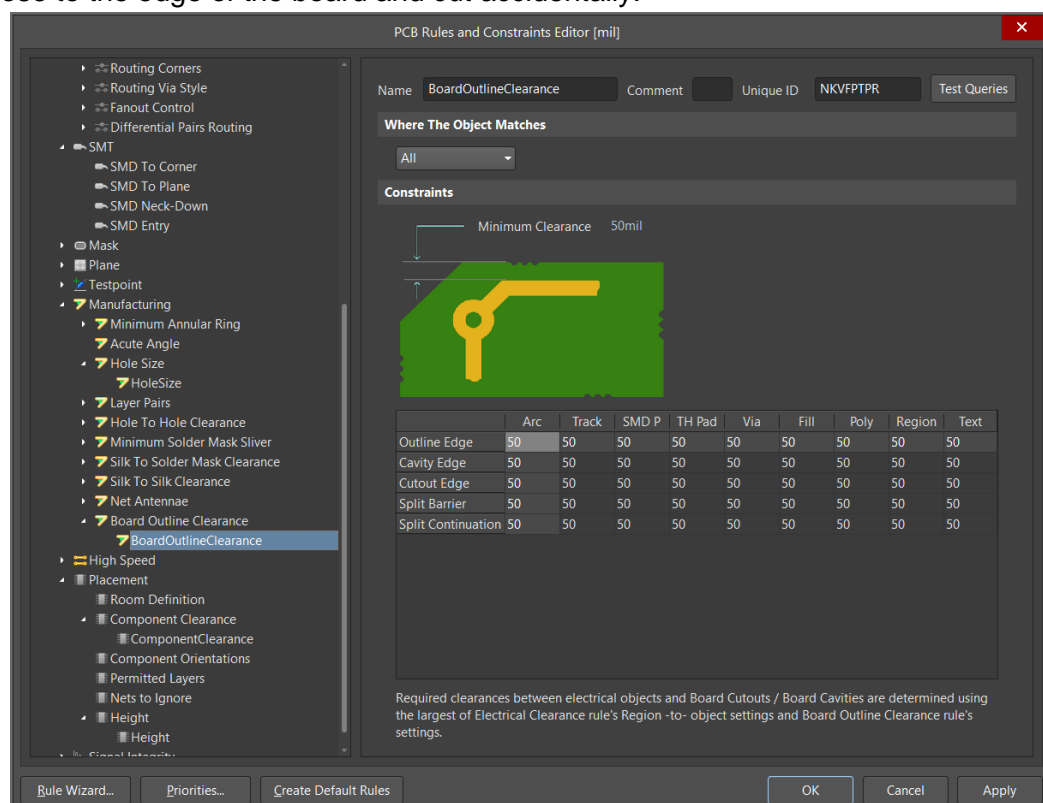
Check that the minimum via size is at least **20mil** for drill and **40mil** for diameter.

Click OK. If your design violates any of those rules (some parts of your layout start to show green) make sure you fix it before proceeding to the next step.

This is a great time to save your files

Step 2: Schematic and PCB synchronisation

It is a terrible practice to keep your schematic and PCB files out of sync, before moving to the next step make sure you have imported all the changes from your Schematic and both files are 100% consistent. Your schematic **must** be a document describing **precisely** your **final PCB** design.

Go to the menu **Design** and then **Import Changes From xxxx**. You should see a dialog like this:
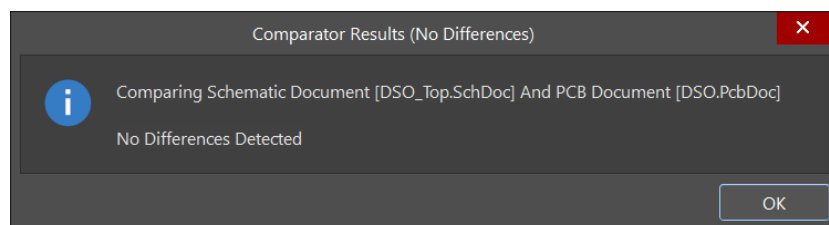
If that is not the case, work on your PCB and/or Schematic until you have no differences.

