Altium

QUICK GUIDE

Constraint Manager

Introduction

The Constraint Manager in Altium Designer is a powerful tool that simplifies and streamlines the complex task of PCB design. Central to this is a unified table-based editor that consolidates all your design rules and constraints, whether they are electrical or physical in nature. This brings everything into one place, saving you the time and effort of navigating through multiple menus and settings.

One of the significant advances offered by this feature is the unified access to nets, classes, and rules, both from the schematic level and the PCB editor. Now, with this integrated approach, designers can effortlessly input and modify constraints at both the schematic and layout levels, which enhances workflow and ensures design cohesiveness.

The Constraint Manager also allows you to redefine rules flexibly, meeting the shifting requirements of your design as it evolves. Added to this is the Class-to-Class Clearance Matrix, an intuitive interface that helps you specify clearances between various component classes, thereby aiding in compliance with industry standards.

Overall, the Constraint Manager is a key enhancement in Altium Designer. It not only makes rule and constraint management simpler but also significantly improves your design workflow by integrating rule management across different stages of the design process.

Benefits of the Altium Designer Constraint Manager

Unified Access to Constraints

With the Constraint Manager, you can access and modify design rules and constraints from both the schematic and PCB editor. This reduces the time spent toggling between different design stages, improving your overall workflow.

Physical and Electrical Constraints

The Constraint Manager allows you to set both electrical and physical constraints within the same framework. This enables a more holistic approach to design, ensuring that all aspects, from signal integrity to mechanical fit, are accounted for.

Table-Based Editor

The feature offers a table-based editor that brings all your design rules and constraints into one place. This centralized location aids in quick rule verification and modification, reducing the chances of design errors.

Improved Productivity

The feature's unified and table-based approach allows for easier and quicker constraint management. This leads to improved productivity as designers spend less time managing constraints and more time on actual design work.

Class-to-Class Clearance Matrix

This tool allows you to specify clearances between different classes of components through an intuitive interface. This helps in meeting compliance requirements with ease, thereby improving the reliability of the final product.

Streamlined Workflow

The feature's ability to manage constraints from both schematic and PCB design stages in one place streamlines the design process. This results in a faster time-to-market for the product.

Redefinable Constraints

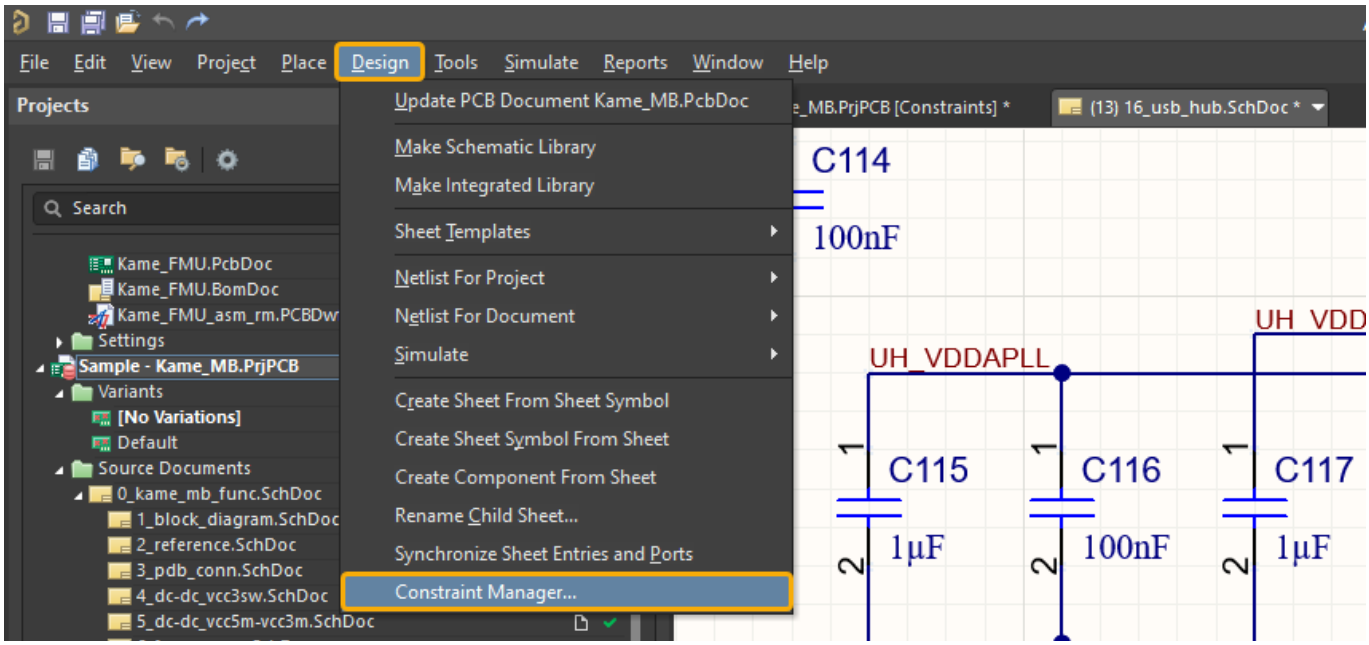
The ability to easily redefine rules gives you the flexibility to adjust to the changing requirements of your design project. This ensures that your design stays aligned with both technical and business goals, without having to rework your constraints from scratch.

Constraint Manager Use Case

Constraint Manager simplifies the PCB design process by letting you manage all your design rules in one place. It connects the schematic and PCB layout, making sure that what you plan is what you get. This makes it easier to communicate design intent and ensures that your board is made right the first time.

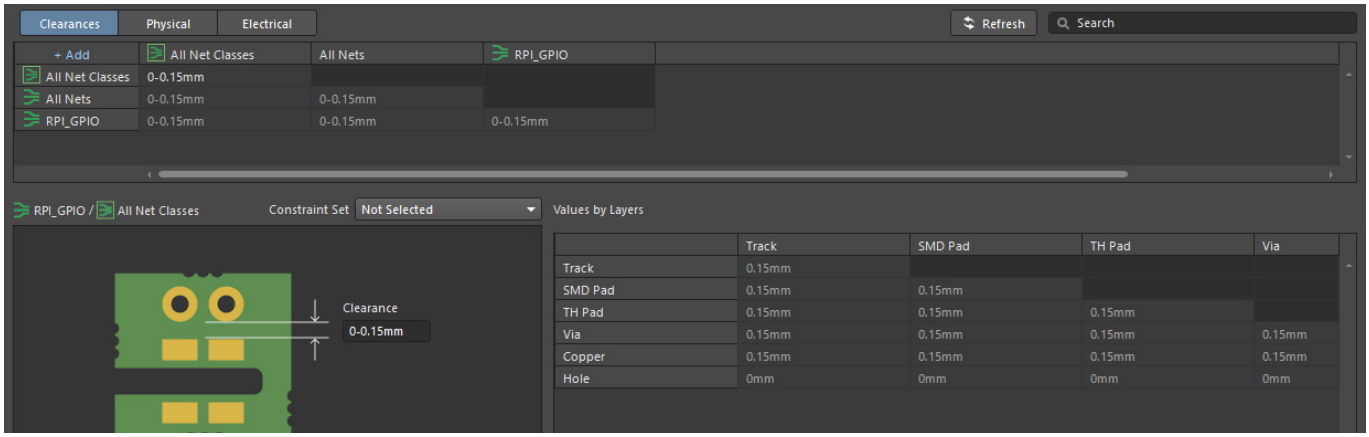
Here is one example that illustrates the capabilities of Constraint Manager in a real-world application:

1. Starting in the schematic editor, go to **Design** and open **Constraint Manager**

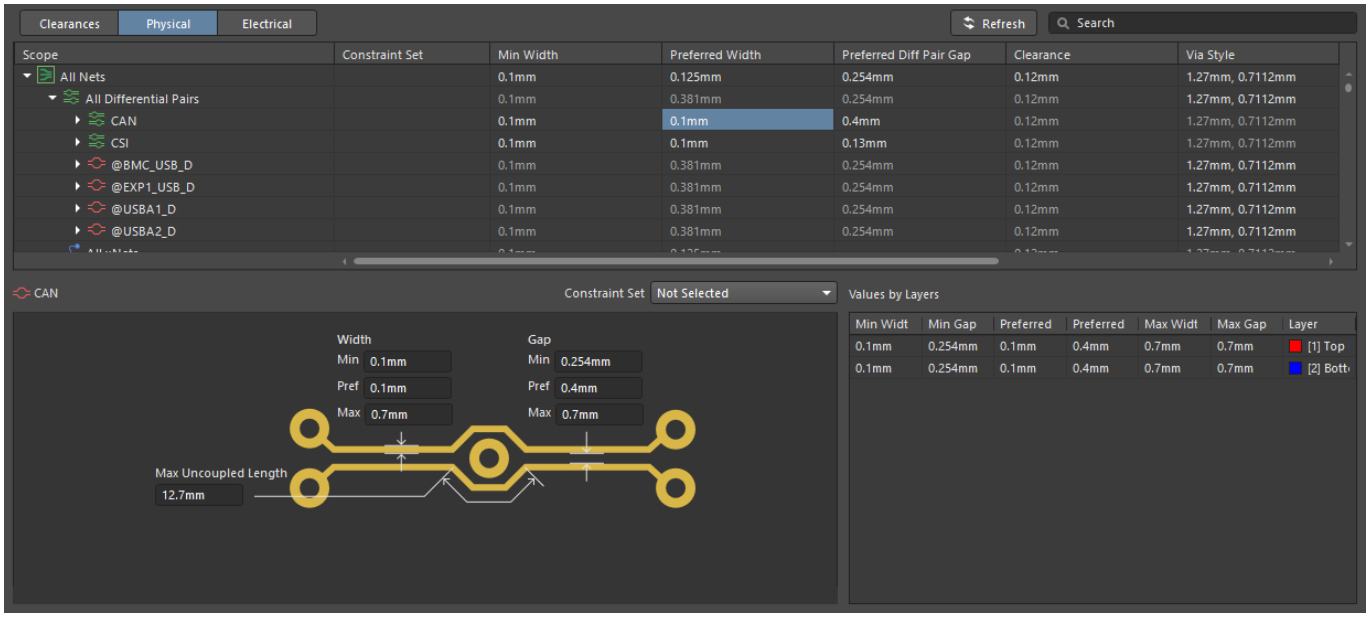


2. After opening the **Constraint Manager document**, we can easily manage rules for all nets in the project. The interface, accessible from the schematic editor, is divided into three tabs:

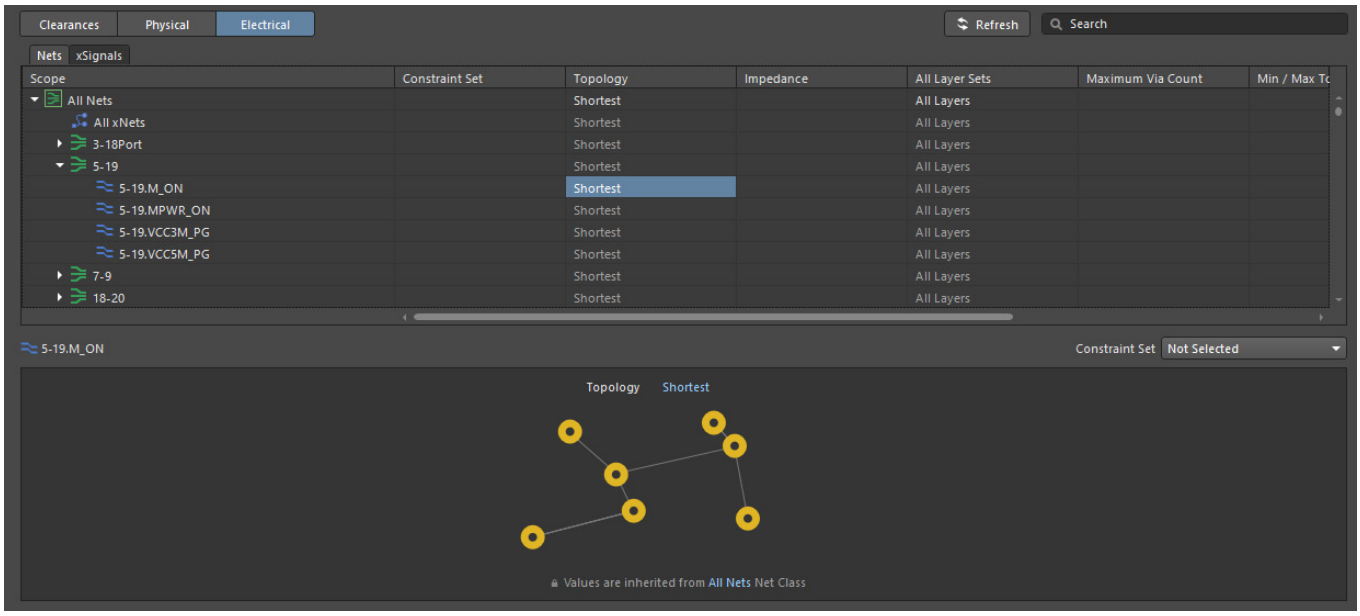
   2.1. Clearances for defining minimum distances between Net Classes,



