

Title: Capture CIS Environment

Product: OrCAD Capture with CIS Option and Allegro DE CIS

Summary: It explains the whole Capture CIS environment.

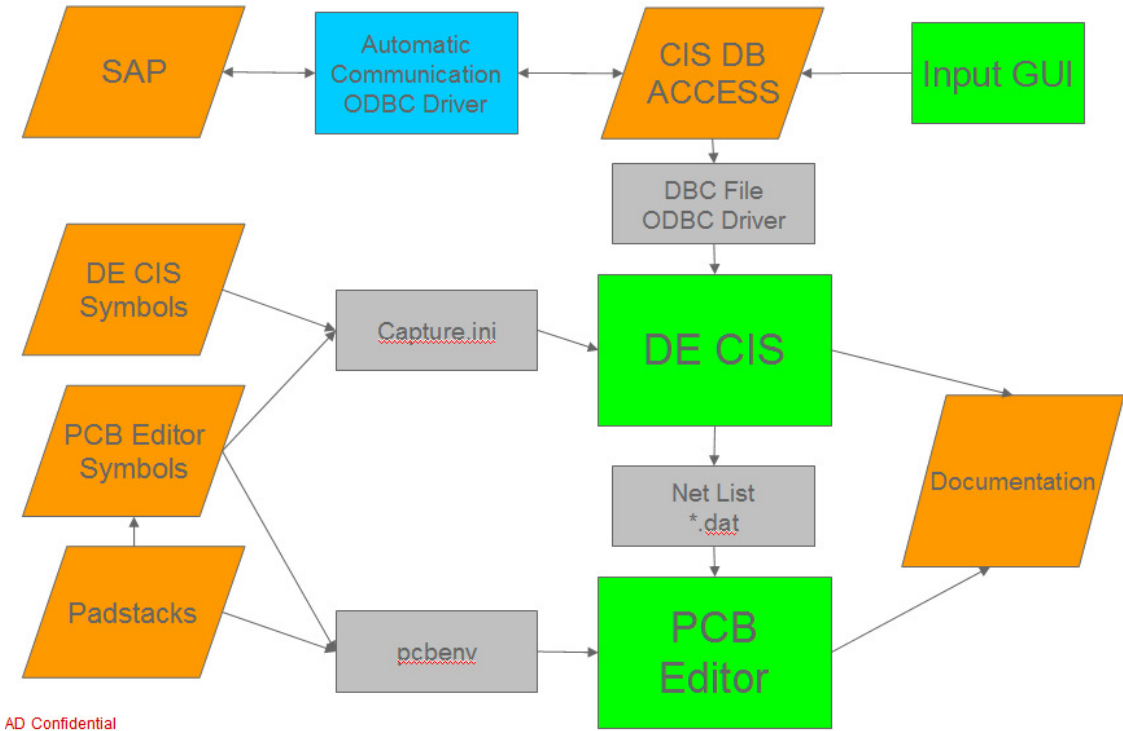
Author/Date: Beate Wilke / 11.11.2009

Table of Contents

- 1 Communication - Database, Component-Libraries, Capture..... 2
 - 1.1 Overview..... 2
 - 1.2 Property Flow..... 3
 - 1.3 Symbol Flow 4
- 2 ODBC Driver 5
 - 2.1 How to setup an ODBC Driver 5
- 3 DBC File..... 8
 - 3.1 Part Database Tab..... 9
 - 3.2 Part Reference Associations13
 - 3.3 Administrative Preferences14
- 4 Capture.ini.....17
 - 4.1 Configuring Footprint Libraries17
 - 4.2 Configuring Capture Libraries17
 - 4.3 Configuring Datasheet folders.....17
 - 4.4 Capture.ini - Location18

1 Communication - Database, Component-Libraries, Capture

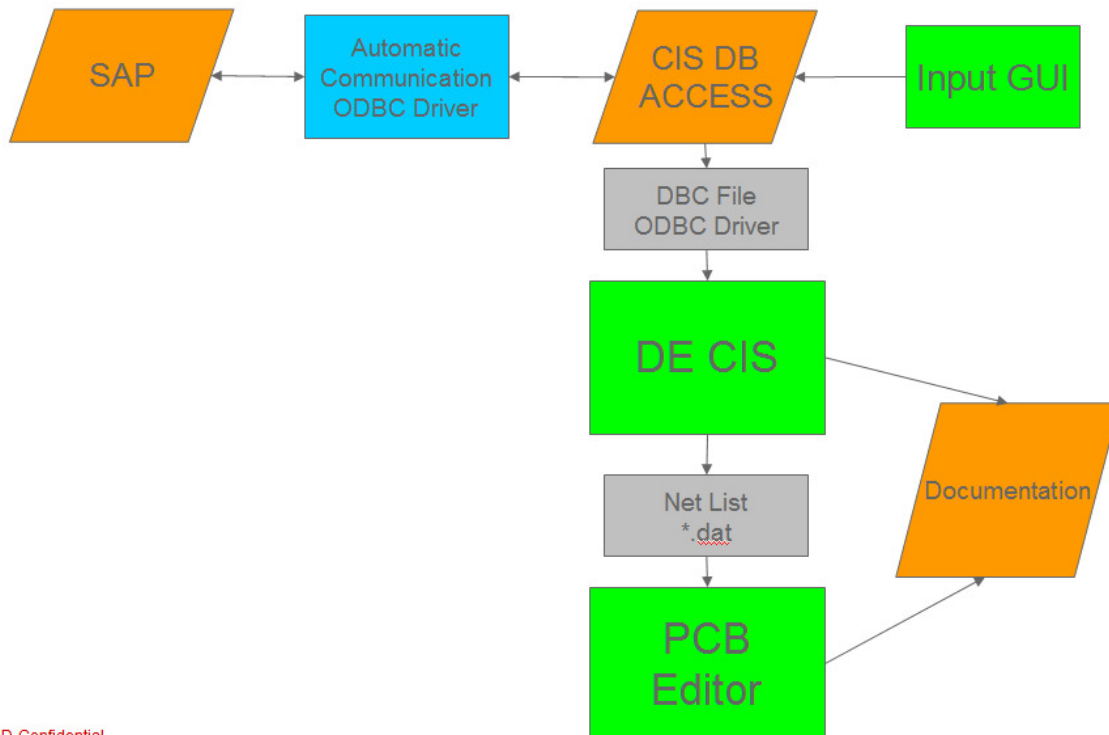
1.1 Overview



This is a simplified overview about a customers data structure. It shows how all the different data sources, files and tools work together.