This is a great time to save your files

Step 3: Polygon Manager

You probably have many polygons in your design, at least one per each power domain as ground plane. You must ensure that your polygons are poured properly.
Go to the menu **Tools** > **Polygon Pours** > **Polygon Manager**. You should see a window like this:
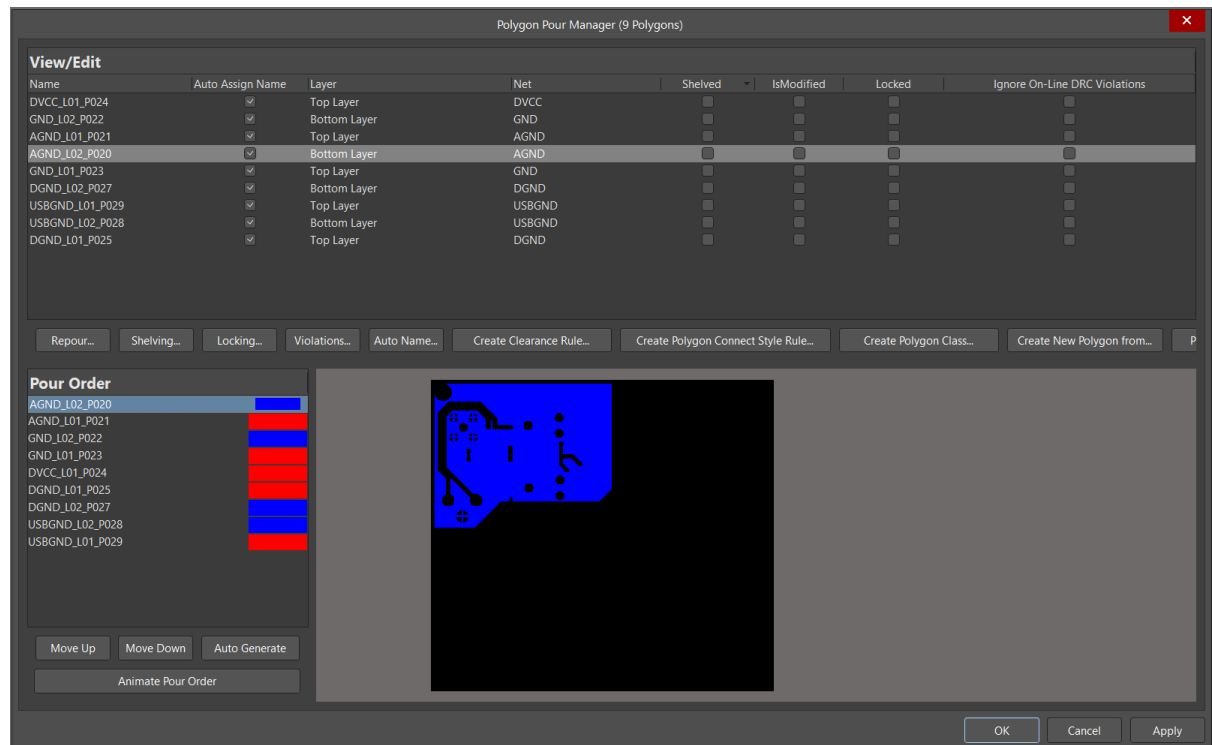
Here you want to make sure that none of your polygons is **Shelved** (hidden), that you don't have unused polygons (you may want to delete some), and check the **Pour Order**.

Pour Order sets the order in which the polygons are calculated. So, if you have a polygon inside a polygon, you want the inner polygon to be poured before the outer one, so the latter is calculated according to the correct shape of the inner one.

Once you finish and click **OK**, you should repour all polygons: **Tools** > **Polygon Pours** > **Repour All**.
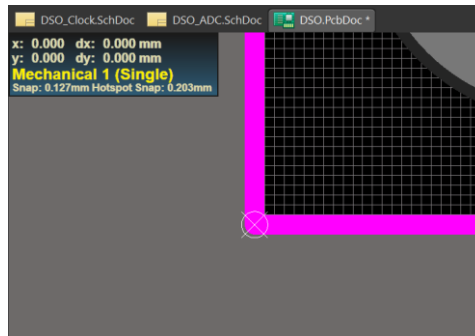
You might want to go back to the Polygon Manager to double check the order.

**This is a great time to save your files.**

Step 4: Board Shape

Make sure your board is **exactly 95mm wide and 95mm tall**:

- Set the Origin in the bottom-left corner of your board. Go to the menu **Edit** > **Origin** > **Set** and click on the bottom-left corner.

