



Introduction

Cadence OrCAD Capture CIS requires a configuration (.DBC) file to make use of your part database. The configuration file:

- Identifies the ODBC data source to use as the part database and specifies the tables to use within that database.
- Identifies the part properties that are transferred to your design when you place or link a database part.
- Sets the visibility for each of the transferred properties.
- Contains the part type associations.

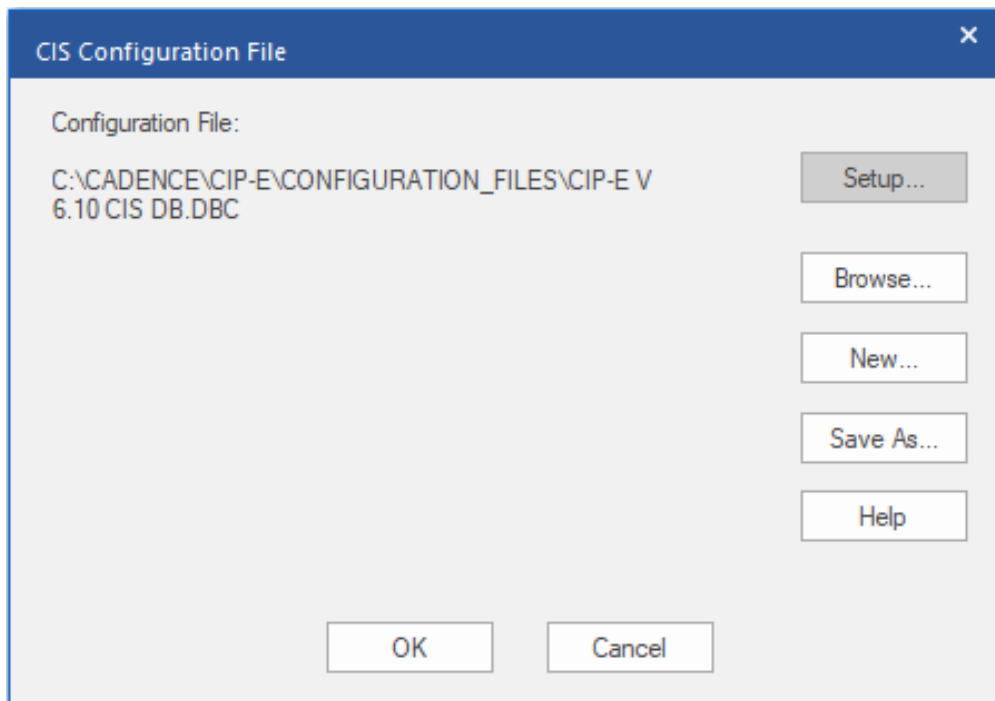
Tip

Keep the configuration (.DBC) file in a read-only directory that is accessible to all OrCAD Capture CIS users. You should make the directory read-only to prevent users from inadvertently changing the configuration.