We have two different main flows, one for the Properties and one for the Symbols.

1.2 Property Flow

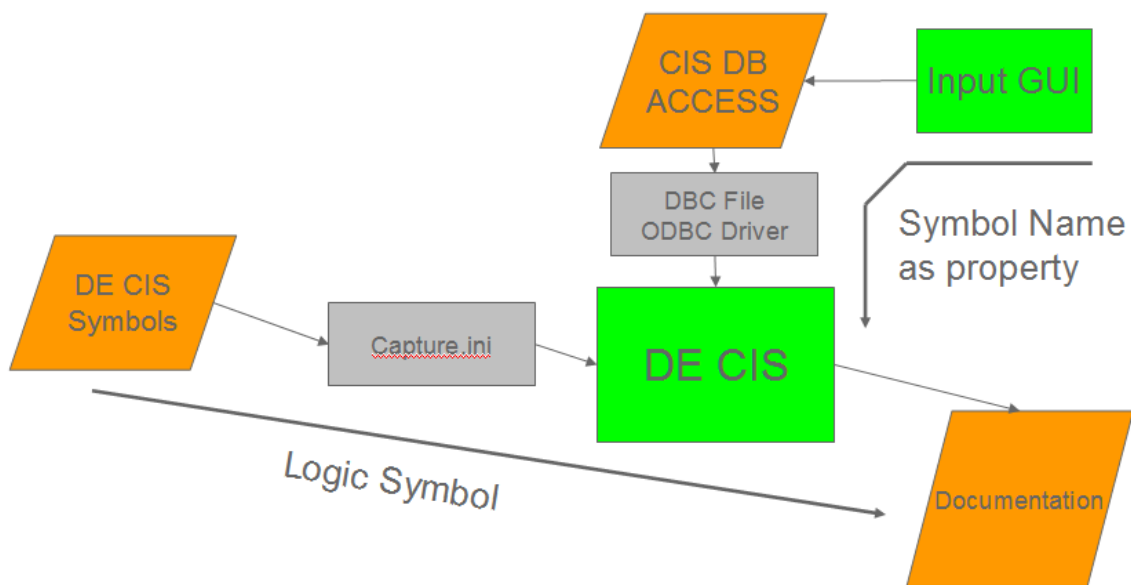


AD Confidential

The Input GUI of the CIS DB and SAP are input interfaces for all CIS Properties. SAP and CIS DB have a bidirectional interface. An ODBC driver has to be defined to read and write data from or to the CIS DB. The ODBC driver is also necessary for DE CIS to read information in the CIS DB. The Input GUI can be an Access form.

When a component is placed in the schematic DE CIS automatically loads the properties from the CIS DB. The DBC file defines the properties which are loaded with which function.

1.3 Symbol Flow



The symbol name is stored in the CIS DB. While selecting and placing a part in the schematics, DE CIS loads the Logic symbol from the symbol libraries.

2 ODBC Driver

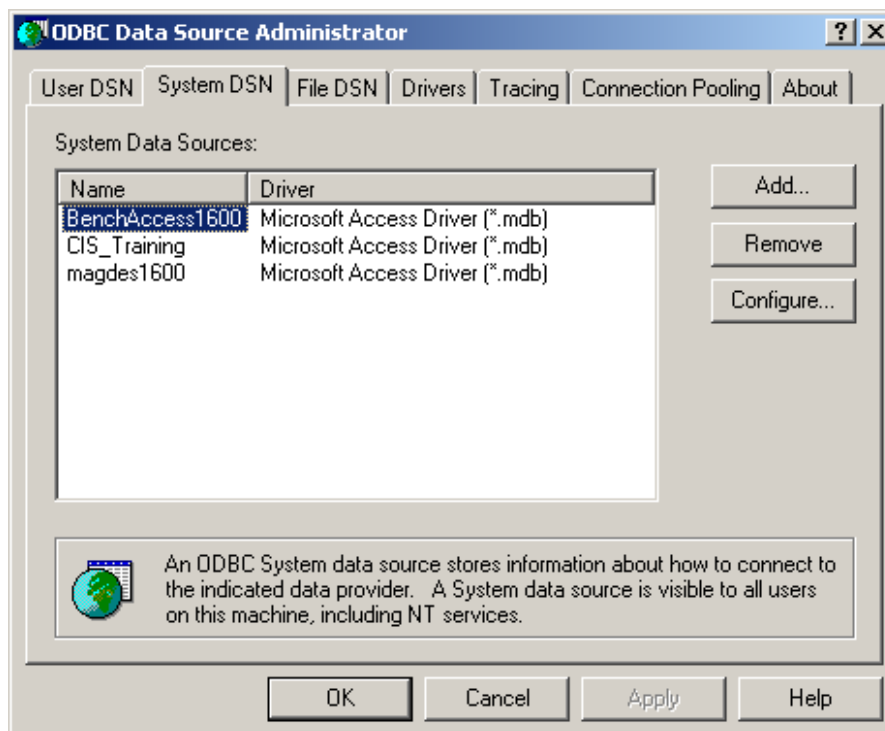
The ODBC Driver has to be defined on every client system. It's important to use always the same Data Source Name because the global DBC file uses this Data Source Name.

How to define an ODBC driver is described in the Lab.

Note: To define an ODBC driver you need administrator privileges on your system.