- Go to layer **Mechanical 1** (Single layer mode (Shift+S) may help) and check that you have the board outline drawn with the dimensions given above. If you don't have it, use the line tool to do it: **Place** > **Line** (10 mil is a good width for it).
- You must **not** have any other element in **Mechanical 1** apart from the outline and the 3D bodies of your components:
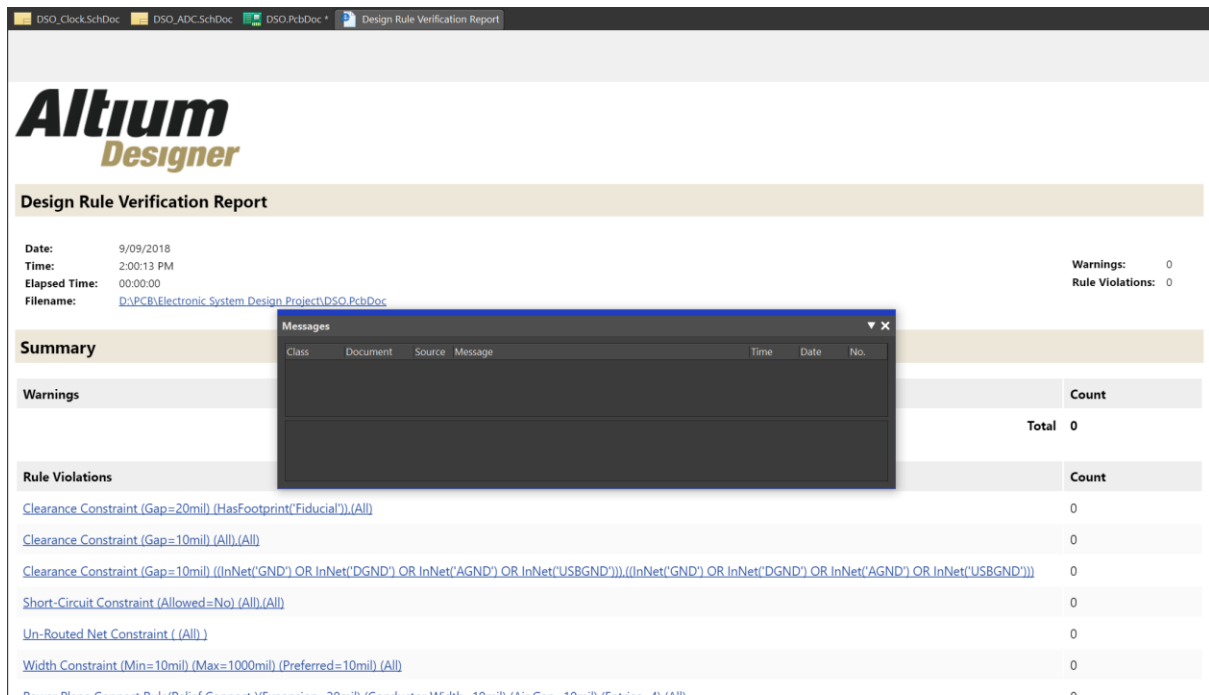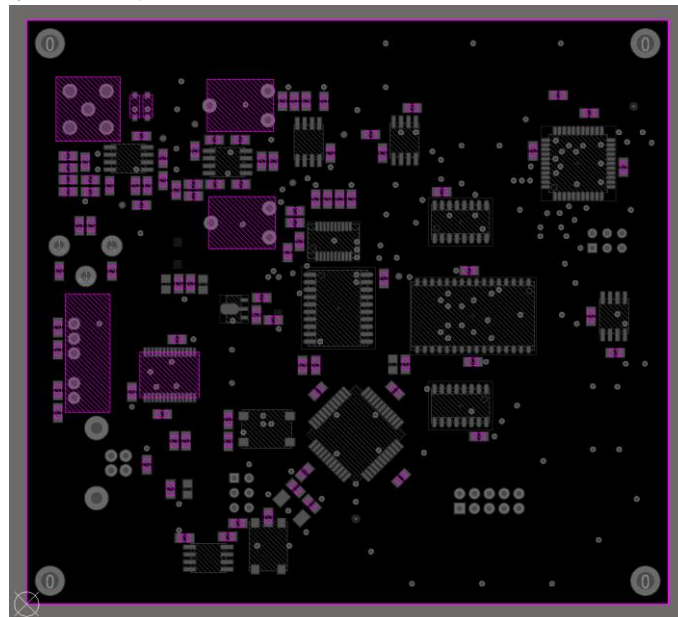

- Select the outline traces (if you select everything with the mouse in single layer mode, the outline should be the only object selected) and go to the menu **Design** > **Board Shape** > **Define from selected objects**.
- Finally, make sure that **there is no object outside the boundaries of your board**.

This is a great time to repour all polygons and save your files.

Step 5: Design Rule Check

Before deciding that your board is 100% ready, you want to run a **Design Rule Check**. Go to the menu **Tools** > **Design Rule Check** and click on the button at the bottom-left corner of the window that just appeared: **Run Design Rule Check…**

If your layout passes the rule check, you should see a report like this:

If you get any error, use the **Messages** window to click each one of them and fix your design.

This is a great time to save your files.

Phew!! Now you are ready to export your design.

Configuring Output Jobs file

Step 1: Output Job File

The best way to generate manufacturing files in Altium is by using an Output Job File, this automates the file generation process, so if you realise that you made a mistake, you can generate your files again just with a couple of clicks.