How to create a DBC file

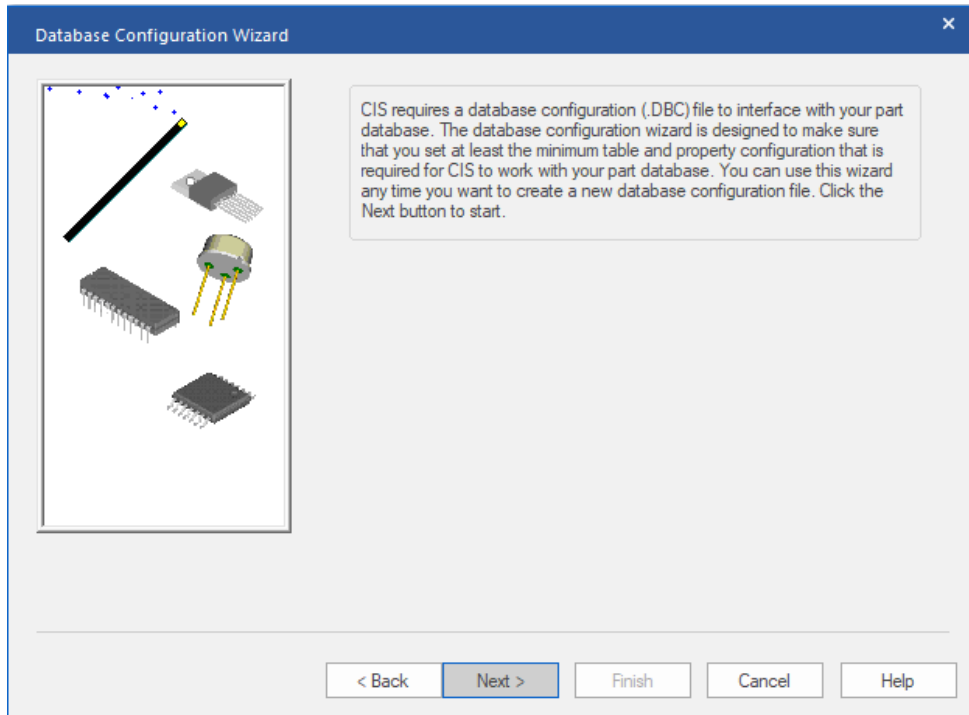
In OrCAD Capture CIS go to Options > CIS Configuration. The following menu appears,

Press 'New'

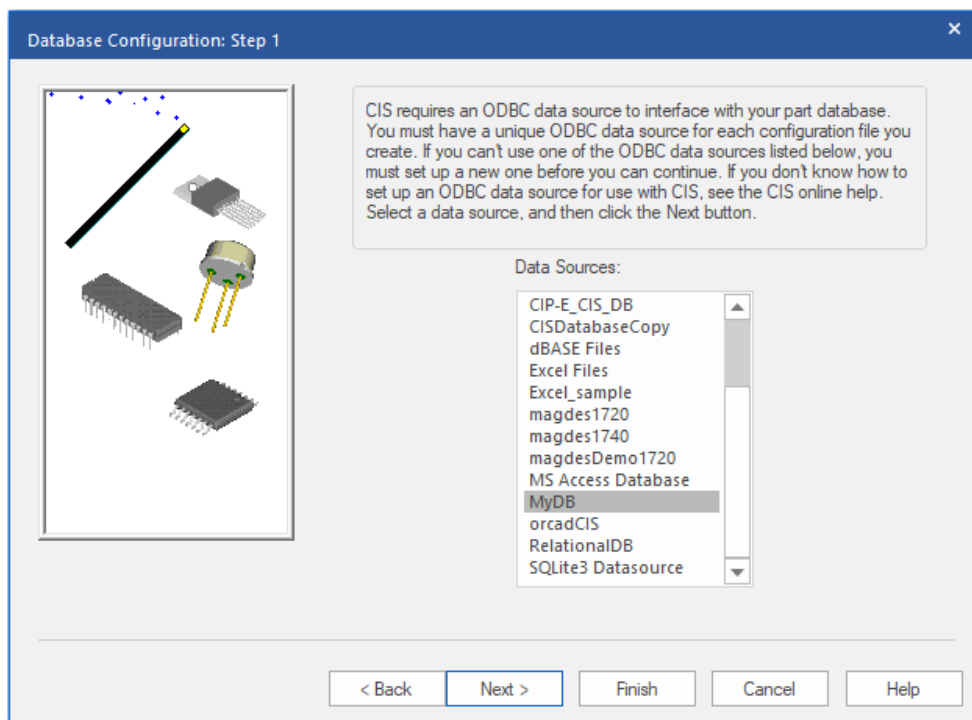


You will get the following screenshot – Click 'Next'

How to create a DBC file

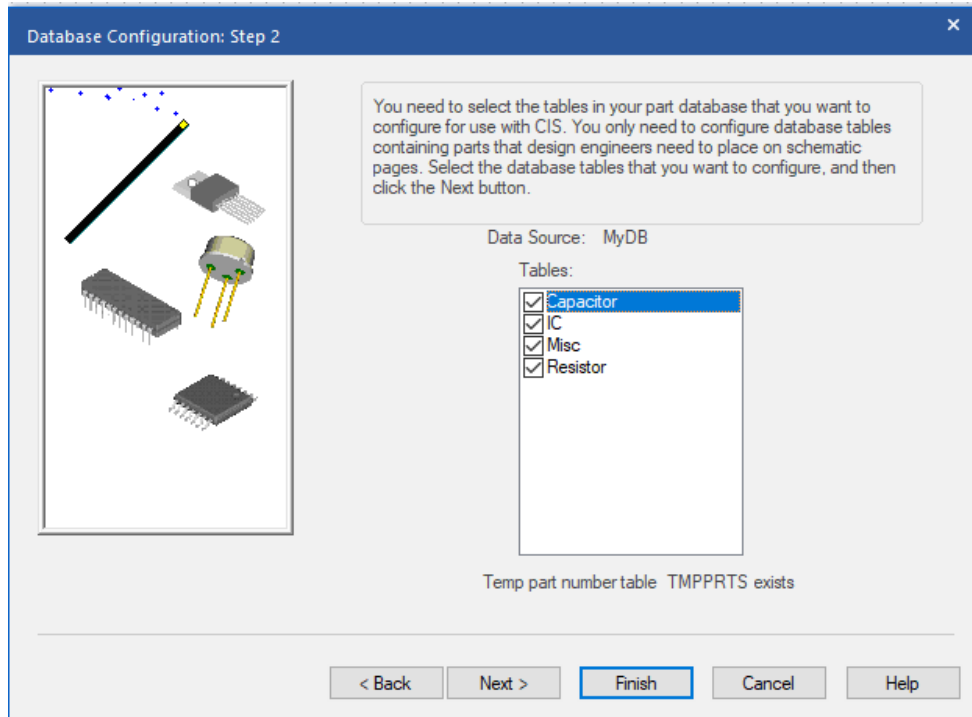


Select your ODBC Compliant data source and click 'Next' (If you haven't defined your ODBC link then stop this process and refer to CTN201 How to Setup an ODBC Data Source for Cadence OrCAD Capture CIS or CTN202 How to set up a system Excel ODBC Database).

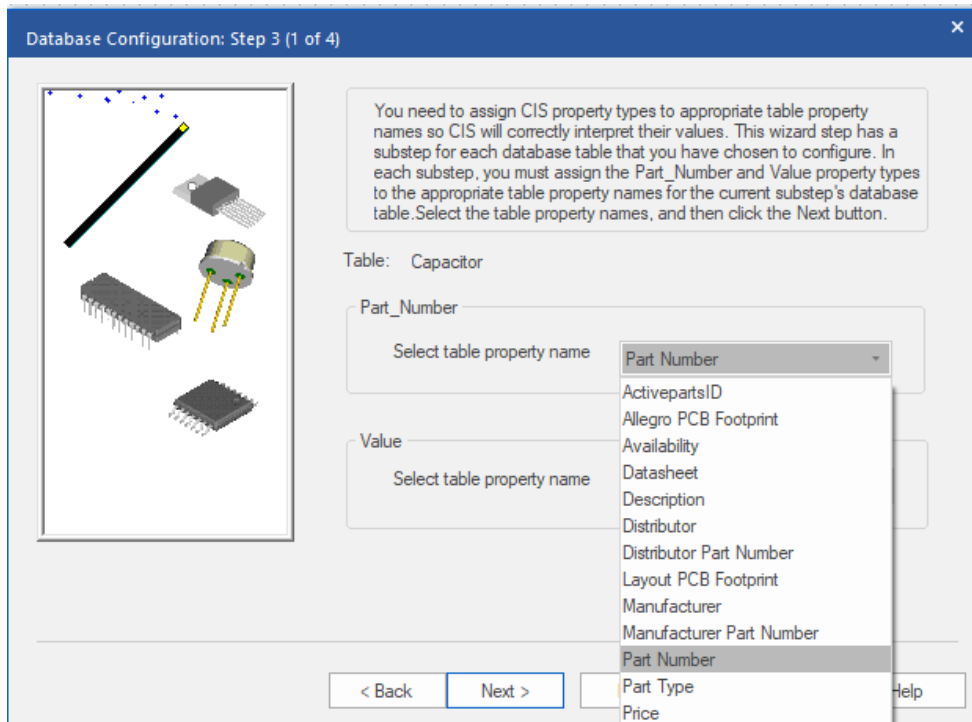


Select the tables that you want to connect to and click 'Next'

How to create a DBC file



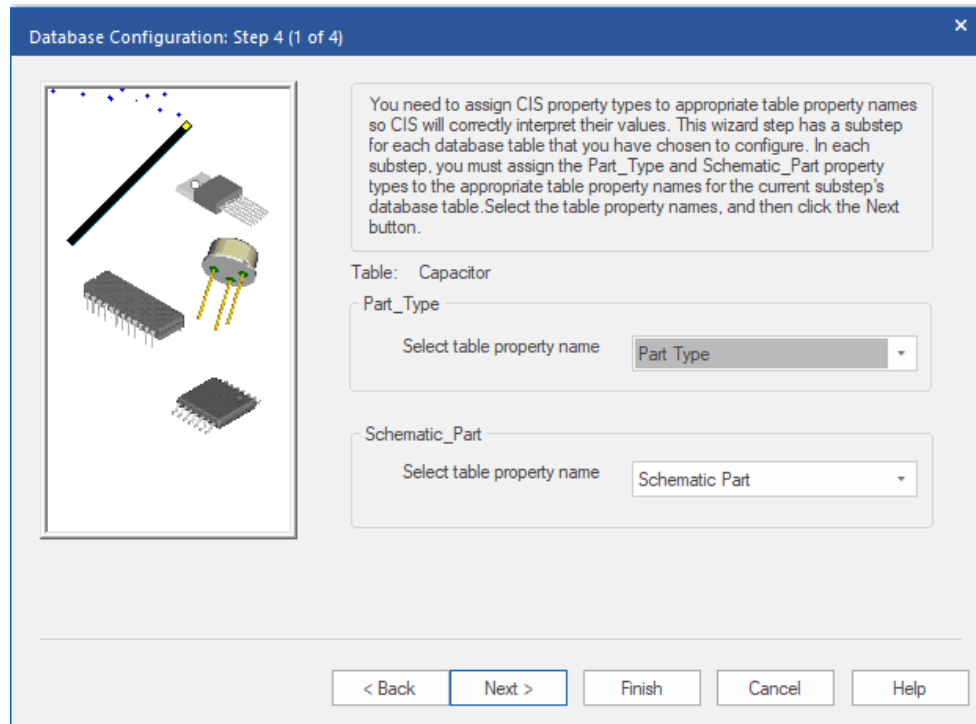
For each table you will need to define the Part_Number and Value to match a table heading in your Database. Use the drop down arrow to select your property heading that match these properties. Mine are Part Number and Value.



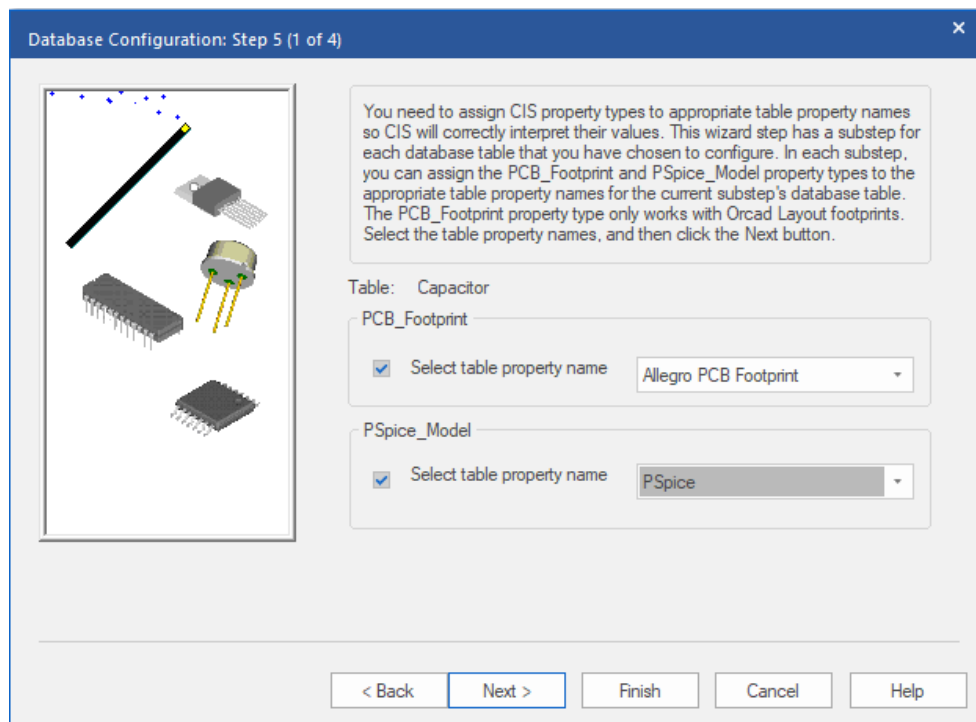
Repeat for all tables in your database (Click Next to work your way through them).

Repeat the above process for Part_Type and Schematic_Part (Again for all tables).

How to create a DBC file

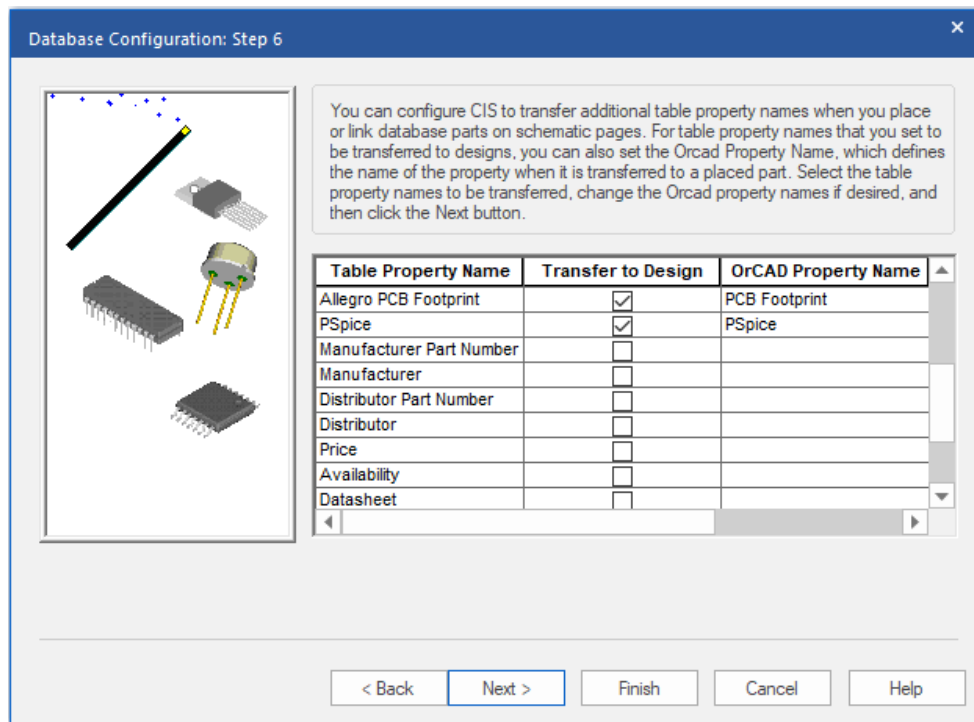


Click 'Next' and then if you are defining PCB Footprints and or PSpice Parts, check the relevant boxes and assign the appropriate table headers. Repeat for all tables.

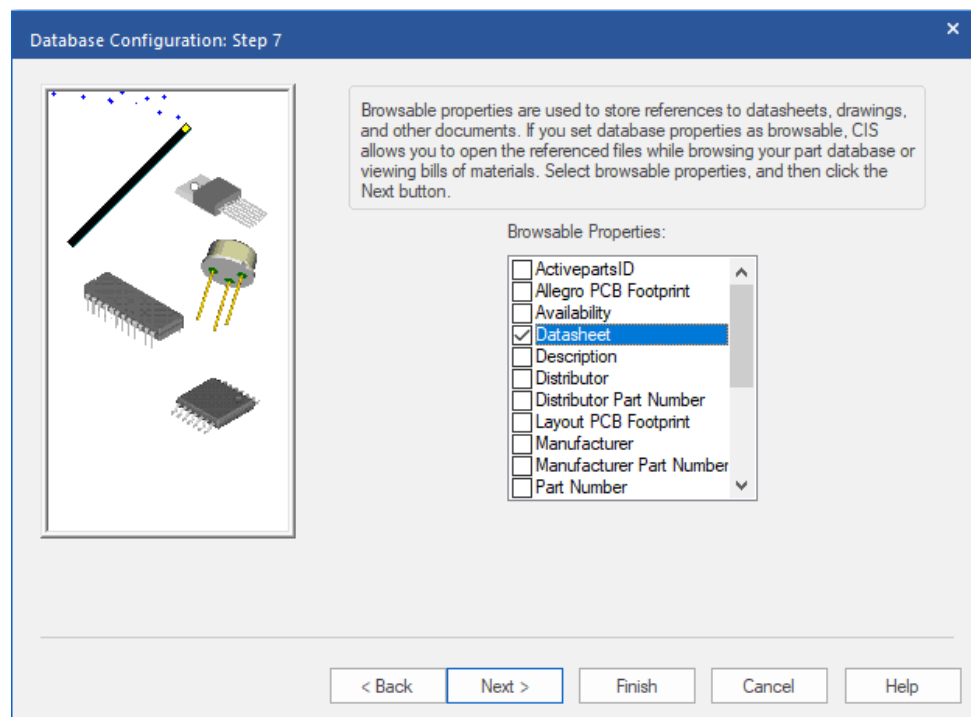


Next, we want to define what properties are transferred to the design. The default ones are selected. (This is only the properties that will be visible in the schematic. All properties are available at the CIS BOM reports stage. I have added Description as an additional property.

How to create a DBC file

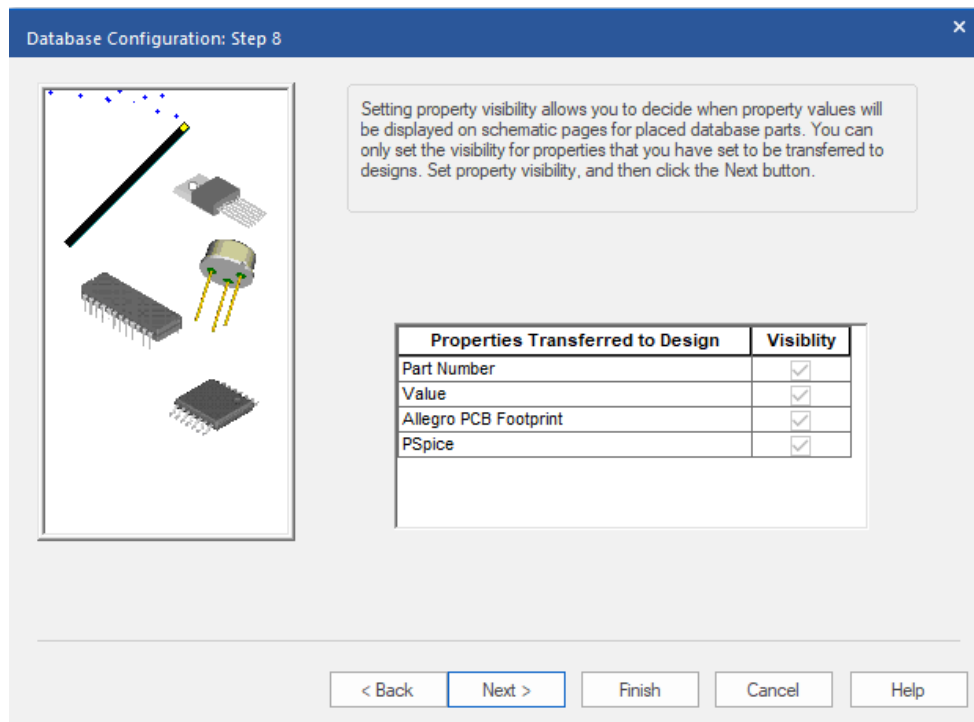


Define the browsable properties. Properties that may have an external link i.e. PDF datasheet or URL.



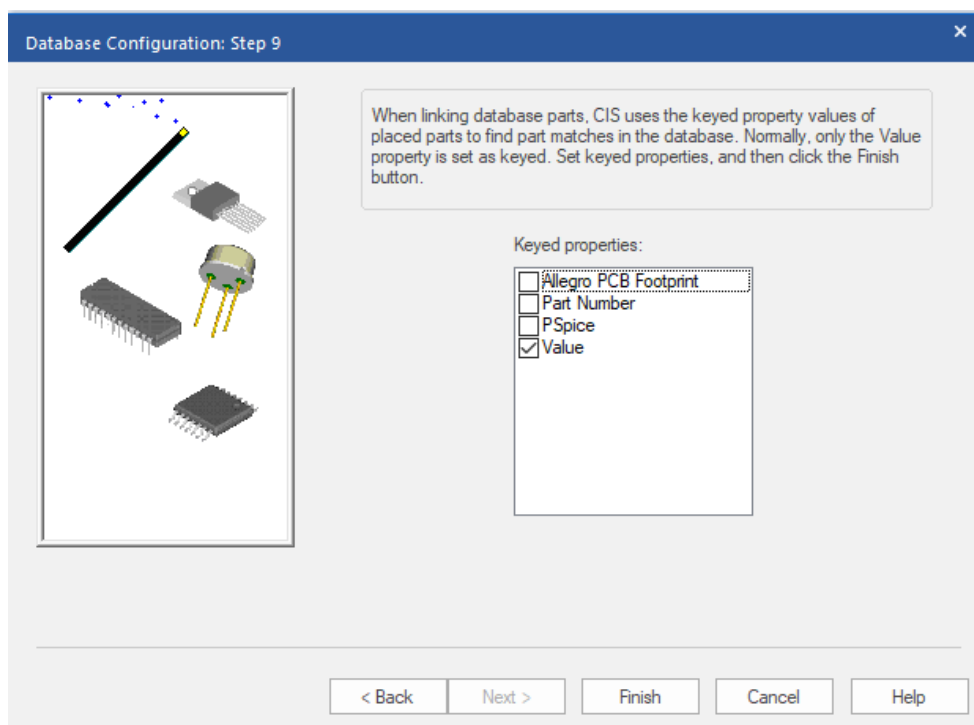
Click 'Next'

How to create a DBC file



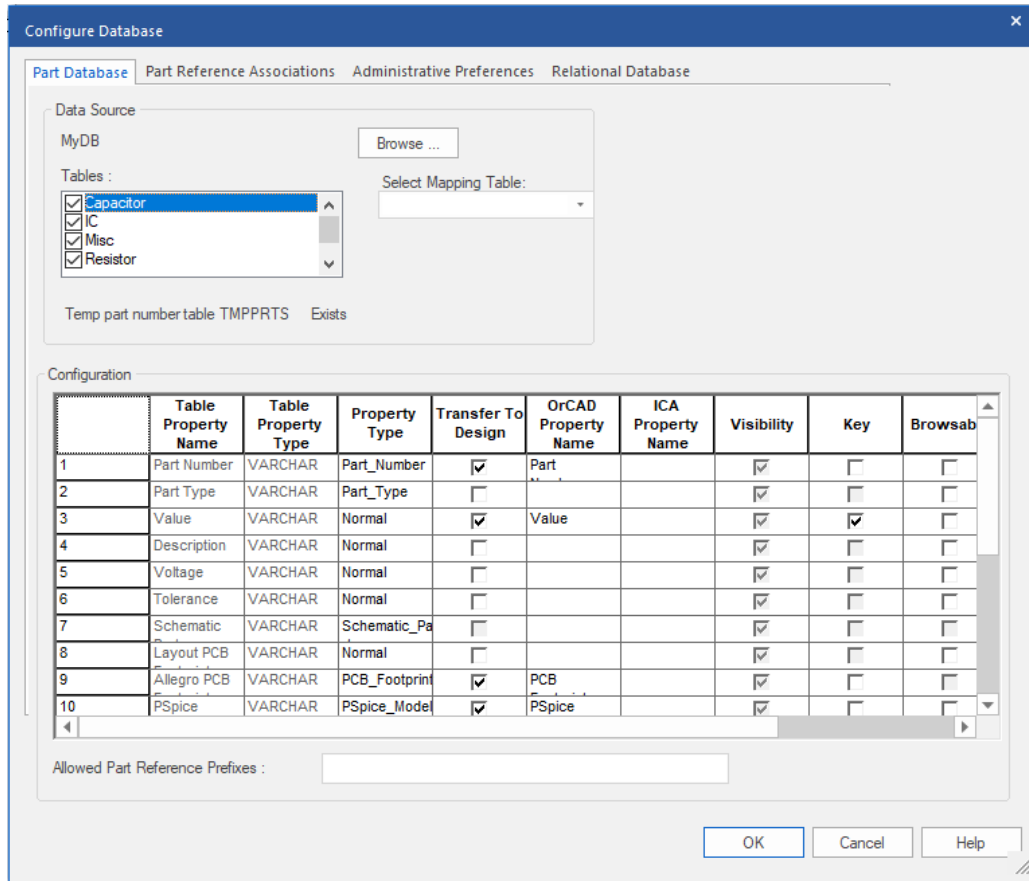
Click 'Next'

Define which keyed property CIS will use to find part matches when using the Link Database part option. Value is the default.



Click 'Finish'. You will get the following screen – which is a summary but also allows you to make changes if required or make other properties visible.

How to create a DBC file



Click 'OK' and you will be asked to browse for a filename to store your DBC file. This process is now complete.

The following are trademarks or registered trademarks of Cadence Design Systems, Inc. 555 River Oaks Parkway, San Jose, CA 95134
Allegro®, Cadence®, Cadence logo™, Concept®, NC-Verilog®, OrCAD®, PSpice®, SPECCTRA®, Verilog®

Other Trademarks

All other trademarks are the exclusive property of their prospective owners.

NOTICE OF DISCLAIMER: Parallel Systems is providing this design, code, or information "as is." By providing the design, code, or information as one possible implementation of this feature, application, or standard, Parallel Systems makes no representation that this implementation is free from any claims of infringement. You are responsible for obtaining any rights you may require for your implementation. Parallel Systems expressly disclaims any warranty whatsoever with respect to the adequacy of the implementation, including but not limited to any warranties or representations that this implementation is free from claims of infringement and any implied warranties of merchantability or fitness for a particular purpose.