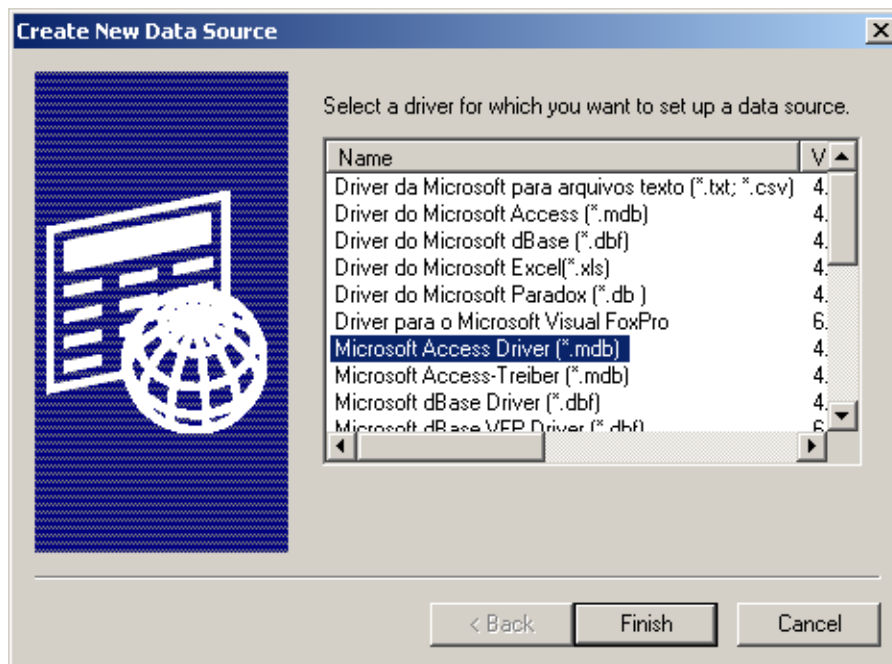
2.1 How to setup an ODBC Driver

For Windows XP select Start -> Settings -> Control Panel -> Administrative Tools -> Data Sources (ODBC). Select the System DSN Tab.



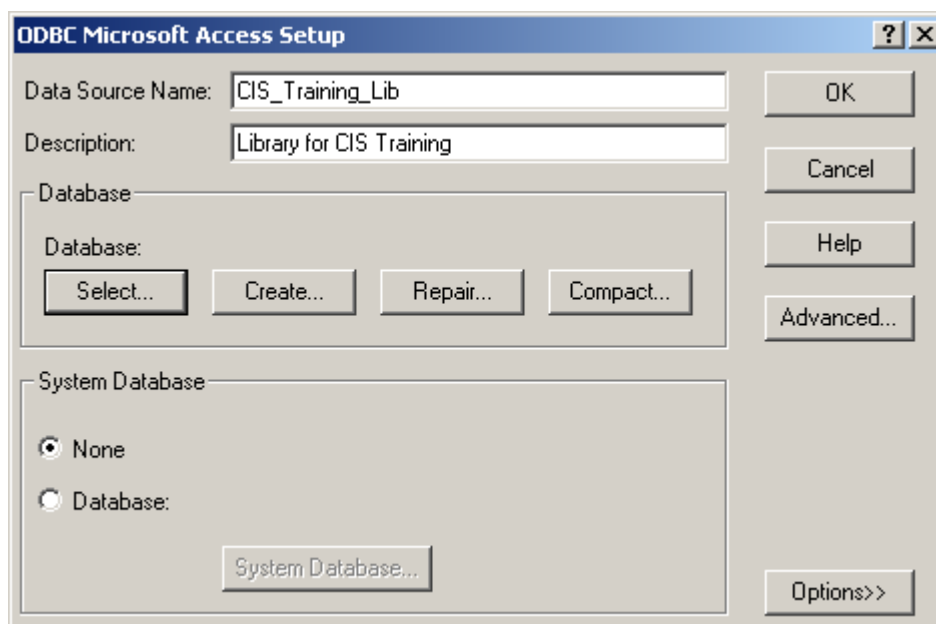
You will find the already installed ODBC driver for the Cadence BenchAccess1600 Library. This Example describes how to setup a driver for an Access Database. You can use the same method for Excel, SQL,.....

Select Add.



Search for Microsoft Access Driver, select it and select Finish.

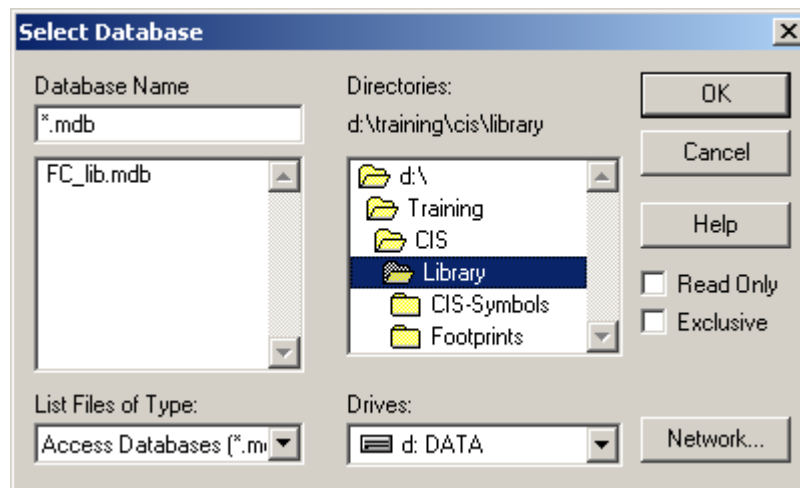
Note: If you have an English OS use the English Driver, if you have a German OS use the German Driver and so on.



Put in the Data Source Name. This is the reference key for the communication between database and CIS.