In the **Projects** panel, right-click on your project and go to **Add New to Project** > **Output Job File**:
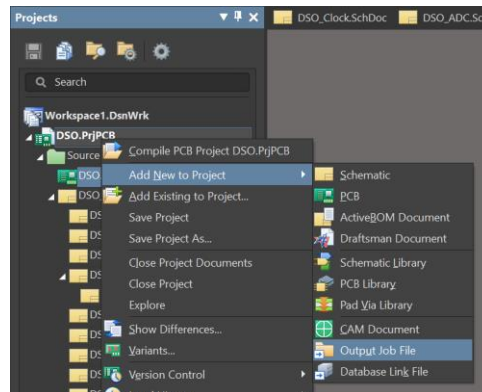
An interface like this should open for your Output Job file:
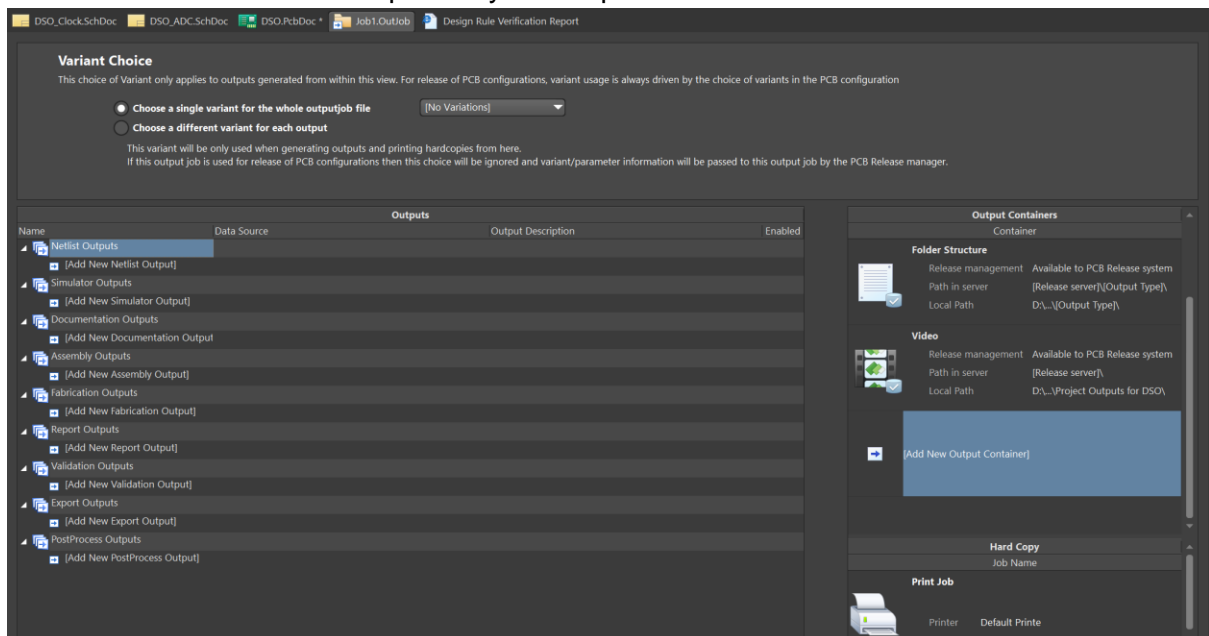
As you can see, here you can program Altium to generate all the files and documentation you might need for your project. For now we will just focus on the **Fabrication Outputs** and **Bill of Materials**.

Save the file inside your project folder with a name of your choice.

Step 2: Gerber Files

In the section **Fabrication Outputs** click on **Add New Fabrication Output**. This will show a menu where you have to select **Gerber Files** and then select the PCB document of your DSO. This will add one item to your list:

Double click on that item to configure your Gerber Files and set the parameters according to the following screenshots:
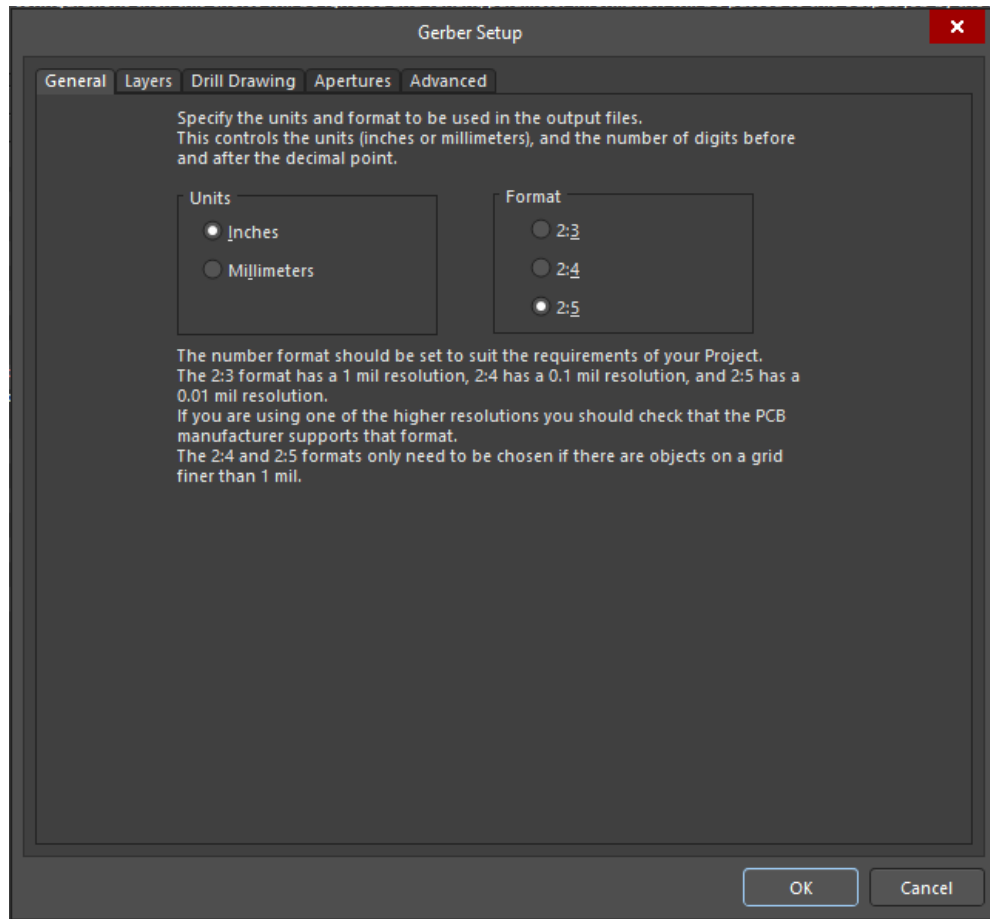
Gerber Setup

