Altium®

製品マニュアル
Modified by Jason Howie on Nov 20, 2017

Underlying Altium Designer's Schematic and PCB editors is a powerful query engine. By entering queries into this engine you can logically scope precisely those objects you require. A query is a string you enter using specific keywords and syntax, which will return the targeted objects.

Queries are primarily defined in a *Filter* panel, but are also used to define scoping for PCB design rules. As you build your knowledge of the Query Language, and the functions, keywords and syntax available, you will be able to type expressions directly. However, until that level of confidence is built, the *Query Helper* can be a beneficial crutch on which to lean! Providing an editing window in which to 'build' an expression, the *Query Helper* contains a full listing of query functions/keywords and related controls, as well as a handy syntax checker.

The vastness of the Query Language may seem a little daunting to begin with, but over time you will learn to appreciate its power – building a set of favorite query expressions with which to target common sets of objects (and committing them to muscle memory). And to quickly get up to speed, information is available for each of the query functions. Highlight (or click inside) any given keyword – in the *Query Helper*, a *Filter* panel, or the **Query** field of a PCB design rule – and press **F1** to access its page within the documentation.

> ✅ For a look at how queries are used in the scoping of design rules, see Scoping Design Rules.

The Query Language reference is composed of the following:

- Schematic Query Functions
- PCB Query Functions
- Library Query Functions

- [System Query Functions](#)
- [Query Operators](#)