2.2. **Physical** for setting physical constraints such as Track Width, Differential Pairs details, Clearance, Via Style, and Polygon Connects,

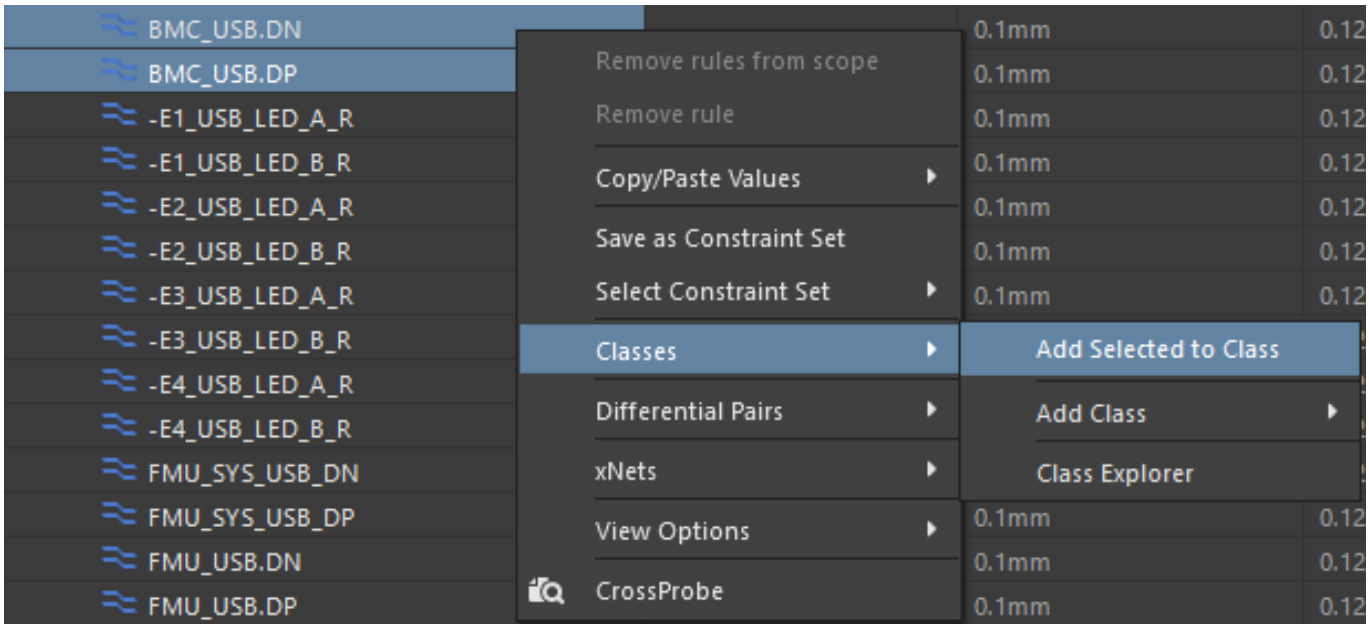


2.3. **Electrical** for specifying electrical constraints like Topology, Impedance, Maximum Via Count, Min/Max Total Length, Max Stub Length, and Max Via Stub Length.

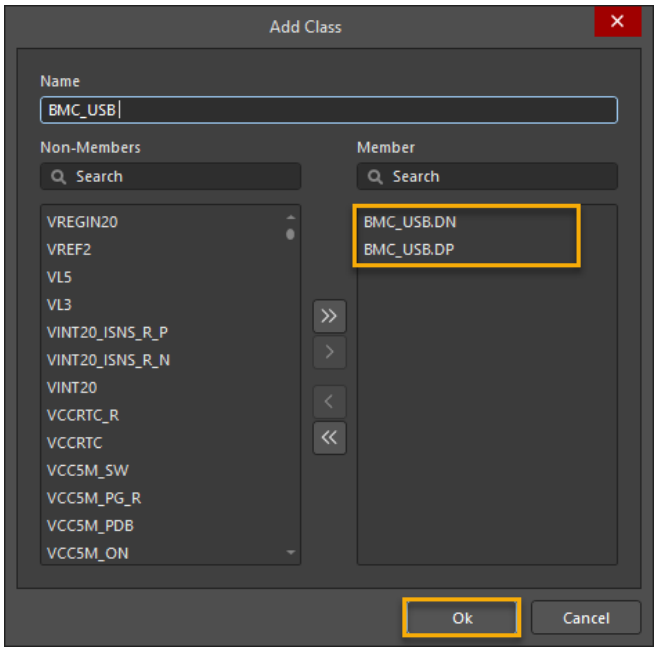


3. All **Nets** can be easily grouped into classes, differential pairs or xNets. To do this, you should select the nets you want to add to group , and then **right-click** then choose the desired type:

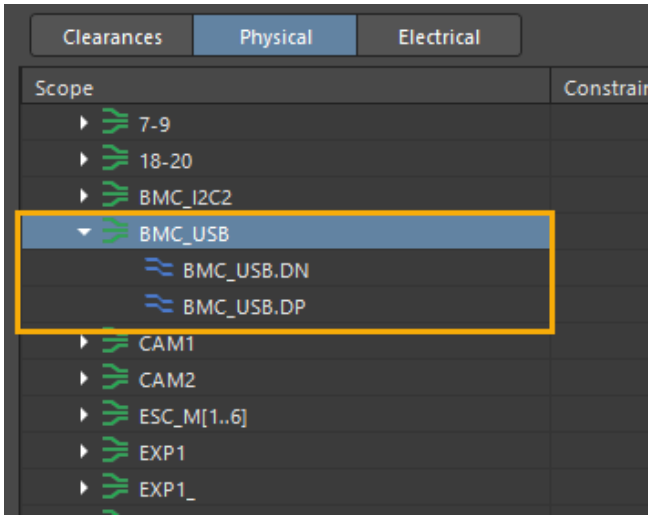
3.1. Classes



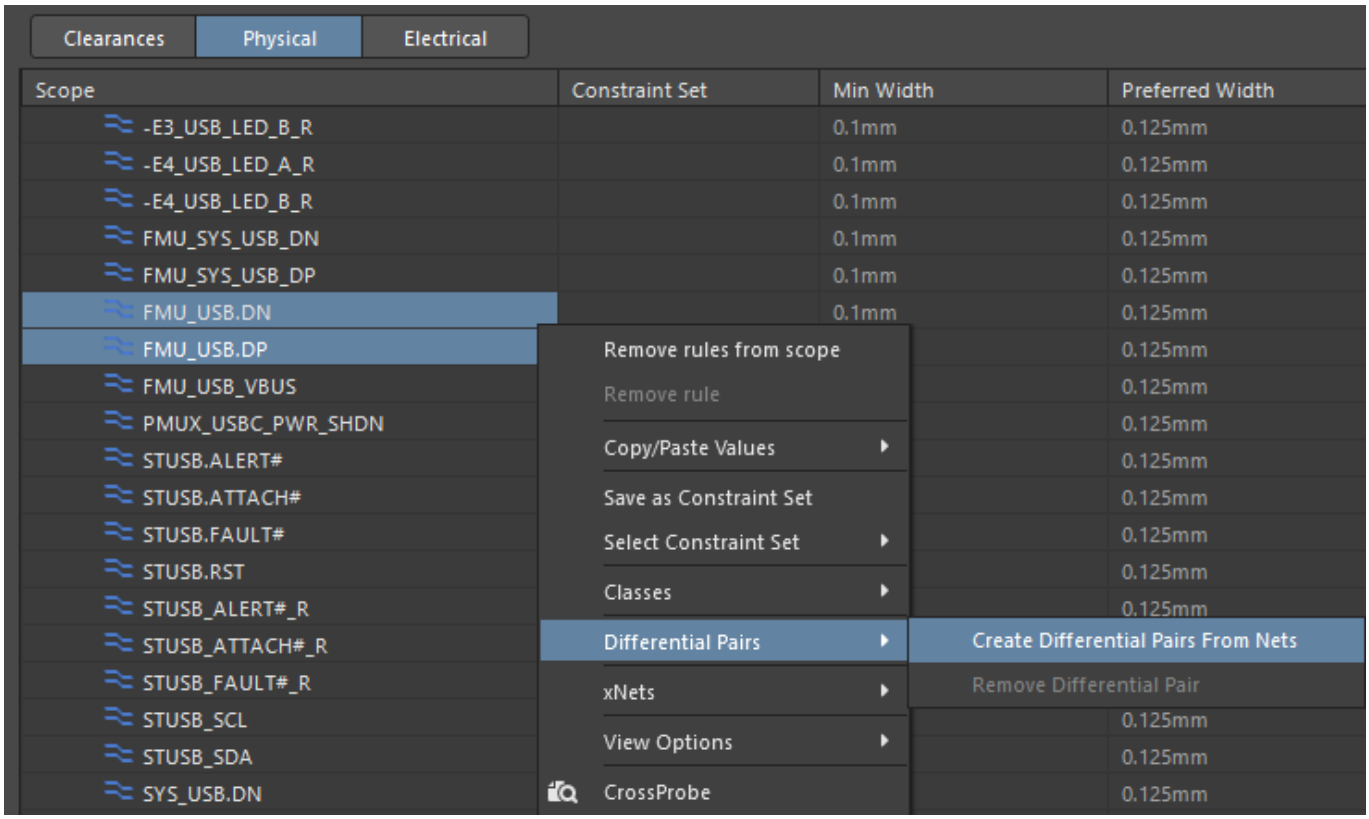
Then, in the appearing window, you can name the new class, add '>>' or remove '<<' signals to a member of the class. After setting all options, click '**OK**'.



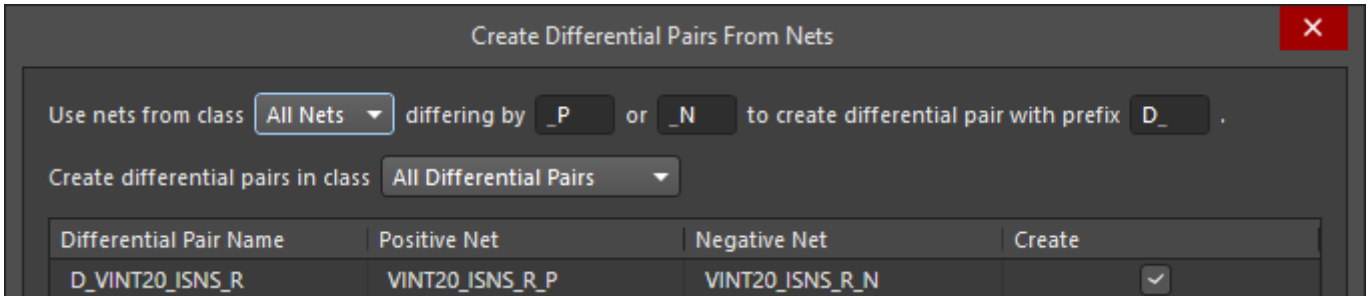
The newly created **Class** will immediately appear in the tabs.