General | Layers | Drill Drawing | Apertures | Advanced

Layers To Plot

Ex...	Layer Name	Plot	Mirror
	Top Overlay	✓	☐
	Top Paste	✓	☐
	Top Solder	✓	☐
	Top Layer	✓	☐
	Bottom Layer	✓	☐
	Bottom Solder	✓	☐
	Bottom Paste	✓	☐
	Bottom Overlay	✓	☐
	Mechanical 1	✓	☐
	Mechanical 13	✓	☐
	Mechanical 15	✓	☐
	Keep-Out Layer	☐	☐
	Top Pad Master	✓	☐
	Bottom Pad Master	✓	☐
▶	Component Layers	☐	☐
▶	Signal Layers	☐	☐
▶	Electrical Layers	☐	☐
▶	All Layers	☐	☐

Mechanical Layers(s) to Add to All Plots

Layer Name	Plot
Mechanical 1	☐
Mechanical 13	☐
Mechanical 15	☐

Plot Layers ▼   Mirror Layers ▼   ☑ Include unconnected mid-layer pads

OK   Cancel

Gerber Setup

General | Layers | Drill Drawing | Apertures | Advanced

Drill Drawing Plots

☐ Plot all used drill pairs   ☐ Mirror plots   Configure Drill Symbols...

