

"Part Choices" Q & A

This webinar not only presented the Part Choice feature in Altium Designer, but it also explained the purpose of parameters. Most of us grew up with software that required us to seek information and fill in blanks manually. This was time-consuming and error-prone. For efficiency, it was best to keep the number of parameters to a minimum. Tools like Altium Designer now provide parametric information with a few mouse clicks, allowing us to either copy and paste parametric information, or in certain circumstances, permit Altium Designer to populate the component. With this ability, it is easy to populate a component with dozens of parameters. The population of information has two easily overlooked purposes. In this webinar, we revisit these two parametric information purposes: intelligent part selection and a clean bill of materials.

Part Choices provides links to the datasheet, real-time supplier quantities and prices, and a web link to component information. Depending on the library structure, a Part Choice can be added to supplement existing parametric information. It can also provide alternative components if the primary component is not available or has an extended lead-time.

The library structure will determine if Part Choices can be used.

Symbol-Centric Libraries – (These are libraries where the component is defined within the symbol. Integrated libraries are considered symbol-centric.) Part Choices can be added to the component. Parametric data from the Part Choice can be automatically copied into the component as parameters. However, if the Part Choice contains a symbol and footprint model, it must be downloaded and copied into the library. Nine Dot Connects recommends that consistent manufacturer name, manufacturer part number, and description formats be used even if the library components are associated with Part Choices. This allows the traditional bill of material report generator and the ActiveBOM editor, which heavily leans on the Part Choices.

Database Libraries - Part Choices can be added to the symbol; however, this breaks the paradigm that the database represents the component and its parametric information. For discrete components like resistors and capacitors, Part Choices will not work within the database structure. In a database, the symbol is simply a graphical representation assigned to two or more components. For example, the database may contain 1000 resistor components; however, all 1000 resistors refer to the exact symbolic, graphical representation.

Presently, Altium does not have a method to import Part Choice information from an external database to the schematic when placing a component. No method will likely be made available given that Altium provides database-like solutions like Concord Pro and Altium 365. However, a Part Choice can be added to a component once placed in the schematic. It does not help with part selection, but it can help create the bill of materials if the ActiveBOM editor is used to create the bill of materials.

Concord Pro or Altium 365 - Part Choices can be added to the component. Parametric data from the Part Choice can be automatically copied into the component as parameters. Components (including the symbol and footprint models) can be directly acquired from the Part Choice.

Comment/Question	Answer/Response
<p>Don't you need to know the package?</p>	<p>Note: Not sure of the context, though this may be a question referring to one of the webinar polls asking about the minimum number of library parameters for a bill of materials or the component characteristic parameters.</p> <p>For the bill of materials, the package would be good to know as a cross-check if the purchaser is savvy enough to perform such diligence. A good description should also contain the package type. However, the bill of material does not require a reference to a package.</p> <p>For the component characteristic parameters, the information can be from 3 possible areas:</p> <ul style="list-style-type: none"> • The footprint associated with the symbol • The description • The datasheet <p>Therefore, adding another parameter for the package tends to be redundant unless sorting on the package is a necessity.</p>
<p>How do you communicate through parameters to someone in purchasing? Which parameters are essential? For instance, a 10K resistor must be 0.1% and 25ppm tempco. I know they are in the datasheet, but how would they know that they are design-critical?</p>	<p>Let's set some expectations. The purchaser expects a manufacturer's name and part number. The purchaser also expects that the designer who submits the part has vetted it for the design. If they seek alternative parts, the designer should verify those (or submitted by the designer.)</p> <p>A good description will have all the critical characteristics of the component. We recommend that the description be added to the Bill of Materials for clarity and reference. However, it is not required to make a purchase.</p>
<p>I heard "value" is used for display. Isn't it used for MixedSim?</p>	<p>Yes, it is used for display purposes. MixedSim (a.k.a. SPICE) makes use of this parameter.</p>

<p>I might have missed it, but where do the symbol and footprint come from in Part Choices ' Edit Supplier Links'?</p>	<p>If the Part Choices is used in the schematic editor's property panel, it will merely associate the Part Choice selection to the symbol.</p> <p>If the Manufacture Supply Search panel is used, the user can associate both the symbol and footprint if they are using Concord Pro or A365 through the acquire function. If using a symbol-centric library, the models must be downloaded and then brought into the libraries by copying and pasting between libraries.</p>
<p>Is there a way to get manufacturer lead times?</p>	<p>Not through the part choices. However, there is a feature in A365 which is supposed to provide this capability through the ActiveBom. This is an editor that forces the designer to create a Bill of Materials and audit the bill of materials.</p>
<p>[In A365 pro and Concord Pro] creating Component Template can define standardized company parameter names</p>	<p>This is true; however, when a part choice is added and parametric data is already present, Altium provides some options which do not preserve the existing parameters.</p>
<p>You mentioned you need a license and subscription. For those who buy a perpetual license, can subscription run out and still get real-time parts status?</p>	<p>We know that if a license is not present, the part choices will not display. As for the subscription expiring, we're not sure because we would not be able to test this.</p>
<p>One of Part Choices' biggest benefits is alternatives, and you did not address that major advantage. This is particularly useful for discrete components</p>	<p>Agreed. Part Choices is beneficial for listing alternative components. Granted, this will only be effective if the designer uses the Active BoM to create the Bill of Materials. The traditional bill of materials relies on the parameters. Granted, when a part choice is added to a component, the supplier name and part number are added as parameters. However, it is supplier data, not manufacturer data.</p>
<p>I just watched a workshop from Altium on Monday about component creation. I noticed that he was not placing Mgr or Mgr part numbers as parameters. He said he does not do that.</p>	<p>What Altium said is true IF you use Active BoM and part choices. If you use the traditional bill of material report generator in the reports menu, you will need the manufacturer and manufacturer's part number as parameters.</p>