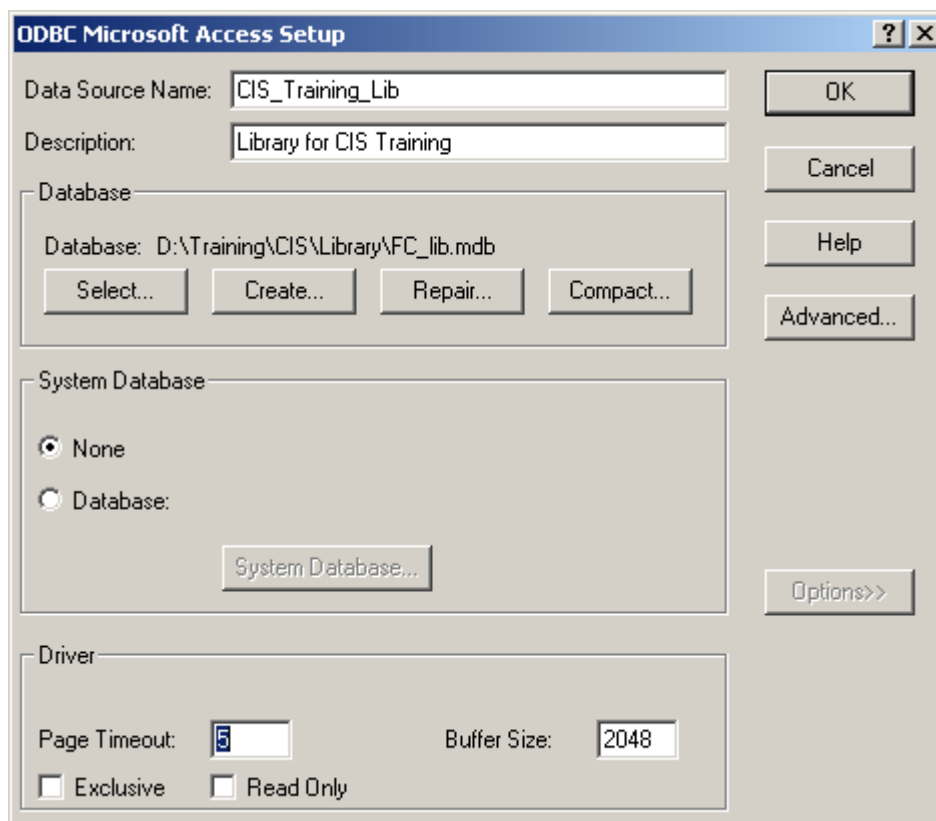
Enter a Description if you like. This is only for documentation.

Select the Select Button to brows for the Access database.



Go to P:\Cadence\CIS_DB and select the CIS_PartLib.mdb. Select OK.

Open the driver options by selecting the Options Button. Read Only must be unchecked.



Select OK to close the ODBC Microsoft Access Setup.

3 DBC File

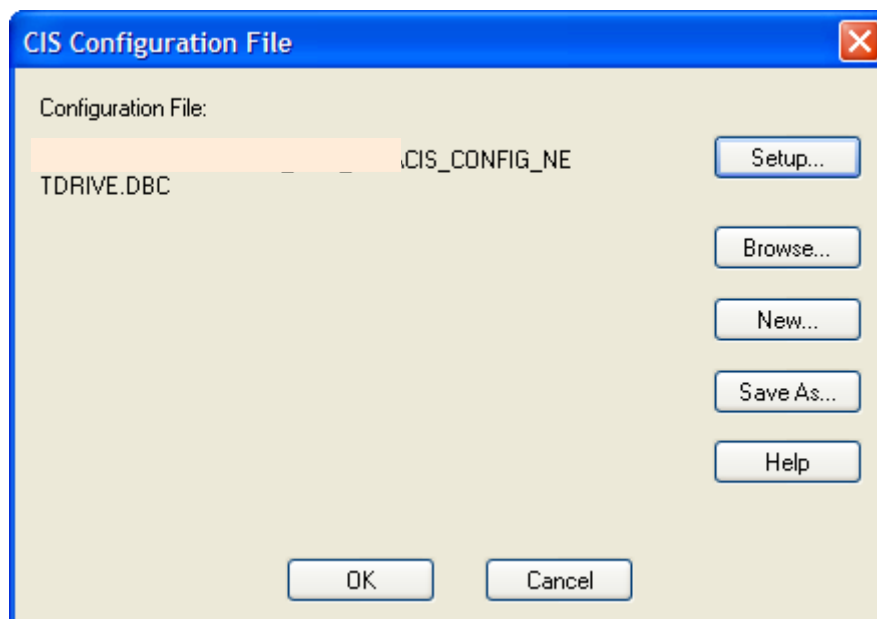
This chapter is only for library administrator.

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

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

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

To work with the DBC File in DE CIS select Options -> CIS Configuration

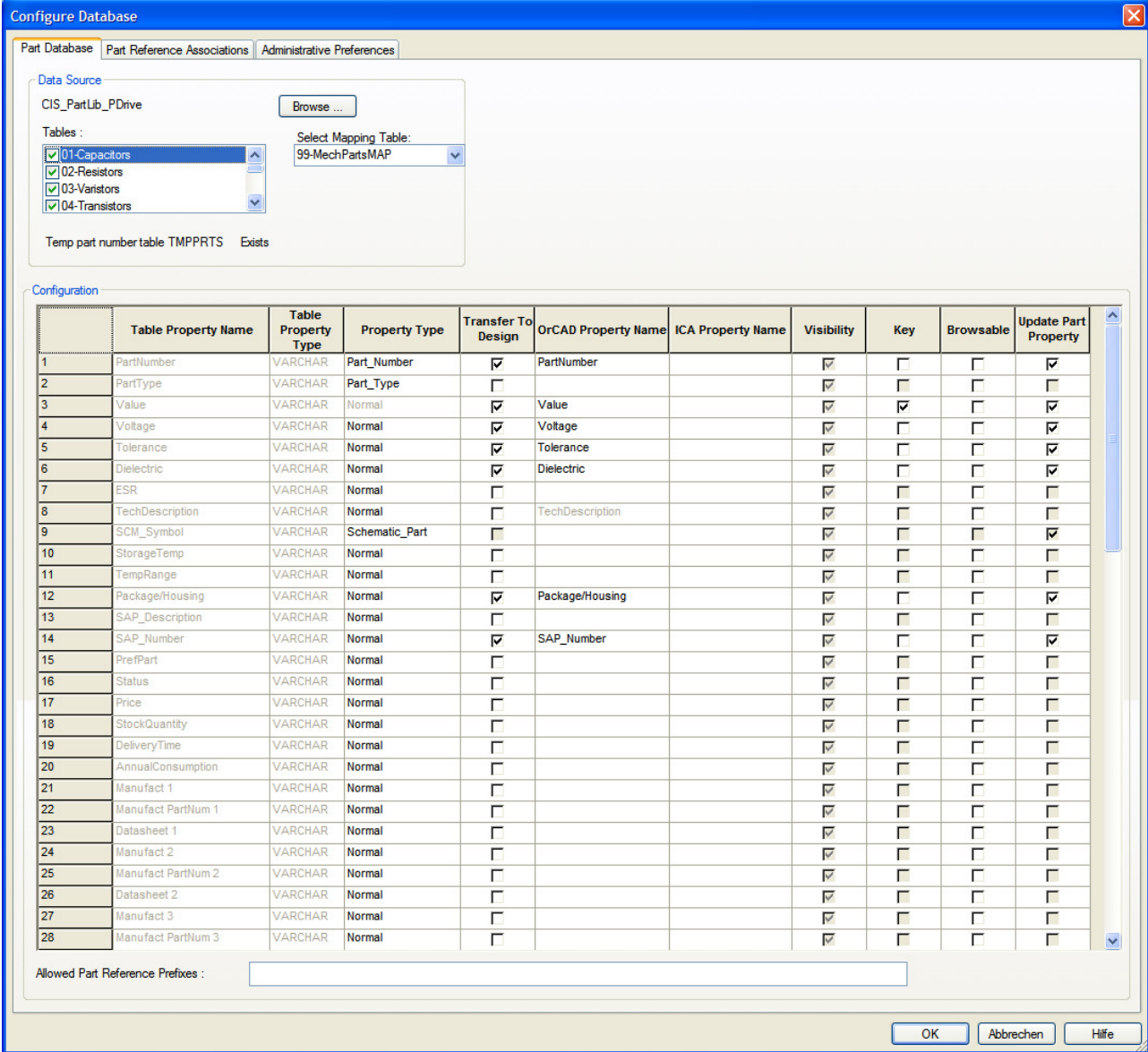


Select:

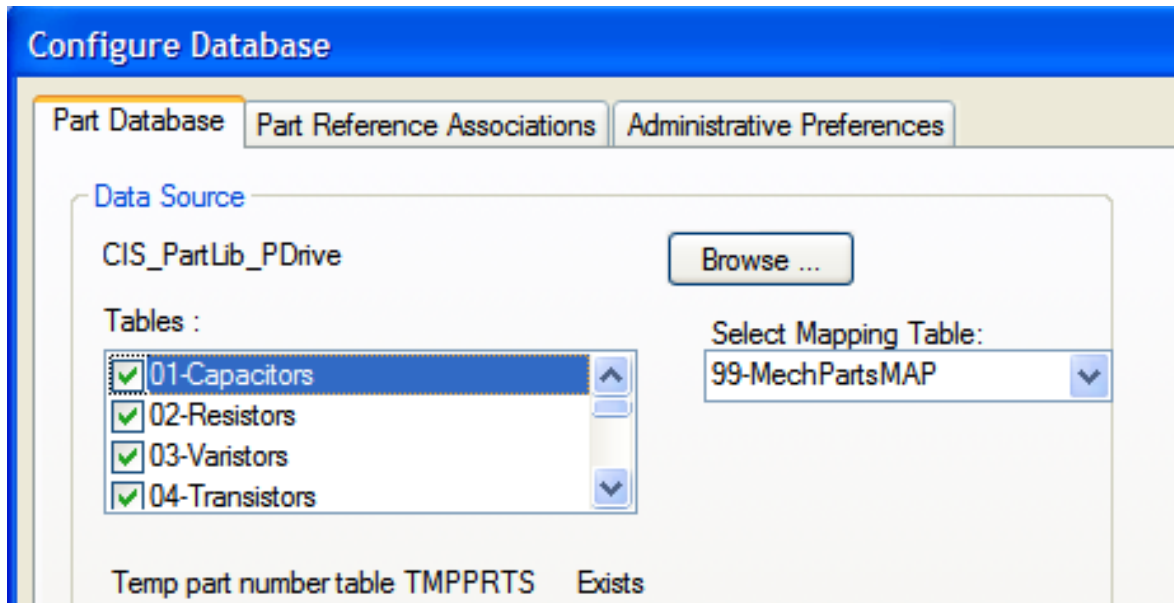
- Setup: To open the current file.
Browse: To browse for a different DBC file.
New: To create a new DBC file. This opens the Database Configuration Wizard.
Save As: To save the current file with a different file name.

3.1 Part Database Tab

After selecting Setup you will get this form:



In the upper left corner you find the name of the Data Source from the ODBC Driver and a list of all available Tables. You can choose what tables should be used by CIS by marking them.



On the right side of the picture you can see that Select Mapping Table is set to 99-MechPartsMap. This Mapping Table allows you to add mechanical parts to your electrical parts. The mechanical Parts are virtual parts which are only added to the BOM to create a complete export.

Configuration					
	Table Property Name	Table Property Type	Property Type	Transfer To Design	OrCAD Property Name
1	PartNumber	VARCHAR	Part_Number	<input checked="" type="checkbox"/>	PartNumber
2	PartType	VARCHAR	Part_Type	<input type="checkbox"/>	
3	Value	VARCHAR	Normal	<input checked="" type="checkbox"/>	Value
4	Voltage	VARCHAR	Normal	<input checked="" type="checkbox"/>	Voltage
5	Tolerance	VARCHAR	Normal	<input checked="" type="checkbox"/>	Tolerance
6	Dielectric	VARCHAR	Normal	<input checked="" type="checkbox"/>	Dielectric
7	ESR	VARCHAR	Normal	<input type="checkbox"/>	
8	TechDescription	VARCHAR	Normal	<input type="checkbox"/>	TechDescription
9	SCM_Symbol	VARCHAR	Schematic_Part	<input type="checkbox"/>	

The Configuration Area defines all properties

Table Property Name View the property name as defined in the part database.

Table Property Type This is the data type for the property. Most properties are type Text, but there may be other data types.

Property Type	<p>One of five types indicating how Capture CIS interprets the property. Except for Normal, as you assign a type, it is removed from the list of choices.</p> <p>Part_Number: One property in the table must have this type. This is used by the part manager to identify the part in the database.</p> <p>Part_Type: Set this type for the property that defines the part's type.</p> <p>Schematic_Part: Set this type for the property that indicates the schematic part name.</p> <p>PCB_Footprint: Set this type for the property that indicates the PCB footprint name.</p> <p>PSpice_Model: Set this type for the property that contains PSpice models. When you set this property type, CIS activates PSpice functionality for placed parts with a value for the database property.</p> <p>Activepart_ID: Set this type for the property that is used to store the unique identifiers for parts that are placed from the activeparts database through the ICA. When you set this property type, you can create reports about parts using the latest information from the activeparts database at the Activeparts website. In addition, you can view the latest information for a part directly from an instance on a schematic page.</p> <p>Normal: Use this for all other properties in the table.</p>
Transfer To Design	<p>Check to have Capture CIS copy this property to the placed part when you place or link the part.</p>
OrCAD Property Name	<p>The name of the property as it appears on the placed part. This characteristic is only available when you check Transfer to Design. Select a property name from the drop-down list or type the property name. If you do not enter a value, the name of the property as it appears in the database is transferred to the placed part.</p>

ne	ICA Property Name	Visibility	Key	Browsable	Update Part Property
		<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
		<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
		<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
		<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
		<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
		<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
		<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
		<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
		<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
		<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

ICA Property Name The name of the internet component assistant property. The ICA property you specify is mapped to the corresponding property in the database. For example, you can map the ICA property Price to the database property Cost. Select a property name from the drop-down list.

Visibility Sets the visibility of the property on the schematic design. You can change the visibility settings only for properties that are transferred to the design. These appear with a white background in the visible column. Properties you cannot change are shown with a light grey background in the visible column.

Key Check to use the property as a key during the initial part search when linking a database part. If you don't have a Value property, don't set any key. You can specify any number of keys. Typically, you only set the key for the Value property. When you link a part, Capture CIS searches for parts with that specific value in the database.

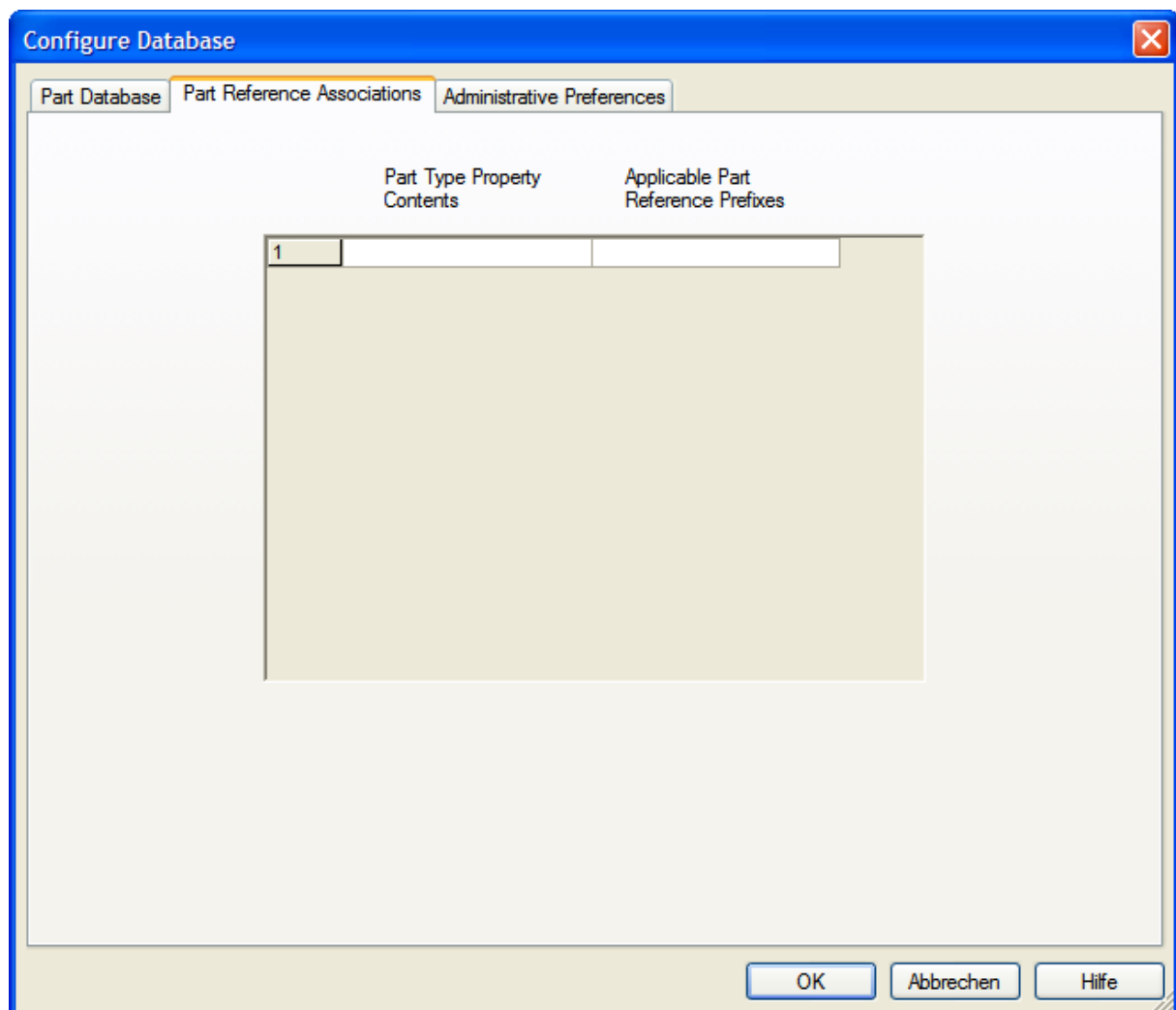
Browsable Check to indicate that this property in the database contains references to data sheets, drawings, or other documents. When you browse your local part database, click on hyperlinked (blue, underlined) values to view the referenced documents online. Capture CIS launches the appropriate browser or application based on the contents of the database property and the filename extension. In addition, when you generate a CIS standard bill of materials, you can click on the

link for a browsable property value in the report to view a document, such as the latest data sheet from its source on the Internet.

Update Part Property

Check to have CIS check the value of the property for placed parts against the database part's value when you update the part status of your designs.

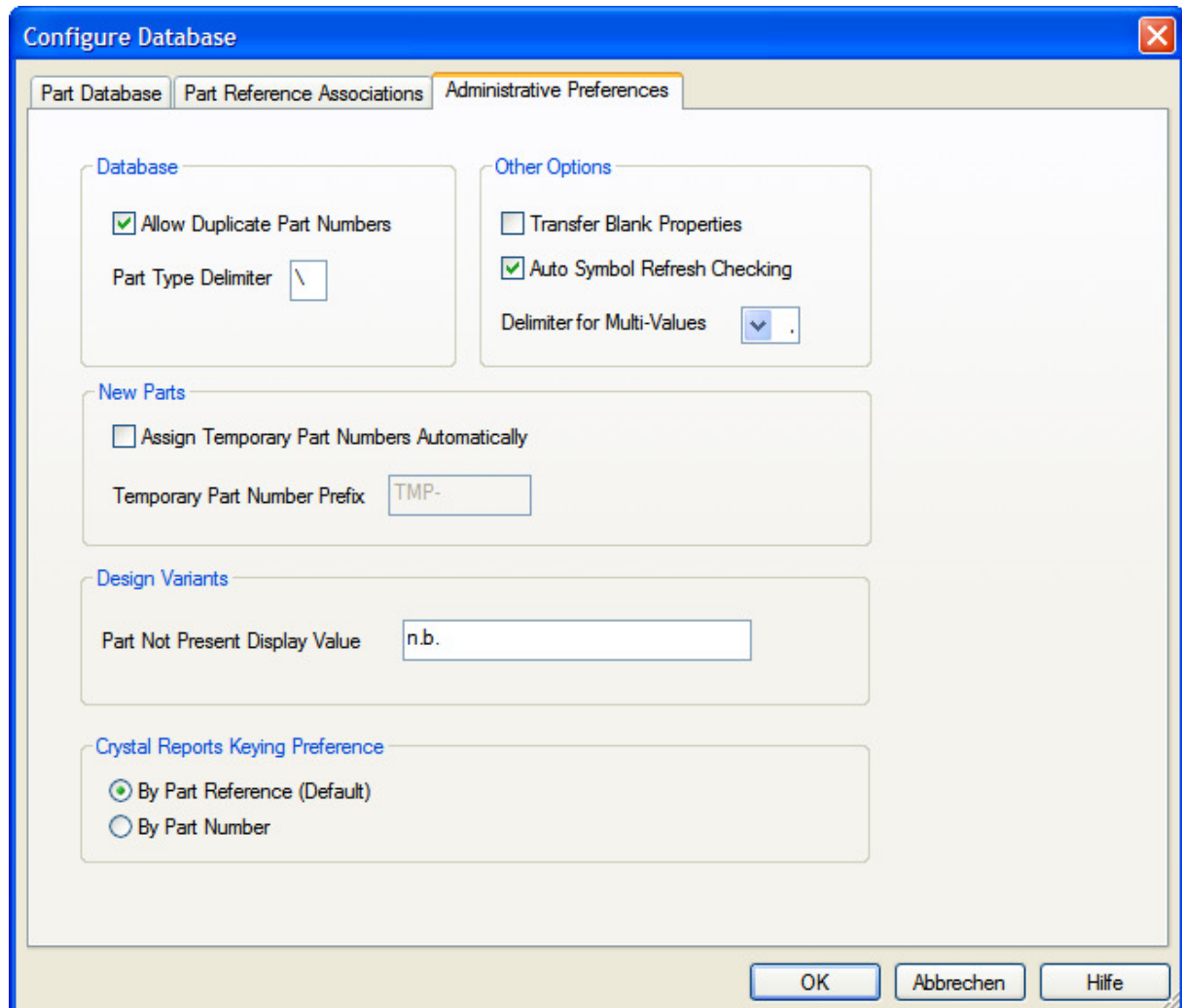
3.2 Part Reference Associations



Part reference associations are used to improve the speed and accuracy of the search for database parts to link to placed parts. You create associations between a particular part type and the prefixes in the part database for that part type. For example, you can create a part reference association for resistors such that all resistors in the part database use the R prefix.

3.3 Administrative Preferences

Administrative preferences allow you to customize some CIS features for your work environment.



Allow Duplicate Part Numbers Check to allow more than one part to use the same number in the database. This is most useful when a part has more than one PCB footprint associated with it. You can enter the part twice in the database, each one using a different PCB footprint.

Part Type Delimiter Indicates the folder hierarchy delimiter used in the database. The default delimiter is the backslash. For example, if you use Capacitor\Ceramic in the database, Capture CIS displays Ceramic as a folder in the Capacitor folder.

Transfer Blank Properties	Check to create a property on the placed part even if there isn't a value for the property in the database.
Auto Symbol Refresh Checking	Check to enable CIS to automatically detect if symbols or footprints were updated in the configured libraries. If any changes are detected, the Refresh Symbols from Lib command in the Update menu in CIS explorer and its corresponding icon on the toolbar are enabled. This indicates that you have to refresh the symbol or footprint information in CIS explorer. If you do not select this check box, the Refresh Symbols from Lib command in the Update menu in CIS explorer and its corresponding icon on the toolbar will always be enabled.
Delimiter for Multi-Values	Select the character you want to use to separate multiple values in the schematic part and PCB footprint table property cells of your part database.
Assign Temporary Part Numbers Automatically	Check to have Capture CIS to create and track temporary part numbers. Temporary parts are tracked in the TMPPTS table.
Temporary Part Number Prefix	Type the prefix you want to use for temporary parts. TMP- is the default. Capture CIS appends the temporary part number to this prefix. You can set up a different prefix (such as the user's initials) for each user's temporary parts.
Part Not Present Display Value	Enter the text description that you want CIS to use for variant parts set to Not Present. The property is displayed in the following locations: Part Number and Value fields in the part manager Design variant columns in variant reports Variant parts on schematic page previews and printouts. The Part Not Present Display Value does not display in Capture's schematic page editor. This property also cannot be repositioned or edited in the schematic page editor. For this reason, you have to print preview or print a schematic page to make sure that the value assigned to the property does not overlap another part or property display. A

long value is more likely to overlap a display, therefore, try to use a fairly short text equivalent for the default Not Present value.

By Part Reference (Default)

Select this option to group and sort parts in bills of materials reports created using Crystal Reports grouped by part reference. Each part will have a unique item number.

You can access part reference associations by clicking the Part Reference Associations tab in the Configure Database Dialog box. CIS uses part reference association during the Link Database Part command operation to improve the speed and accuracy of the search. When you choose Link Database Part, Capture CIS searches only the associated database table and displays the parts of the appropriate type.

Note: Defining part reference associations only improves part search speeds for true databases (for example, Microsoft Access)—there is no speed improvement from setting up associations if you are using a spreadsheet or a text file for your database.

By Part Number

Select this option to sort parts in bills of materials reports created using Crystal Reports by part number. Since part numbers are unique, the resulting report assigns one item number to all parts with the same part number.

4 Capture.ini

When you install DE CIS the system generates a default Capture.ini. During the work with DE CIS all changes and settings are stored in the capture.ini. Every user needs his own Capture.ini to store his personal settings. You have only to add your path definitions for Footprint and Part Libraries.

4.1 Configuring Footprint Libraries

In order to display PCB footprints, you must have access to PCB Editor Libraries. If you have not pointed your footprint libraries at the time of installation, you can add the following lines to your `Capture.ini` file.

For configuring Footprint viewer for Cadence Allegro, add the following lines:

```
[Footprint Viewer Type]
Type=Allegro
[Allegro Footprints]
Dir0=D or C:\(your install directory)\tools\pcb\share\symbols
```

You can also add custom footprint library locations by adding “DIR1” after the “Dir0” line and add your full path name to the libraries.

4.2 Configuring Capture Libraries

You can configure your own custom Capture part and symbol libraries, as well as the OrCAD Capture libraries, by adding the library locations in the `Capture.ini` file under **[Part Library Directories]**. (This is an example, only.)

```
[Part Library Directories]
Dir0=C:\Program Files\EMA\SYMBOLS
Dir1=C:\OrCAD\OrCAD_16.0\tools\Capture\Library\
```

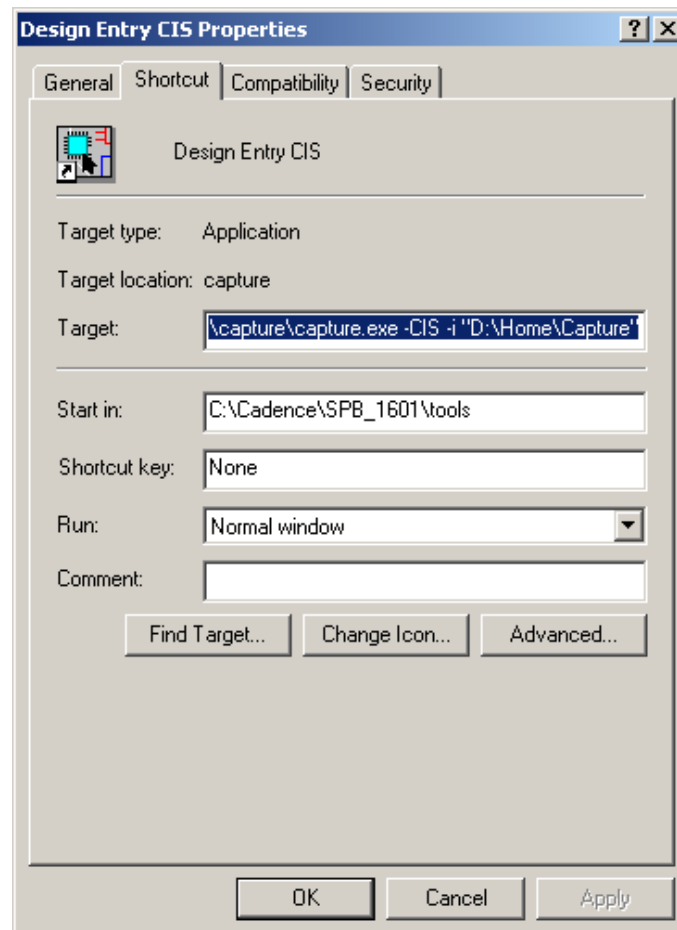
4.3 Configuring Datasheet folders

If you have different folders for datasheets add the path information to capture.ini. The syntax is:

```
[CIS Browse Directories]
dir0=path1
dir1=path2
```

4.4 Capture.ini - Location

For XP select Start -> Programs -> Cadence SPB 16.01 and right click on Design Entry CIS. Open the properties menu.



In the field Target you find these settings:

e.g.: C:\Cadence\SPB_1601\tools\capture\capture.exe -CIS -i "D:\Home\Capture"

C:\Cadence\SPB_1601\tools\capture\capture.exe is the path to the Capture.exe.

-CIS starts Capture with CIS Option

-i "D:\Home\Capture" is the path to the Capture.ini

Select OK to close the window.

By default this path is <CDSROOT>/tools/capture. We changed it to a folder where the user has full privileges.

Note: If the user do not has full system privileges to the install area of the Cadence tool it's important to change this path to a folder with full privileges. The Allegro HOME folder would be perfect.