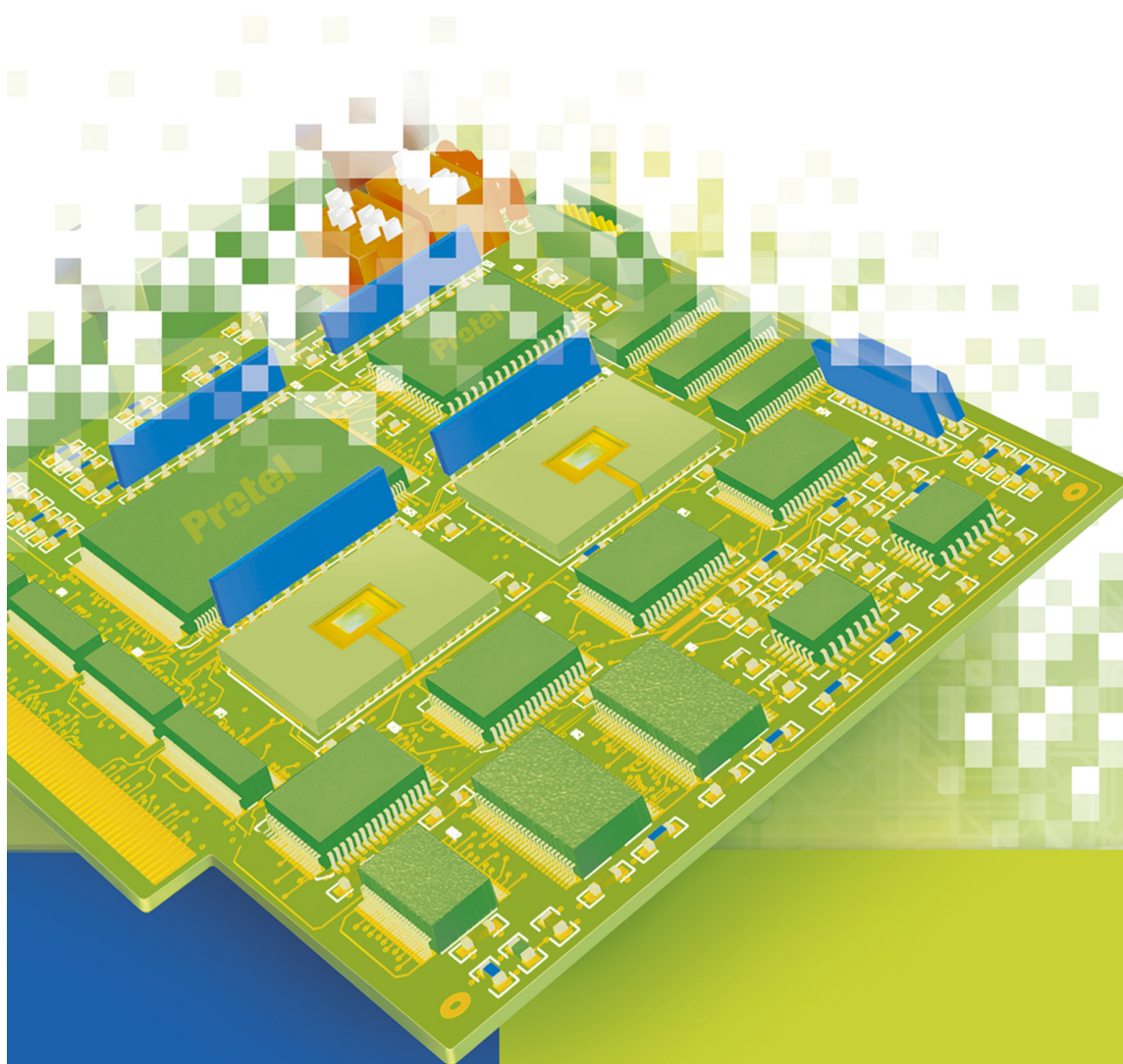


Multi-channel designs

ProtelDXP™

Tutorial



Protel®

Board-level design system from Altium.

Table of Contents

Multi-channel designs in Protel DXP	2
Creating a multi-channel design	2
Viewing the channel designator assignments	3

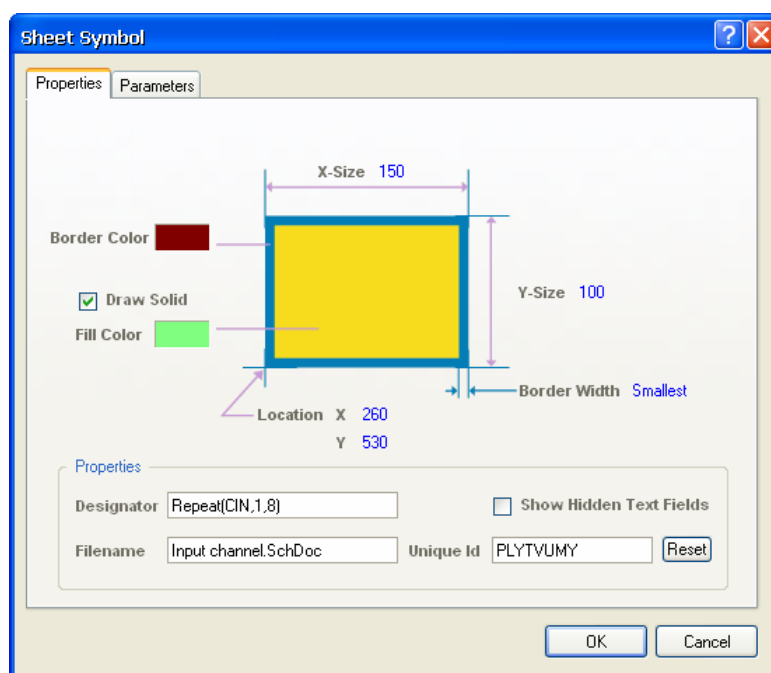
Multi-channel designs in Protel DXP

This tutorial shows how to create a multi-channel design using Protel DXP.

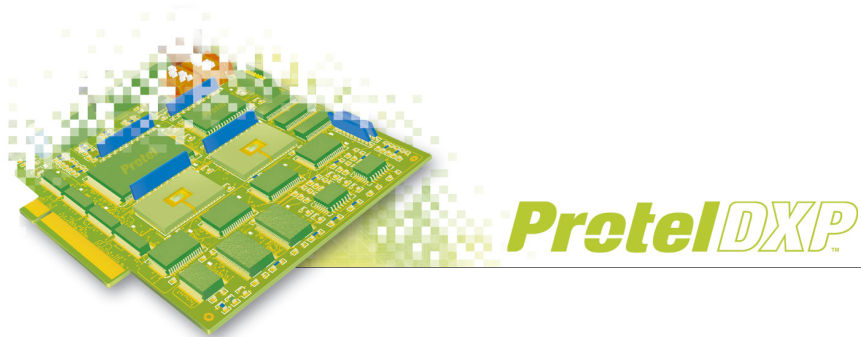
A multi-channel design refers to the same channel many times. The channel needs only to be drawn once as a separate schematic sub-sheet and included in a project. You can easily nominate how many times the channel is used through a sheet entry on the parent sheet of the schematic project. The Designator Manager creates and maintains a table of channel connections which is stored as part of the Projects file. A multi-channel project is supported throughout the design process, including back annotation of designator changes to the project file.

Creating a multi-channel design

1. Create the circuit that you wish to become the channel on a separate schematic and add the new schematic to the PCB project file.
2. On the parent sheet, use the **Place » Sheet Symbol** command to create a sheet symbol to represent the new channel schematic. The name of the sheet symbol is used to uniquely identify each component in each channel.
3. Double-click on the new sheet symbol to display the **Properties** tab of the *Sheet Symbol* dialog.



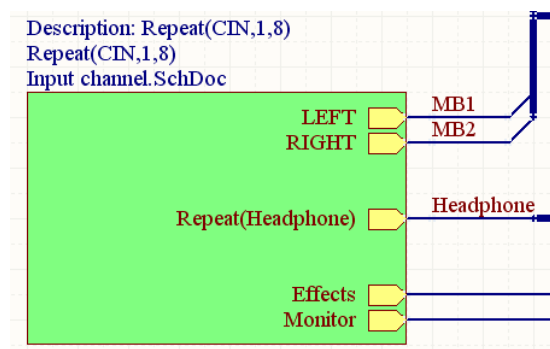
In the example above, the Name of the sheet symbol is CIN. You can use any name but short names are recommended for the sheet symbols to keep the designators short. This is because the sheet symbol name and a channel number will be added to the designator name when the project is compiled, e.g. R1 will become R1_CIN1.



4. In the **Filename** field, enter the name of the channel schematic you want to use, e.g. Input channel.SchDoc.
5. Nominate how many times you wish to reference the channel schematic by entering the Repeat Channel command in the **Name** field. The format is:

`Repeat (sheet_symbol_name , first_channel , last_channel)`

So in the example, the command `Repeat (CIN , 1 , 8)` in the Name field will reference the input channel schematic eight times (1,8) via the sheet symbol 'CIN'.



Nets that are common to all sub-sheets, for example, the Monitor net, are connected in the normal way. Nets that connect individually to each sub-sheet are bought in as a bus, with one bus element connecting to each sub-sheet.

Note that the net label on the wire does not include a bus element number and the sheet symbol includes the Repeat keyword. When the design is compiled, this bus is resolved into the individual nets (Headphone1, Headphone2 through to Headphone8) with one assigned to each channel.

6. Compile the project by selecting **Project » Compile PCB Project**. When the multi-channel design is compiled, there is still only one sheet shown in the Schematic Editor, however now, there are tabs displayed along the bottom of the schematic sheet in the design window, one for each channel. The tab names are the sheet symbol names plus the channel number, e.g. CIN1, CIN2.
7. Once the design has been compiled, it is transferred to the PCB Editor in the normal way (**Design » Update PCB**). The transfer process will automatically create a component class for each schematic sheet in the design, a room for each component class and group the components in each class in their room, ready for placement.
8. After placing and routing one channel, select **Tools » Copy Room Formats** in the PCB Editor to copy the placement and routing of that channel to the other channels.

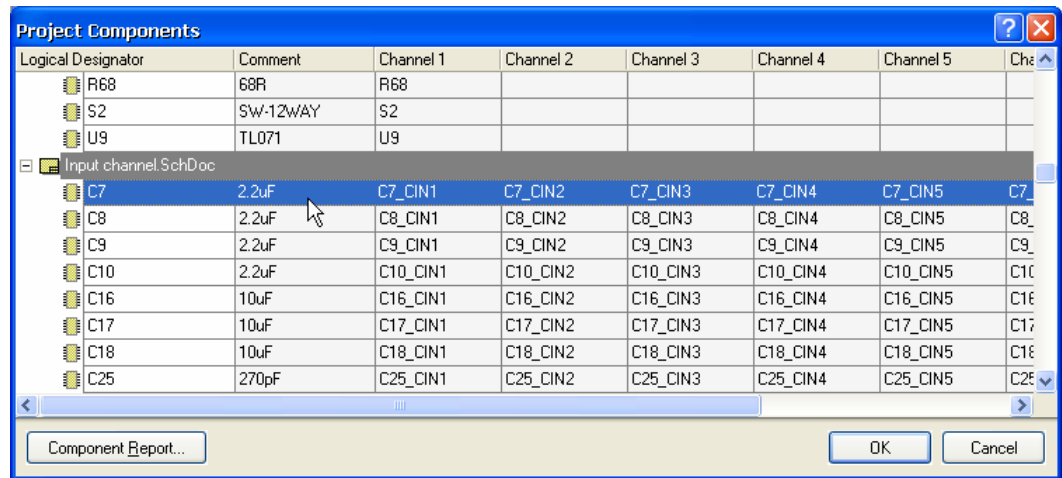
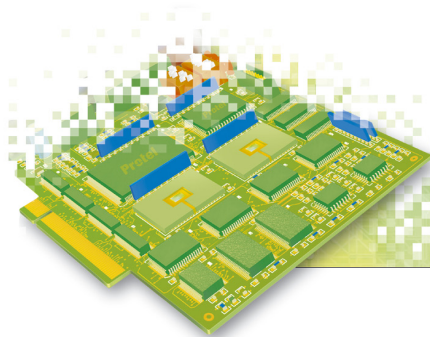
Viewing the channel designator assignments

To check the designators that are associated with components in a multi-channel design:

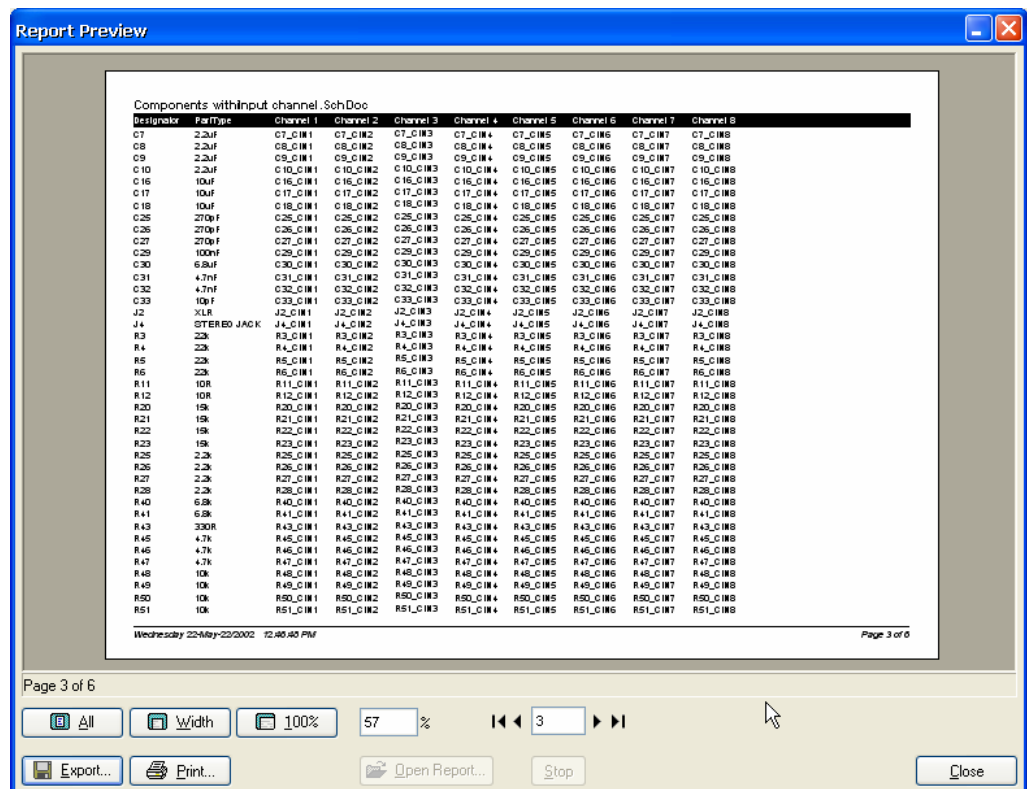
1. Select **Project » View Channels** to display the *Project Components* dialog.

The table will show the number of channels associated with a schematic name in the project. Each channel will have the designator names augmented with channel number, e.g. designator C7 becomes C7_CIN1 for channel 1 through to C7_CIN8 for channel 8.

Remember that there is always only one schematic sheet of the channel; the designator assignments for each channel are stored in a table (**Project » View Channels**).



- Click on a Logical Designator to jump to that component which is centered in the schematic in the design window.
- Click on the **Component Report** button to display the *Report Preview* dialog showing a print preview of the Project Components report.



- Click **Print** to print the report. The *Print* dialog displays. Click **OK** to send the report to the printer.
- Choose **Export** from the *Report Preview* dialog to save the Project Components report as a file, for example as a spreadsheet (.xls) or a .pdf.
- Save the file and you can then open it in its appropriate program (e.g. Microsoft Excel or Adobe Reader) by clicking on **Open Report**.
- Click on **Close** to exit the report preview mode and click **OK** to close the *Project Components* dialog.