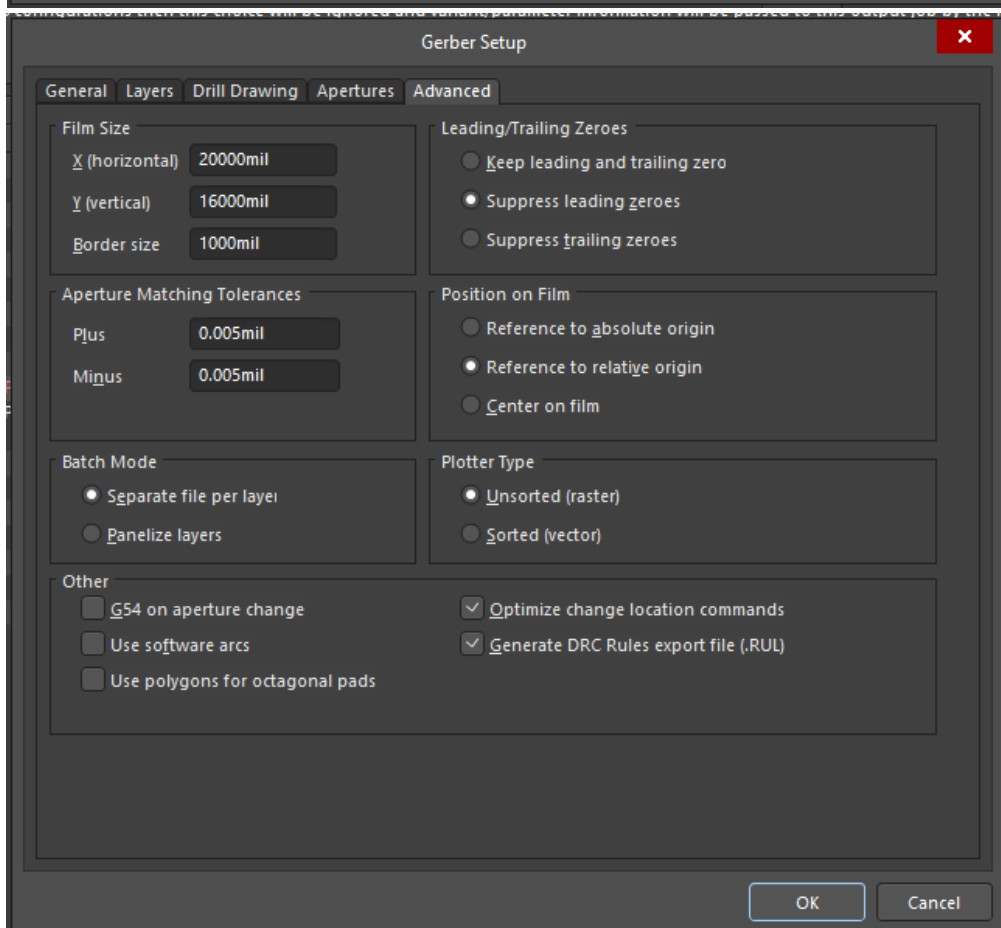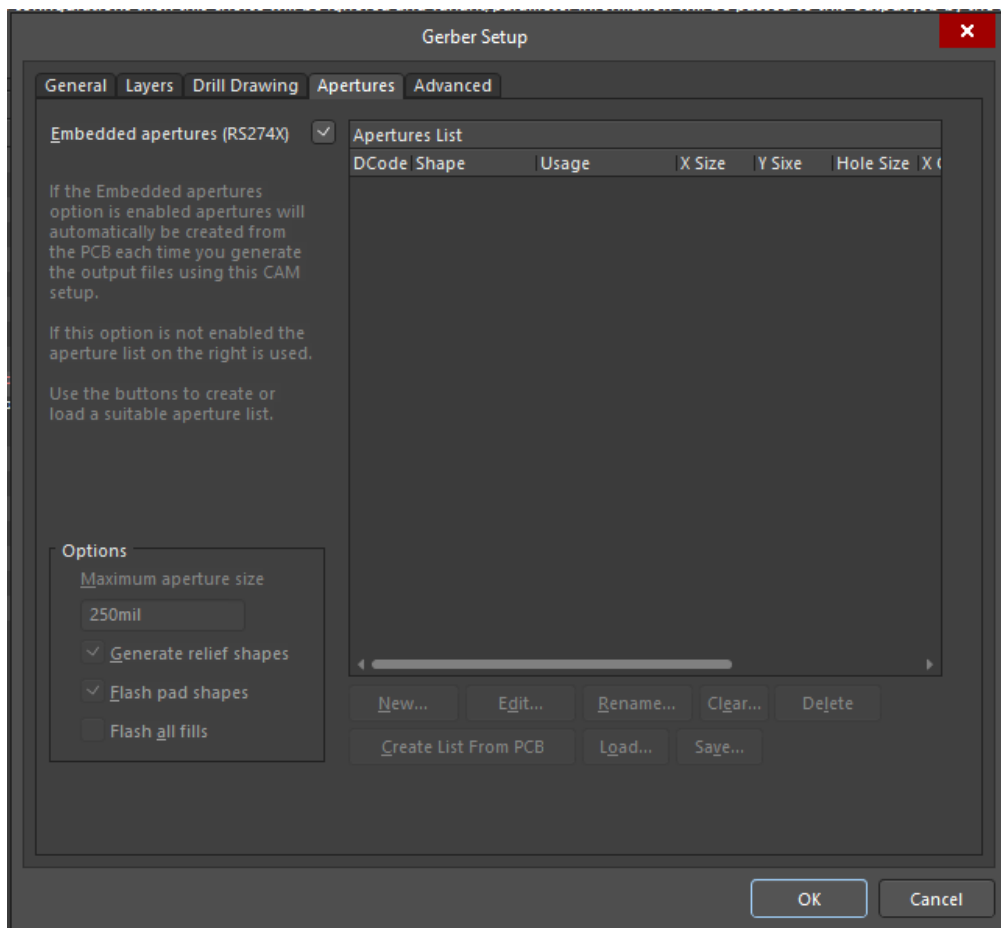☐ Top Layer-Bottom Layer

Drill Guide Plots

☐ Plot all used drill pairs   ☐ Mirror plots

☐ Top Layer-Bottom Layer

OK   Cancel

Gerber Setup

General | Layers | Drill Drawing | **Apertures** | Advanced

Embedded apertures (RS274X) ☑

If the Embedded apertures option is enabled apertures will automatically be created from the PCB each time you generate the output files using this CAM setup.

If this option is not enabled the aperture list on the right is used.

Use the buttons to create or load a suitable aperture list.

Options

Maximum aperture size

250mil

☑ Generate relief shapes

☑ Flash pad shapes

☐ Flash all fills

Apertures List

DCode	Shape	Usage	X Size	Y Sixe	Hole Size	X (

New... | Edit... | Rename... | Clear... | Delete

Create List From PCB | Load... | Save...

OK | Cancel

Gerber Setup

General | Layers | Drill Drawing | Apertures | **Advanced**

Film Size

X (horizontal) 20000mil

Y (vertical) 16000mil

Border size 1000mil

Aperture Matching Tolerances

Plus 0.005mil

Minus 0.005mil

Batch Mode

◉ Separate file per layer

○ Panelize layers

Leading/Trailing Zeroes

○ Keep leading and trailing zero

◉ Suppress leading zeroes

○ Suppress trailing zeroes

Position on Film

○ Reference to absolute origin

◉ Reference to relative origin

○ Center on film

Plotter Type

◉ Unsorted (raster)

○ Sorted (vector)

