Using Altium Designer for Cabling

Though Altium Designer is a PCB layout tool, the ease of using the schematic editor and the inherent symbol-centric library methodology allows Altium Designer to be used "off label" for cable design. It is noted that for substantial cabling efforts, the user should consider Solidworks Electrical or Zuken E3, which are designed for cable layout and can provide more cable drawing functionality. However, Altium Designer provides a considerable advantage over AutoCAD or Visio due to the user's ability to add information to the graphics. This allows organize and reuse of the cable component, and an automated way of creating bills of material.

There are limitations when using a tool in a fashion for which it was not designed. For example, a PCB bill of materials deals with specific quantities, whereas cable wires and heat shrink are described as "AR" or "AS Required." Off label support by Altium Inc cannot be expected.

To take advantage of Altium Designer as a drawing tool, the class will delve into features and methods not commonly implemented or are repurposed to get the desired result.

What will be covered:

Physical Vs. Symbolic – Cable components on a drawing can be represented as physical, symbolic, or both. Without this core understanding, creating cables in Altium Designer will not work. This class will explore the physical and symbolic aspects of the common cable components to ensure that they can be properly represented in the drawing.

Static Library – Altium Designer uses a static library methodology. When placing a component onto the schematic, it is a copy of the component in the library. This class will explore the relationship between the library and the schematic and what post-placement impact there is on the component.

Bezier Curve – The primitive known as the *bezier curve* will be used frequently when creating components and when drawing wires. This primitive allows the designer to customize the curves rather than trying to piece together separate arc primitives. This class will explain how to use the Bezier curve.

DXF/DWG – The most efficient way to obtain a physical representation of a cable component is to obtain the mechanical model, such as a STEP file and converting it into a 2D view in DXF/DWG format. This class will demonstrate the import procedure.

Components Driven BoM– The only way to create a bill of materials in AD is to have the component represented as a component primitive. This class will demonstrate the library – schematic – BOM relationship.

Multi-Module Components – Cable components are represented in several ways; therefore, the multi-module symbol will be used frequently. This class will show how to efficiently use the multi-module feature within the schematic library for each common cable component type.

Scaling Considerations – Altium does not allow components to be rescaled when placed in the schematic editor. Therefore, the scaling needs to be given due consideration in the library, which will be explored in the class.

Parameter–This class will demonstrate how one can take advantage of the text primitive's properties and the component parameter to achieve the desired visual look without locking the text.