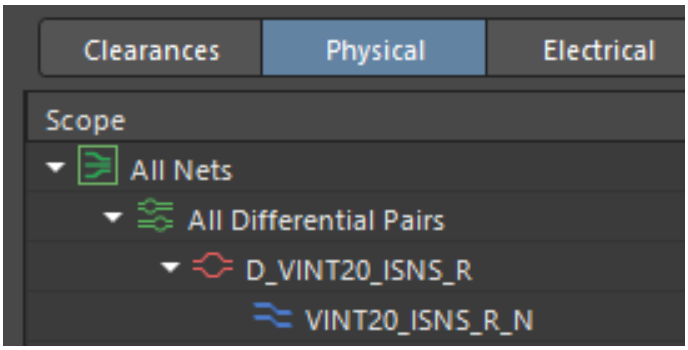
3.2. Differential Pairs



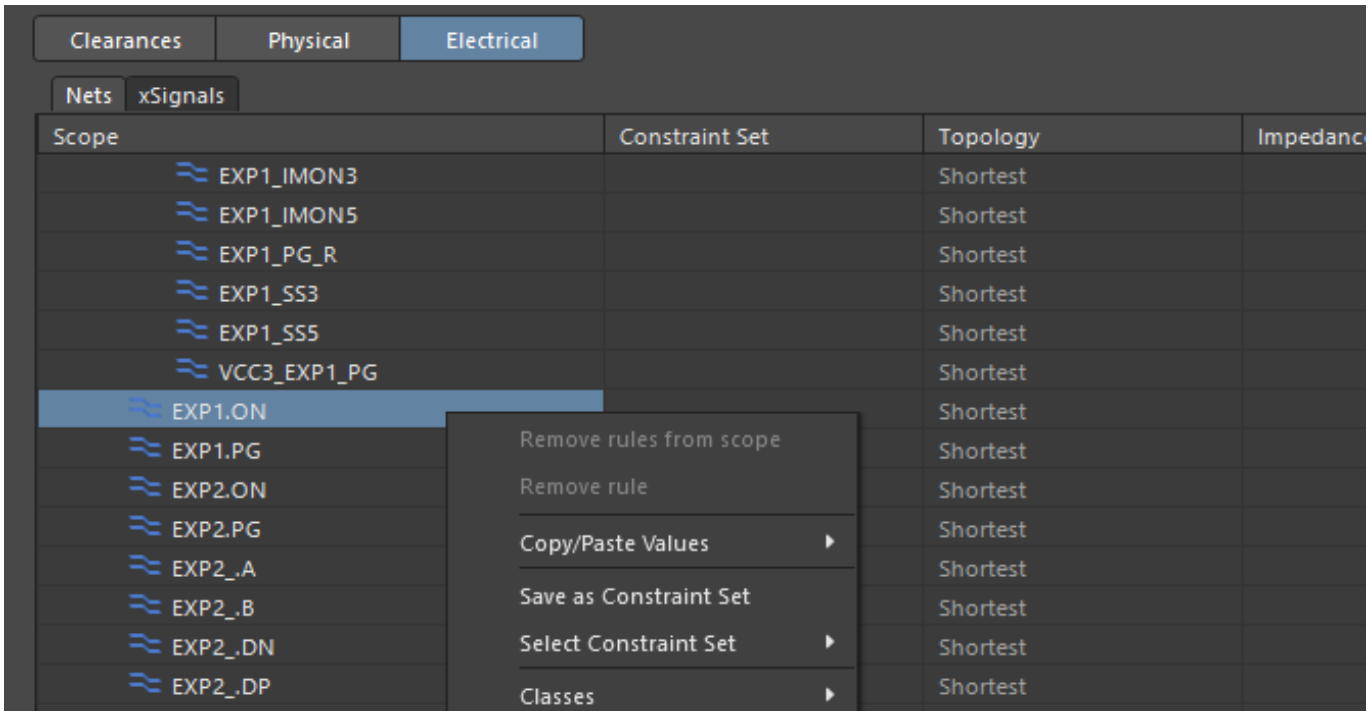
either change or leave the **differentiating suffixes** (_P and N) and the **prefix** (D_) that will be used to create the differential Pair Name. After making your selections, click 'Execute'.



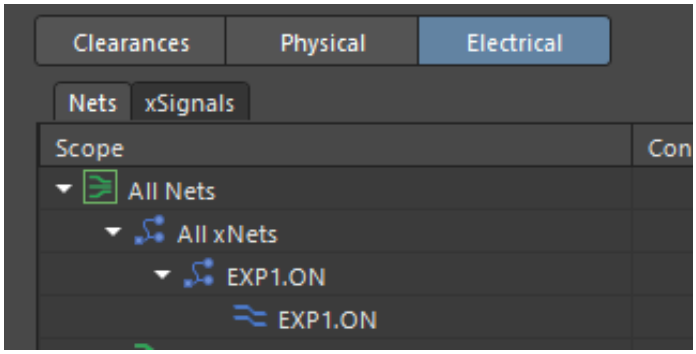
The newly created **Differential Pair** will immediately appear in the tabs.



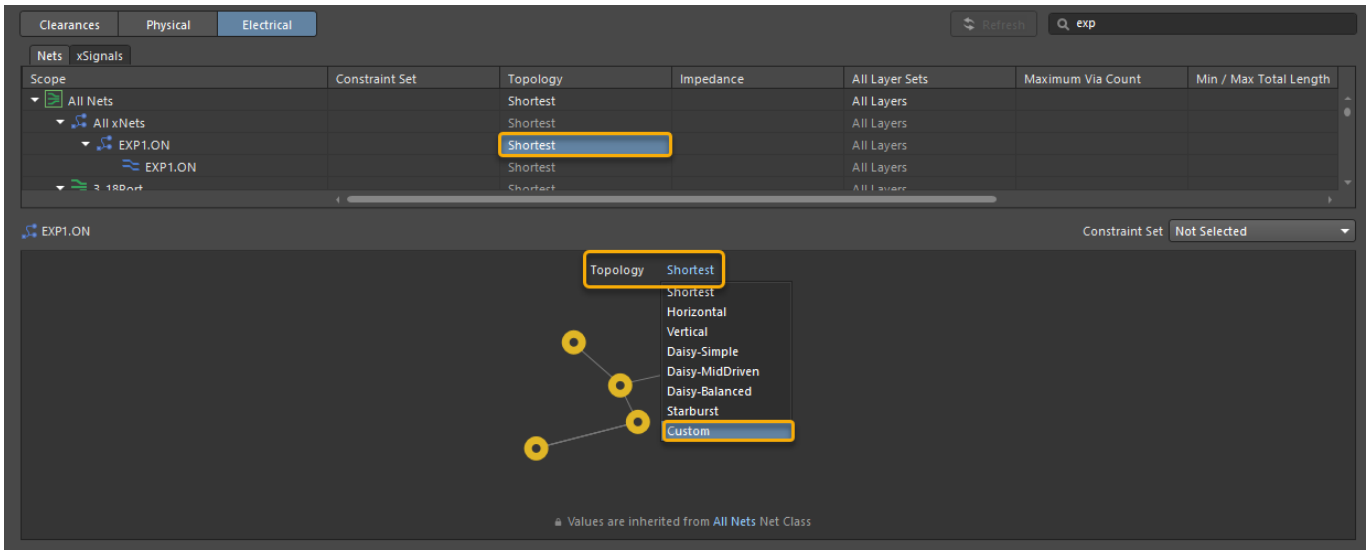
3.3. xNets



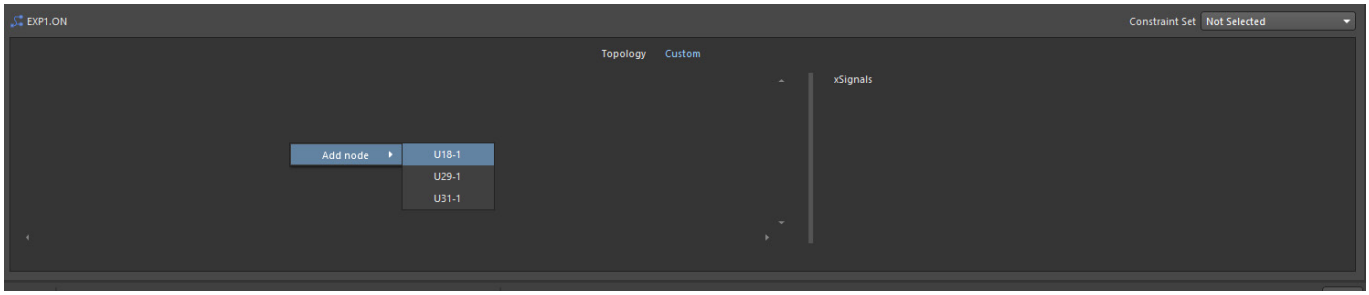
The newly created **xNet** will immediately appear in the tabs.