Other

☐ G54 on aperture change

☐ Use software arcs

☐ Use polygons for octagonal pads

☑ Optimize change location commands

☑ Generate DRC Rules export file (.RUL)
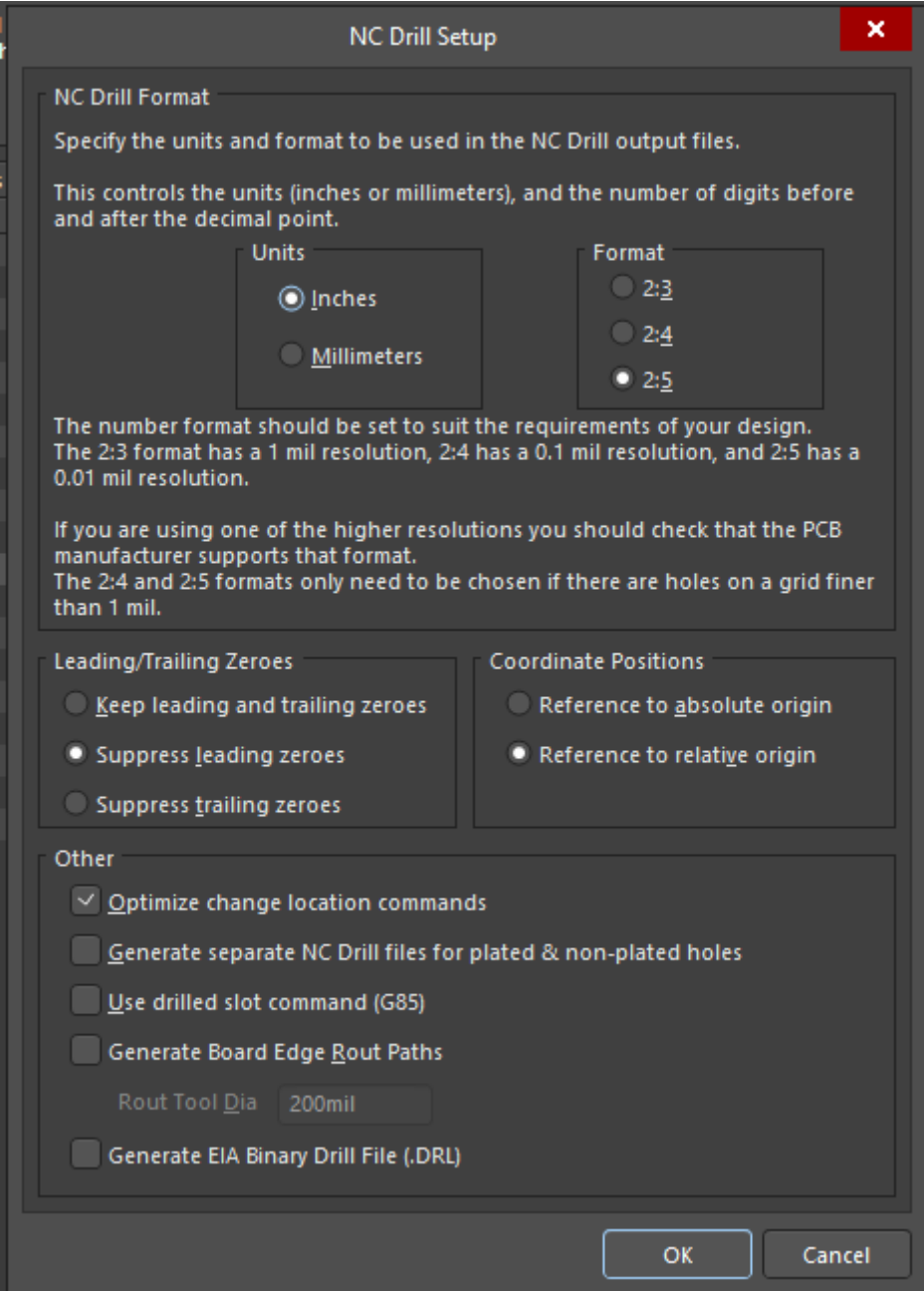
OK | Cancel

Step 3: NC Drill Files

Gerber Files only have the shape of the layers of your PCB. You also need to generate a file with the drill information, so the manufacturer can drill all the vias and holes of your PCB.

Just as you did before, click on **Add New Fabrication Output**, then **NC Drill Files** and finally select the PCB file of your DSO (make sure is the same as the one in Gerber Files).

Double click on the new item and check the parameters:

