Creating/Inserting a multi-pin connector

This document describes one method to insert a multi-pin connector in SolidWorks Electrical Schematics.

The first step is to create the connector, this can be done in the component tree or by placing a connector pin symbol on a scheme page in your project.

This set of images shows how to insert a component through the Component tree:

Once the component has been created or the first symbol has been placed into the scheme you can right click on the component and select ‘Insert Symbol’, select the appropriate symbol and place it into your scheme. You will do this for the number of pins in your connector (in this example 10 times).
Once your symbols are placed you may want to turn off the TAG attribute of the symbols (in this example I turn off the TAG attribute for symbols representing pins two through ten.

NOTE: When inserting the symbols, if you press the ‘thumbtack’ you can insert all the symbols in order without having to use the insert symbol command for each pin.
*If you want show a partial connector (for example pins 1, 2, 4, 6) you can insert four symbols then modify which symbols represent specific pins in the Manufacture Parts tab of the symbol properties. Look in the GoEngineer Knowledge base (http://kb.goengineer.com) for the article ‘Inserting a partial connector’.