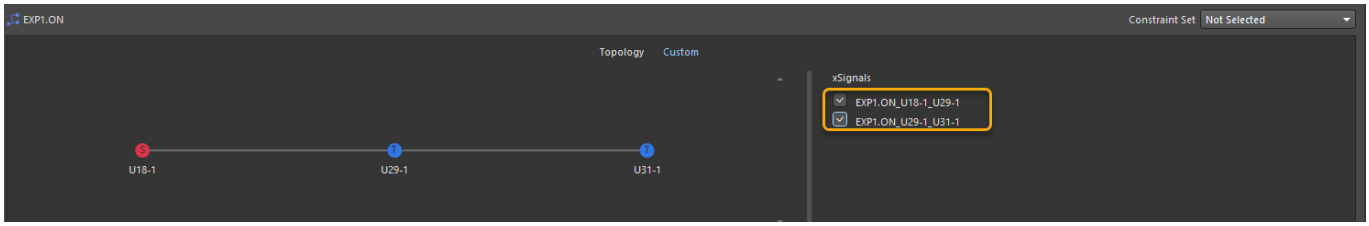
4. In the added **xNets**, you can set individual **xSignal** rules. To do it go to the '**Topology**' column. In the bottom panel, then near the topology menu, select '**Custom**' from the dropdown list



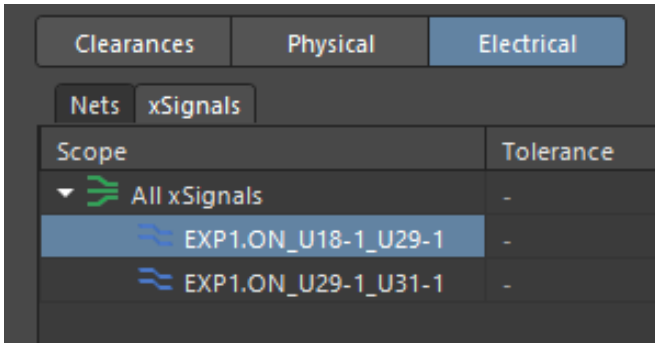
Then, in the topology window, add all the nodes by **right-click** at empty window then **Add node** then add all nodes.



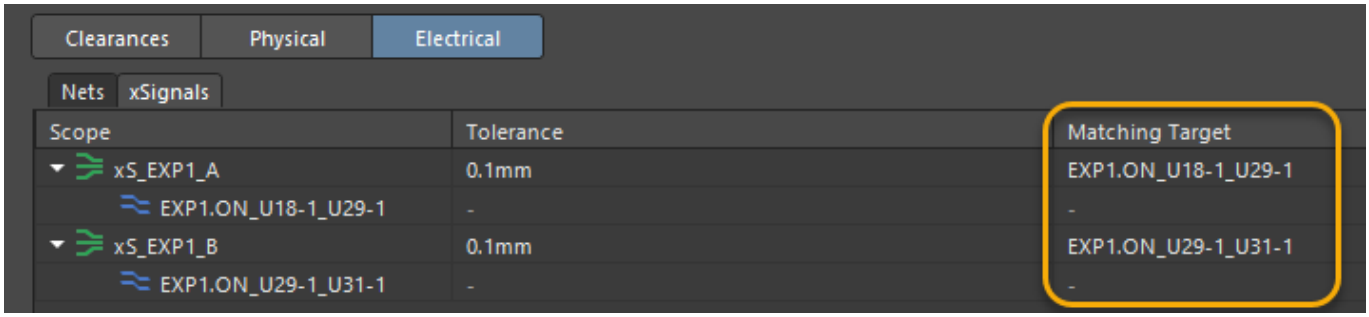
Next, check the checkbox for the xSignal you want to define.



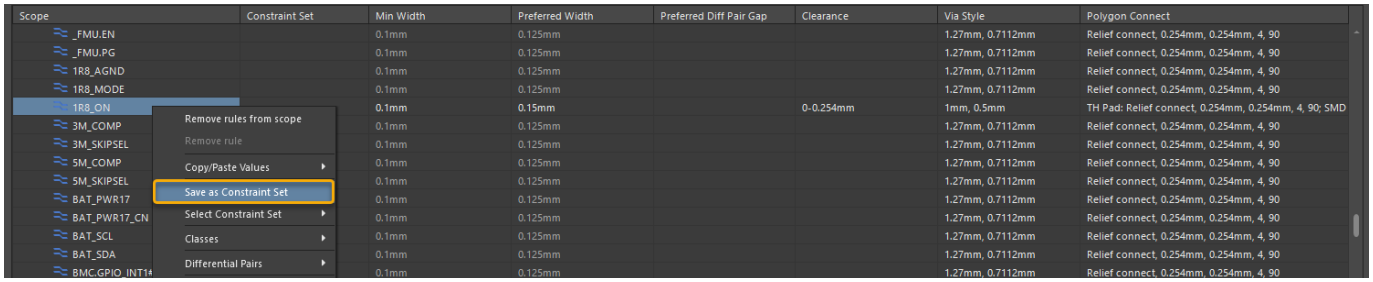
Now, in the xSignal sub-tab, the previously selected pairs will appear.

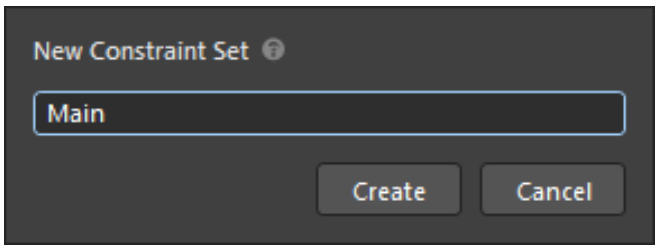


To define length **Tolerances** and **Marching Target**, each xSignal should be added to the class.



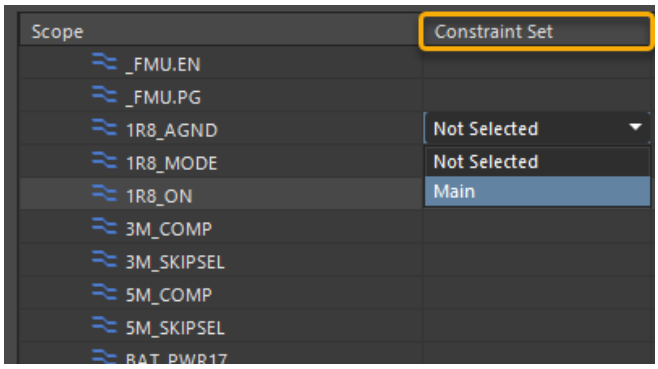
5. The new feature is the ease of defining a **Constraint Set**, which is a defined set of parameters that we can later conveniently assign to nets or classes. To define such a Set, **right-click** on the row of rules you desire, then choose the option **Save as Constraint Set**.



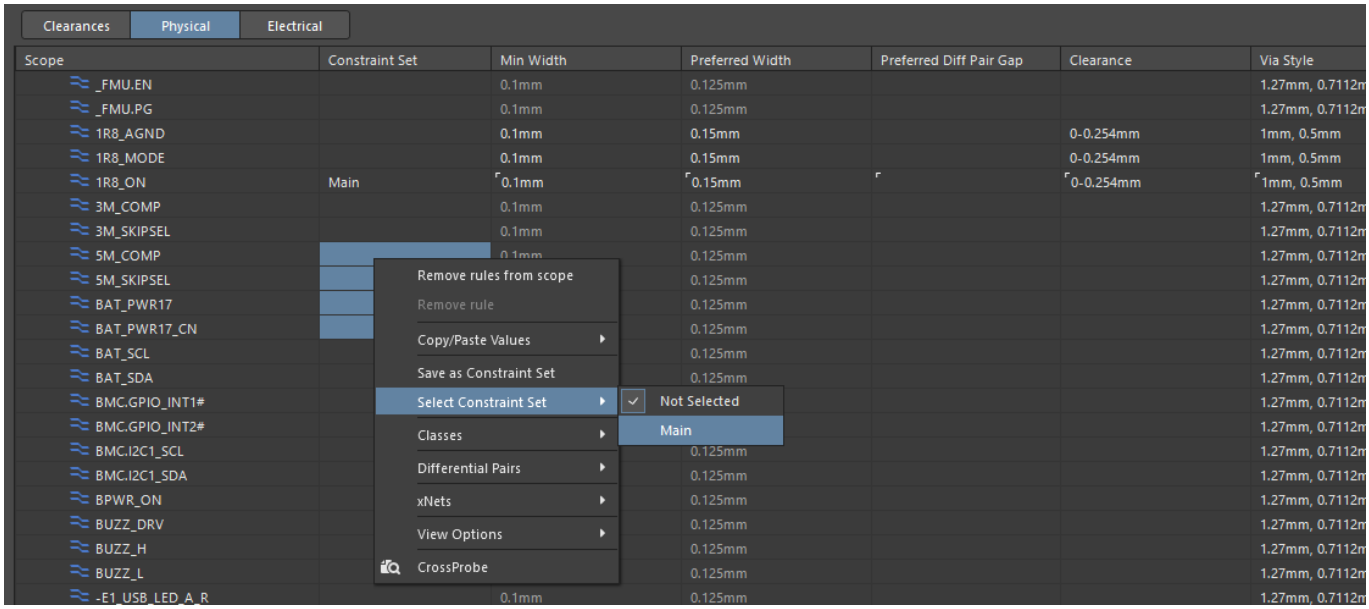


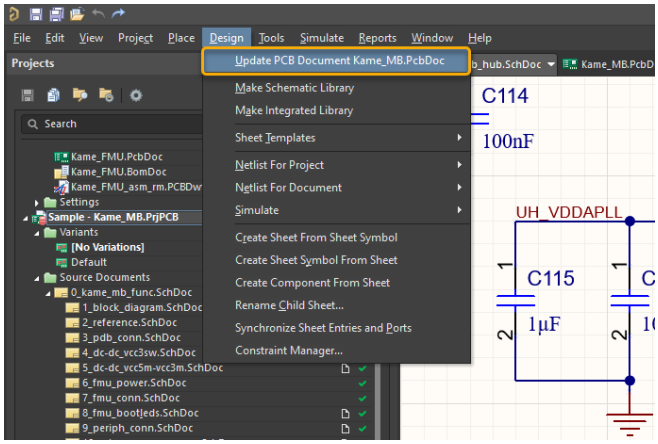
6. In the appearing window, you should enter the desired name of the constraint set.

7. Now, we can assign our constraint set to any class or net by clicking on the cell in the **Constraint Set** column and selecting our set from the **dropdown list**.



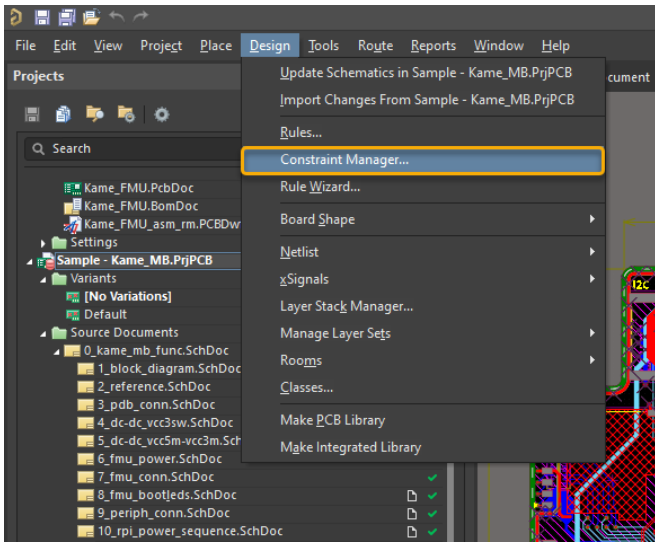
or select more than one and **right-click**>**Select Constraint Set**>'Constraint Set Name'



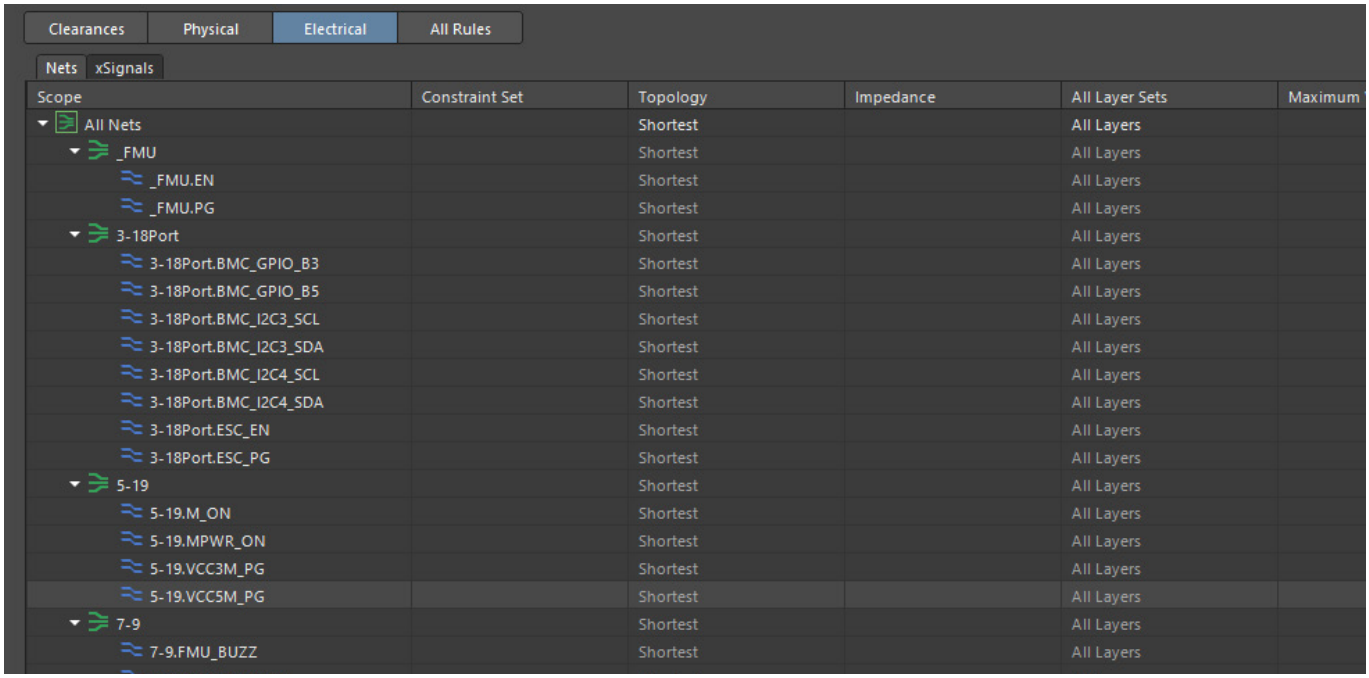


8. After finishing the editing in the Constraint Manager, you need to transmit all the information to the PCB by selecting the command **Design > Update PCB Document**.

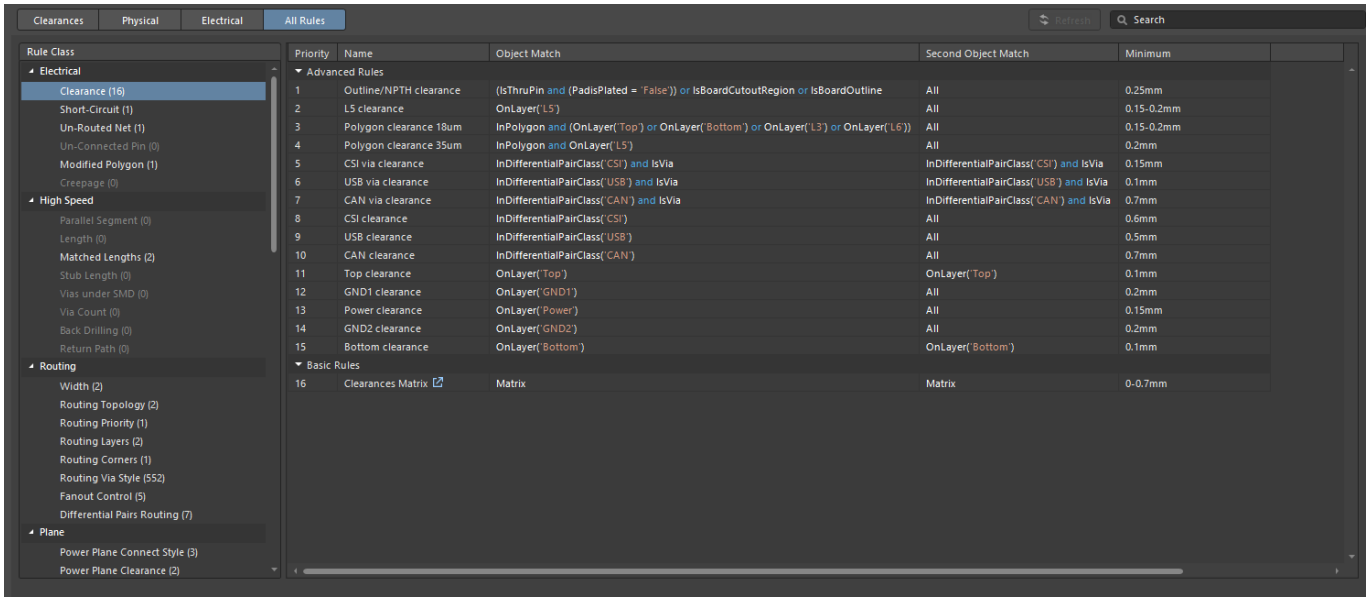
9. In the PCB editor, similarly to the Schematic, you can access the Constraint Manager by selecting **Design > Constraint Manager**.



10. The Constraint Manager, accessible from the PCB level, is also divided into tabs. However, in addition to the **Clearances**, **Physical** and **Electrical** tabs available in the schematic, in the PCB there is also an **All Rules** tab, which is a compilation of all rules and is inspired by the old version of the Rules Manager.



All rules:



Conclusions

Centralized Rule Management: Constraint Manager serves as a one-stop hub for rule definition and management, streamlining the design workflow for increased productivity. This centralized approach eliminates the need to jump between multiple tools or panels, making it easier to maintain a consistent set of rules across the design.

User-Friendly Tabs: With user-friendly tabs like Clearances, Physical, and Electrical, Constraint Manager makes it easy to focus on specific design aspects. This targeted approach allows designers to more efficiently optimize various elements, such as spacing between traces or electrical properties, without getting overwhelmed by the full complexity of the project.

Intuitive Navigation: Intuitive navigation within Constraint Manager promotes quicker learning curves for new designers and greater efficiency for seasoned professionals. Its layout is designed to be easily understood, minimizing the time needed to get acquainted with its functions and capabilities.

Comprehensive 'All Rules' Tab: The All Rules tab offers a comprehensive view of all constraints, making it easier for engineers to identify conflicts or inconsistencies. With all the rules listed in one place, designers can quickly assess the status of their design rules, which saves time and reduces the risk of errors.

Continuous Improvement: Ongoing improvements and feature rollouts for Constraint Manager demonstrate its value as an indispensable tool in electronics design. Its continual evolution ensures that it stays aligned with industry needs, making it a reliable tool for both current projects and future advancements in electronics.