Step 4: Output Files

Now we are going to tell Altium that we want to generate the files for the items we just added. Click on the **Folder Structure** section in the right hand-side panel and then select **Gerber Files** and **NC Drill Files** by clicking on the circle at the right of each item:
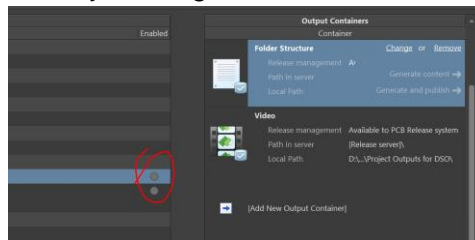
That should give you the following result:
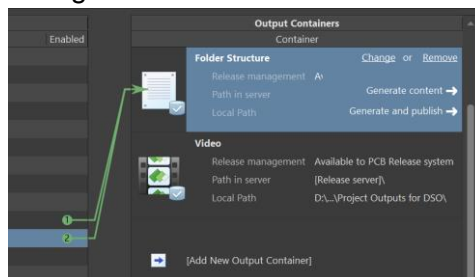
Now, in the **Folder Structure** section click on **Change** and then **Advanced** in the window that just opened:
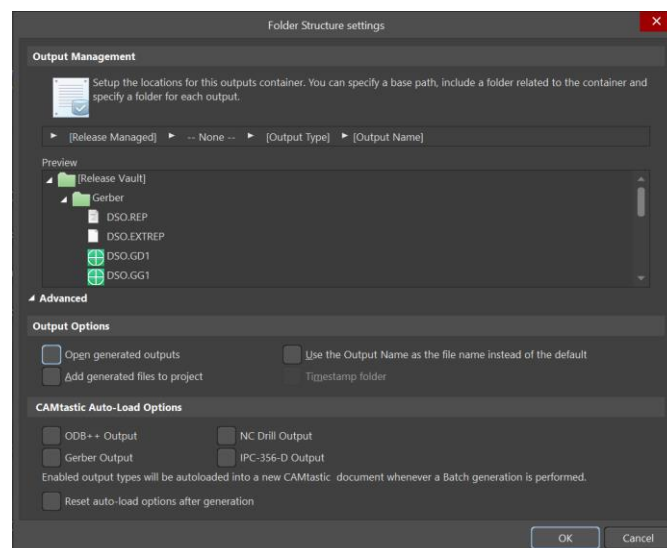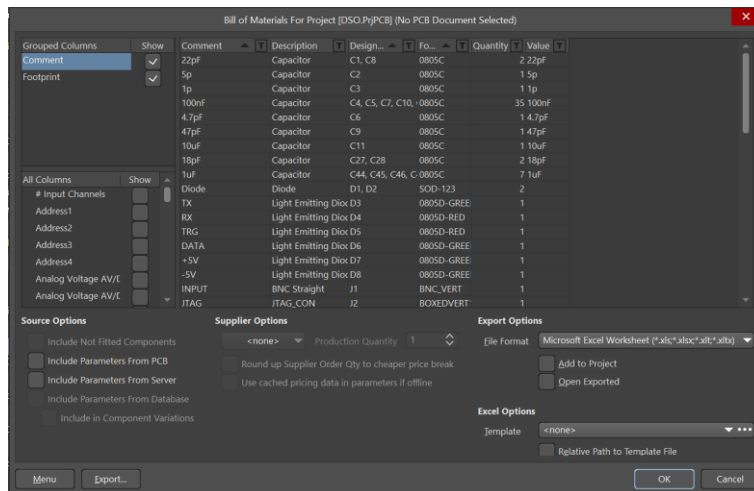

Untick **Open generated outputs** and **Add generated files to project**.

Step 5: Bill of Materials

In the **Report Outputs** section, click on **Add New Report Output** and then in the menu **Bill of Materials** > **[Project].**

If you double click on the Bill of Materials item just added, you should see something like this:

Here you can configure what fields will be included in the file and how you are going to group the components. If you configured the parameters of the parts in your libraries and schematics, you can also output provider part numbers and prices as well by clicking on **Solution 1** in the panel at the left hand-side.

Add the Bill of Materials to the Output Files following the same procedure described in Step 4.
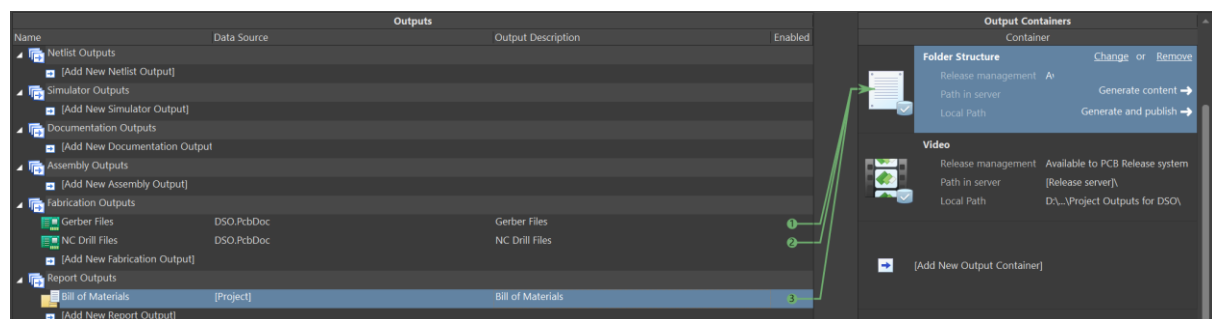
Save your Output Job file.

Step 6: Generating the files

If you are creating multiple releases of your project, this is the only step you need to repeat every time. Go to the **Folder Structure** section in the panel at the right and click on **Generate content**.

Now your files are generated, ready to compress and send: