Migration Guide from Cadence OrCAD 16 to Altium Designer 18

Making the Switch to Altium Designer

Altium®

Table of Contents

Altium ®

Introduction

This guide is intended for users who are switching from OrCAD / Allegro PCB Designer to Altium Designer. Even if you opened Altium Designer first time and are not familiar with working in this software yet, this guide will allow you to significantly speed up exploring of its basic features. The guide was written by highly experienced experts of OrCAD / Allegro PCB Designer and Altium Designer features who studying these widely used software systems for years. Considering your experience with Cadence design software you can easily switch to Altium Designer which has significant advantages over other solutions.

The guide is built that way you can compare OrCAD and Altium at all stages of design process – from creating a project and libraries to generating of manufacturing data. The guide covers the following topics:
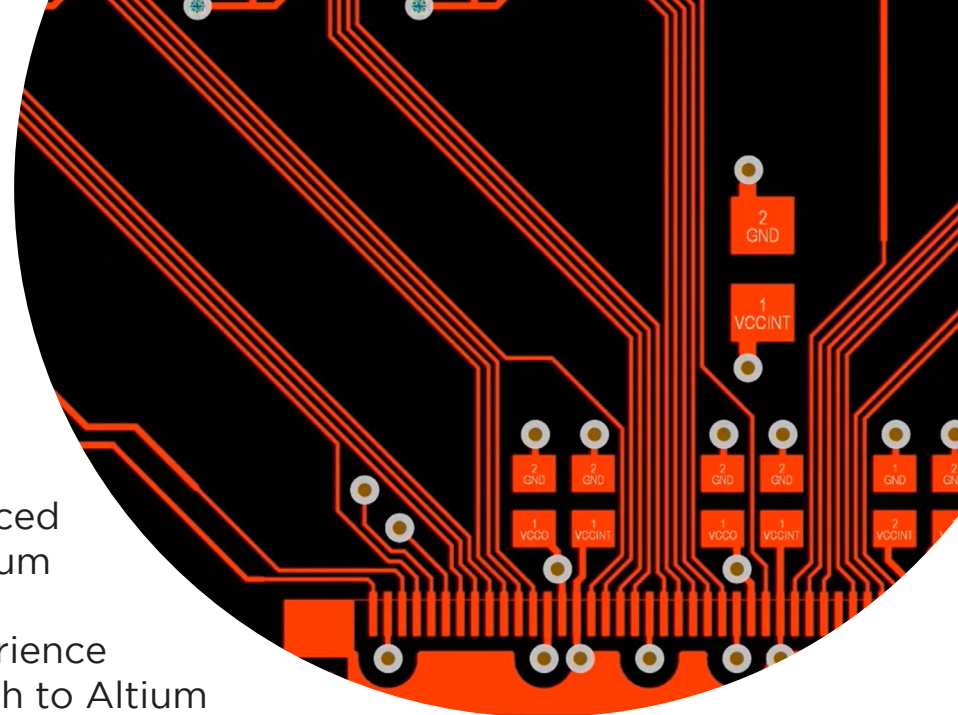
- Altium Designer user interface
- Basic settings
- Creating component libraries
- Creating footprint libraries
- Integrated library compilation
- Schematic capture

- Schematic post-processing
- PCB template configuration
- Placement and routing
- Design rule check
- Manufacturing data generation

Each section considers related Altium Designer and OrCAD PCB Designer tools together. Of course such questions as Bill of Materials report generation, Annotation, Search and Navigation and so on will be covered.

The main advantage of Altium Designer over other design software is unified design environment, which allows any design editor you need be launched. The software implements common approach to the management of schematics and layout design objects. This fundamental difference from other design software such as Cadence OrCAD / Allegro or Mentor PADS helps you quickly master Altium Designer tools and move to practice. Handy controls, such as Active Bar and the Properties panel, implemented in SCH and PCB editors, make Altium Designer an incredibly convenient and intuitive PCB design software.

Today, Altium Designer is the only PCB design software affordable for organizations of any level, as well as for individual designers. Altium Designer has all the features required for designs of any complexity. Even if you had been working for a long time in other software solutions, you can easily migrate your work to Altium Designer with excellent importers built into Altium Designer. Added to this is perfect integration into electromechanical design workflow in MCAD systems. Powerful 3D environment in Altium Designer has high performance and high quality visualization. You can even create multi-board projects where several PCBs are interconnected electrically.

So let's get started!
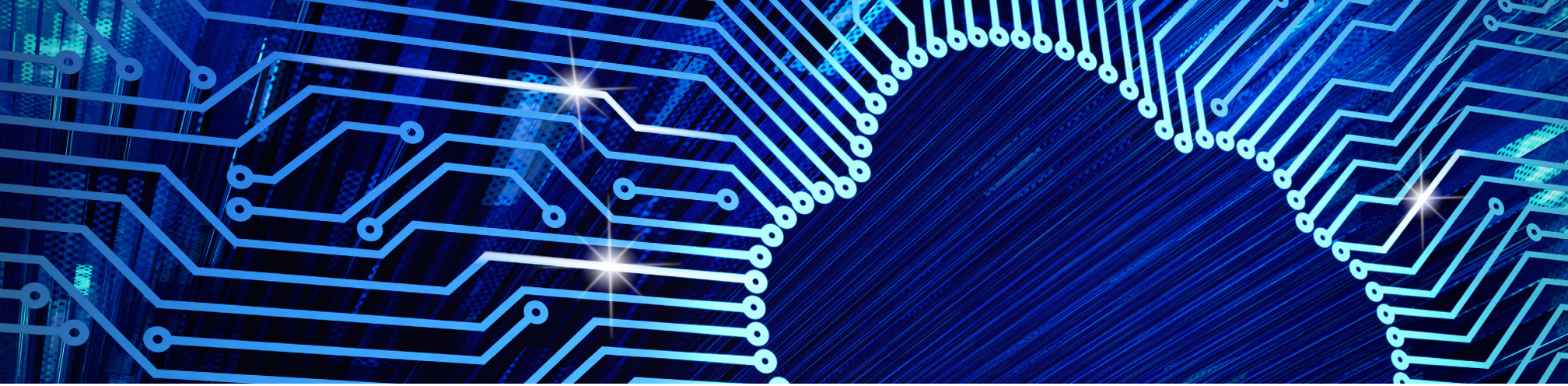
Altium®

1. Editors and interfaces

Unified user interface and applications

Altium Designer is unified design environment where any editor is being launched. This does not prevent you from working with several monitors and, for example, transfering PCB editor to another monitor (just drag document tab to another monitor). The software's control system relies on various panels that can be easily activated with the Panels button at the bottom right of the application. The main ones are Project and Libraries panels. A panel can be in open state, such as Projects, or be minimized, such as Libraries (Fig. 1.1). You can freely move panels from one edge of the window to another. There is so called Active Bar, which includes most frequently used tools.

If you double-click document in the Projects panel, for example, a schematic sheet, this document will appear in the workspace, and its tab will be shown above the document. You can open multiple documents and switch between them using Ctrl+Tab keyboard shortcut or by clicking on document tabs (Fig. 1.1).
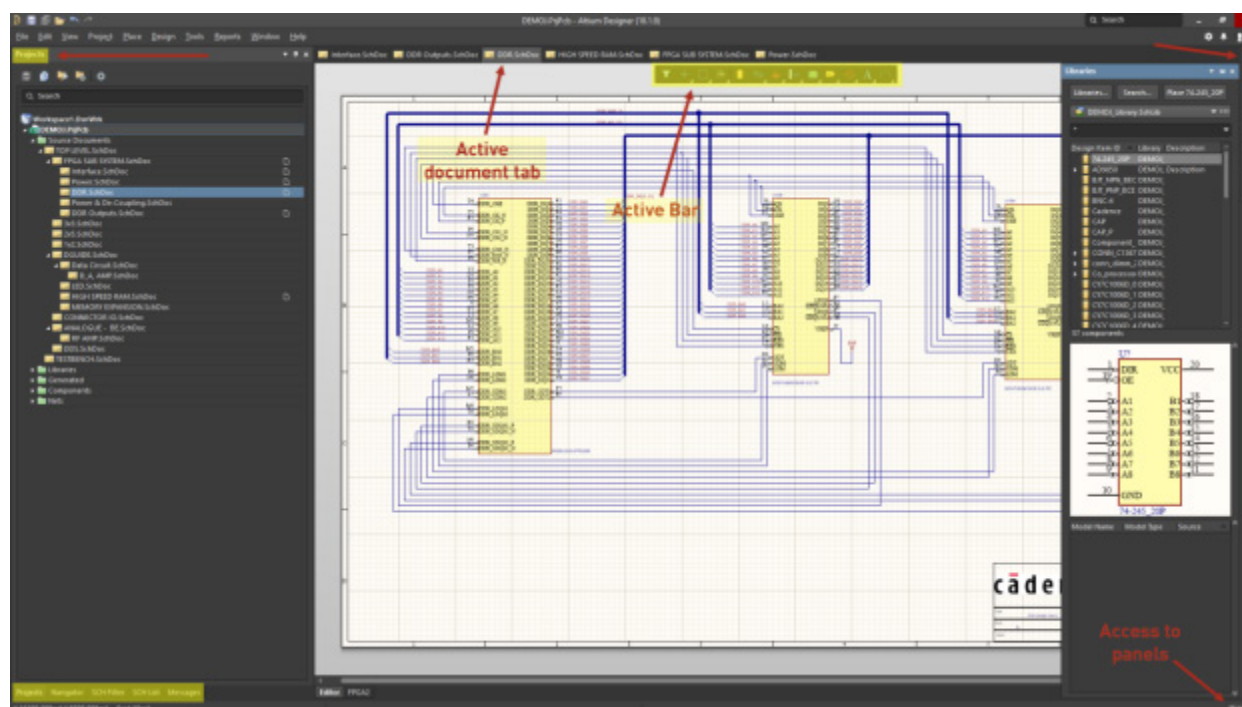

Fig. 1.1 - Altium Designer user interface

You can right-click on active document tab and choose the Split Vertical command from context menu for displaying this document in a separate window. The Merge All command of this context menu allows you to return to single window state.

Your Demoj project is opened, and its structure is displayed in the Projects panel with the new PrjPcb extension. Open the Libraries\Schematic Library Documents folder from project structure then open the Demoj_Library.SchLib library. The Sch Library panel tab becomes available at the bottom right of the window. Click on it to review the structure of translated project library. This library was created automatically from all components that are placed in the schematic. It is also available in the Libraries panel. You can open this panel as shown on Fig. 1.1 and choose it from the drop-down list at the top of the Libraries panel. A library should not necessarily be part of a project.

OrCAD PCB Designer

As you can see, there are a lot of differences between organization of Altium Designer and OrCAD PCB Designer. OrCAD is set of different applications that launched from Start menu or when transferring a netlist, as is the case with the OrCAD PCB Editor. OrCAD Capture is completely different from OrCAD PCB Editor in terms of its control system and ideology – these are completely different applications.

OrCAD Capture also has its own Project Manager, but not all files included in a project are displayed here (Fig. 1.2).
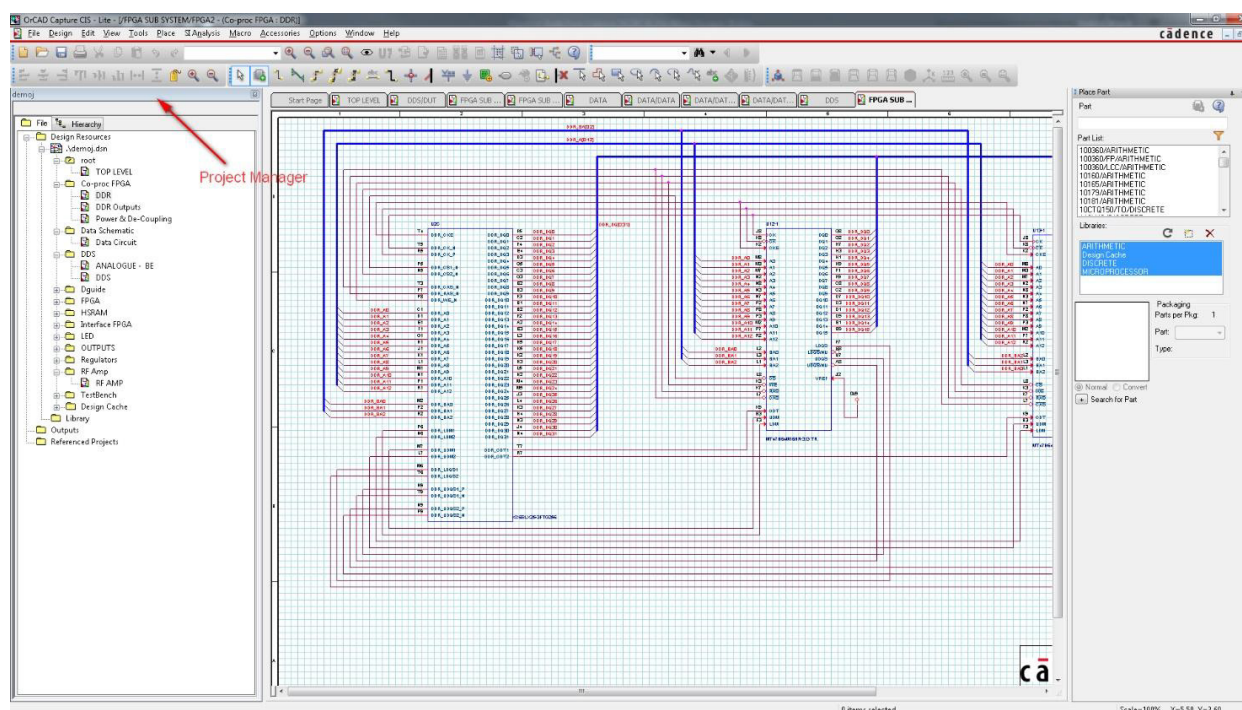

Fig. 1.2 - OrCAD Capture user interface

OrCAD Capture has traditional user interface where all commands are launched either from the main menu or from toolbars. For working with libraries the Place Part panel is opened. However, the set of panels is limited here and, for example, all properties of individual design objects, nets, components, power and ground symbols, and so on are edited with the dedicated Property Editor. In Altium Designer, the properties of these objects are available for editing in the Properties panel, which is more convenient.

2. Design workflow

This section covers design workflow in Altium Designer. It is very similar to that in OrCAD PCB Designer but is more organized here.

The PCB design workflow that you had been using in OrCAD PCB Designer is essentially to work with two main applications – OrCAD Capture for creating components and capturing schematic, and OrCAD PCB Editor for creating footprints and layout. It also can include circuit simulation in OrCAD PSpice, signal integrity analysis in OrCAD Signal Explorer (OrCAD PCB SI), FPGA projects preparation in OrCAD FPGA System Planner, and autorouting with SPECCTRA (Allegro PCB Router) (Fig. 2.1).
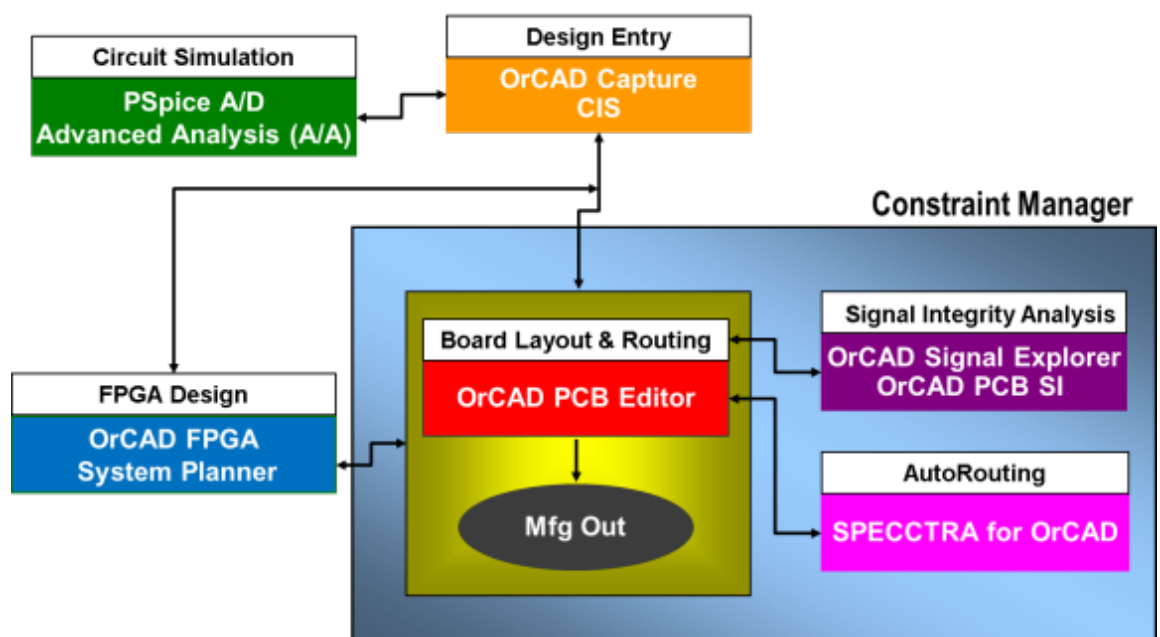

Fig. 2.1 - Design workflow in OrCAD PCB Designer

Data transferring between OrCAD Capture and OrCAD PCB Designer is strictly based on a text netlist consisting of three pst*.dat files. All component and net properties, design rules, class and room information are also transferred via netlist, taking into account the special configuration file allegro.cfg.

Altium Designer offers a more streamlined approach – a PCB project is created based on the main project file, which combines all source documents – schematic sheets, layout files (it is possible to include several layout files in a project), library files, etc. When the ECO process is performed, all changes, including classes, rooms, design rules, etc., are shown in the dedicated window where you can reject or leave any of the changes to be transmitted. You can fine-tune the ECO process with the project options. In addition, CAMtastic application for viewing and processing manufacturing output files is also included in Altium Designer (Fig. 2.2).
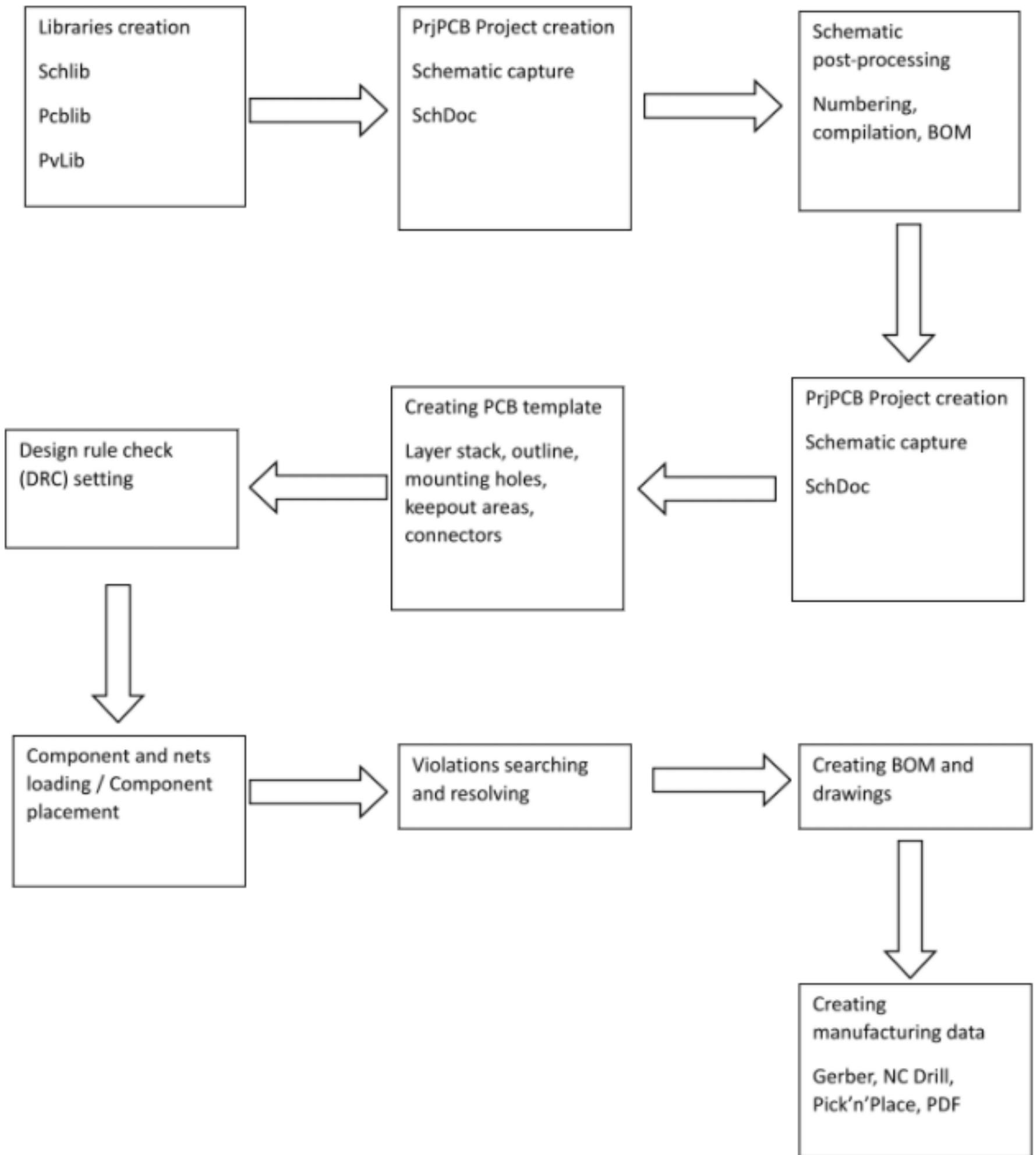
```
┌─────────────────────┐        ┌─────────────────────┐        ┌─────────────────────┐
│ Libraries creation  │        │ PrjPCB Project      │        │ Schematic           │
│                     │        │ creation            │        │ post-processing     │
│ Schlib              │  ⟹    │                     │  ⟹    │                     │
│                     │        │ Schematic capture   │        │ Numbering,          │
│ Pcblib              │        │                     │        │ compilation, BOM    │
│                     │        │ SchDoc              │        │                     │
│ PvLib               │        │                     │        │                     │
└─────────────────────┘        └─────────────────────┘        └─────────────────────┘
```

Fig. 2.2 - Design workflow in Altium Designer

Design workflow diagram:

- **Libraries creation** — Schlib, Pcblib, PvLib
  ⟹
- **PrjPCB Project creation** — Schematic capture, SchDoc
  ⟹
- **Schematic post-processing** — Numbering, compilation, BOM
  ⬇
- **PrjPCB Project creation** — Schematic capture, SchDoc
  ⟸
- **Creating PCB template** — Layer stack, outline, mounting holes, keepout areas, connectors
  ⟸
- **Design rule check (DRC) setting**
  ⬇
- **Component and nets loading / Component placement**
  ⟹
- **Violations searching and resolving**
  ⟹
- **Creating BOM and drawings**
  ⬇
- **Creating manufacturing data** — Gerber, NC Drill, Pick'n'Place, PDF

Fig. 2.2 - Design workflow in Altium Designer

3. Project file structure

In this section we will look at the structure of an Altium Designer project and its files. There are a number of key features in comparison with OrCAD PCB Designer, for example, the presence of a separate file for each schematic sheet, a single project file, etc.

As you know, and as already mentioned above, OrCAD is a set of different applications. This is also clearly visible in the file structure. In the Project Manager panel, only the schematic part of the project is displayed. First of all, this is a schematic file (*.dsn), which includes all sheets and the entire hierarchical structure, component libraries (*.olb) and a number of other files that are displayed on the Project Manager panel of OrCAD Capture (Fig. 3.1).
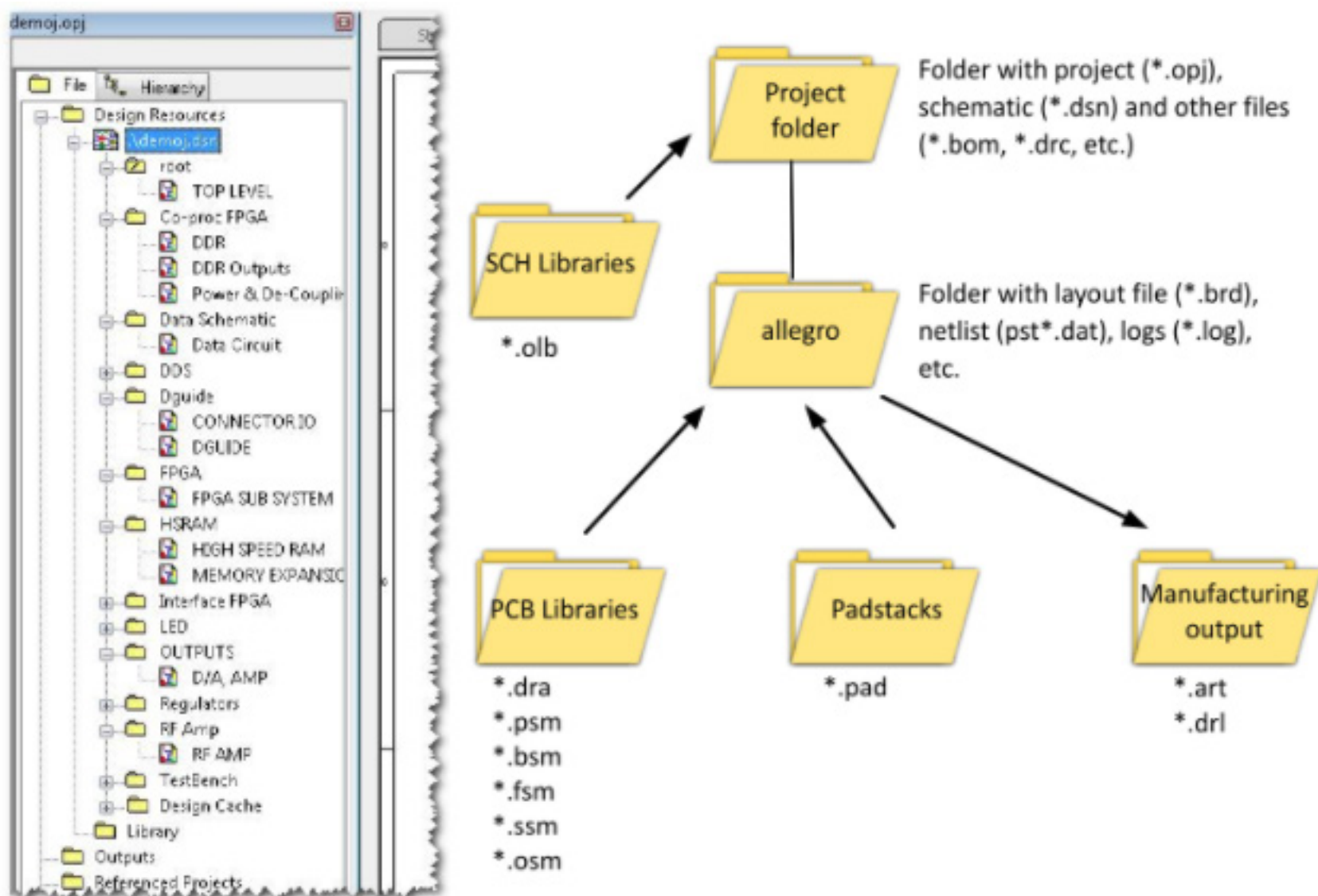
Fig. 3.1 - Project structure in Project Manager panel of OrCAD Capture and standard structure of OrCAD PCB Designer project folders

The Project Manager panel does not show data of Orcad PCB Editor layout files (*.brd) and various libraries (*.psm, *.bsm, *.ssm, etc.). As you will see, this is the fundamental difference between the two systems. OrCAD file structure appears more archaic and kind of disparate system because of its development history.

In the simplest case, an Altium Designer project includes three types of files:

- ⊙ *.PrjPcb – main PCB project file
- ⊙ *.SchDoc – schematic sheet file
- ⊙ *.PcbDoc – PCB layout file

By analogy with OrCAD PCB Designer, there is the dedicated panel in Altium Designer for project file management – the Projects panel. However, unlike OrCAD, all files included in a project are displayed here, including library files, manufacturing output files, and even other software files, for example, Excel-based BOM files, etc. The project folder structure is like the one in OrCAD (Fig. 3.2).
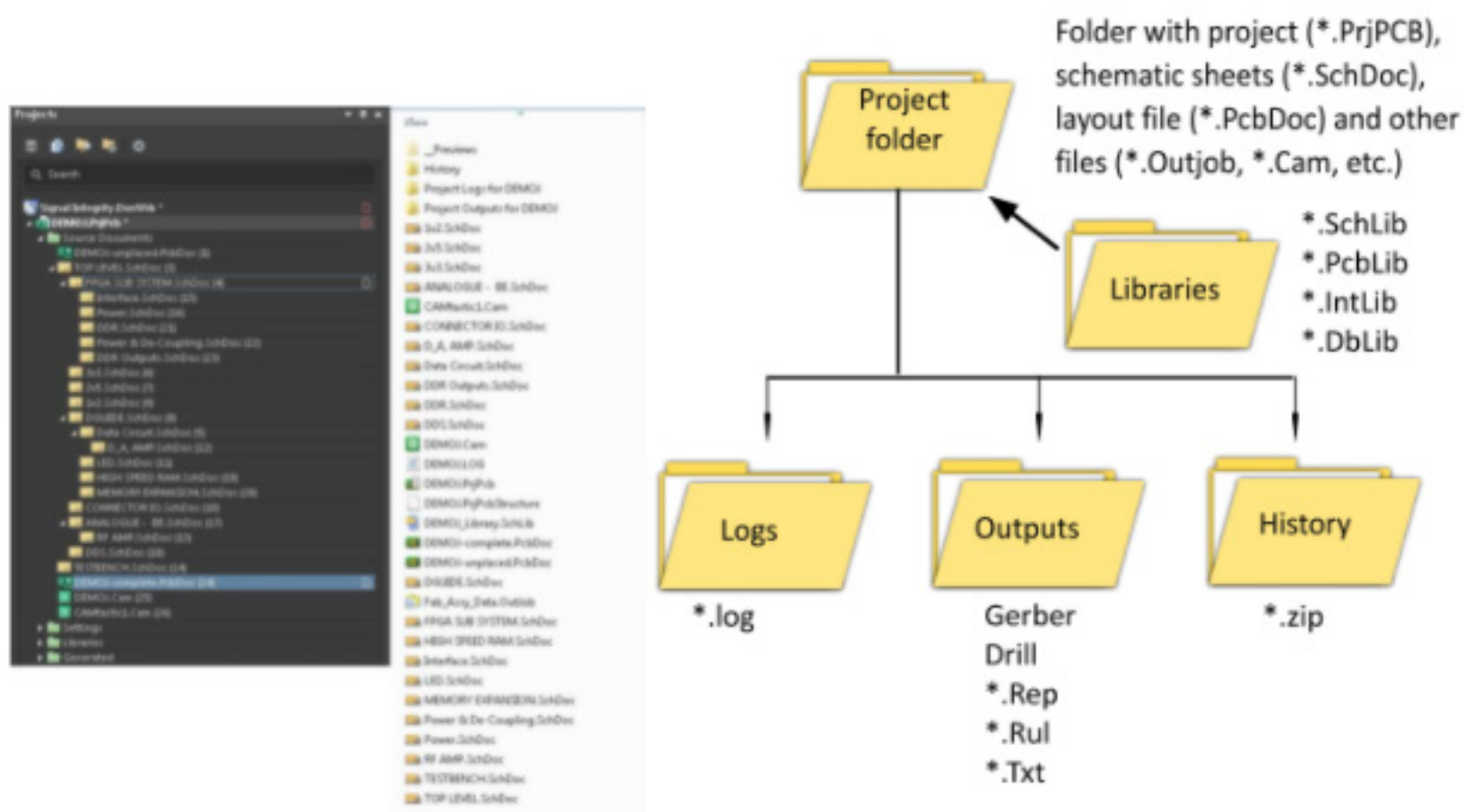
Fig. 3.2 - The Projects panel and the standard structure of Altium Designer project folders

Another fundamental difference is that each schematic sheet in Altium Designer is a separate file. This is intended for file management convenience, tracking change history and the ability to restore each project file separately. You can see that output files folder, log folder and a dedicated History folder, where you can restore saved copies of documents from, are also created in the project folder.

You can drag and drop files between projects in the Project panel. Files located outside the project directory, but included in its structure, are displayed as a shortcut with arrow in its icon.

In Altium Designer, you can create a new project with the *PCB Project dialog*, accessed with the **File » New » Project » PCB Project** command of the main menu, and it will displayed in the Project panel as an empty *.PrjPcb file.

Design of a new PCB begins with creation of a new project file (*.PrjPcb). You should know the following information about the PCB design in Altium Designer:

1. Project file *.PrjPcb includes only settings and links to all source documents – schematic sheets (*.SchDoc), layout file (*.PcbDoc), component libraries (*.SchLib), etc.
2. Each schematic sheet is a separate *.SchDoc file.
3. All source files of a project should be stored in the folder the project file is located. This will simplify all further work.
4. The *Project* panel shows status of each document and the project itself: ▣ – modified, ▢ – opened.
5. To close a project, you can right-click on the project file and choose the **Close Project** command from the context menu. In this menu, you can also find the command for closing all documents.
6. To remove a document from project, you can drag it to a blank area of the Project panel or right-click on it and choose the **Remove from Project** command from the context menu.
7. Any document that is not included in a project displayed under the Free Documents folder.

You can add new files to a project with the **Add New To Project** command from the context menu (right-click on the project file), as shown on Fig. 3.3.
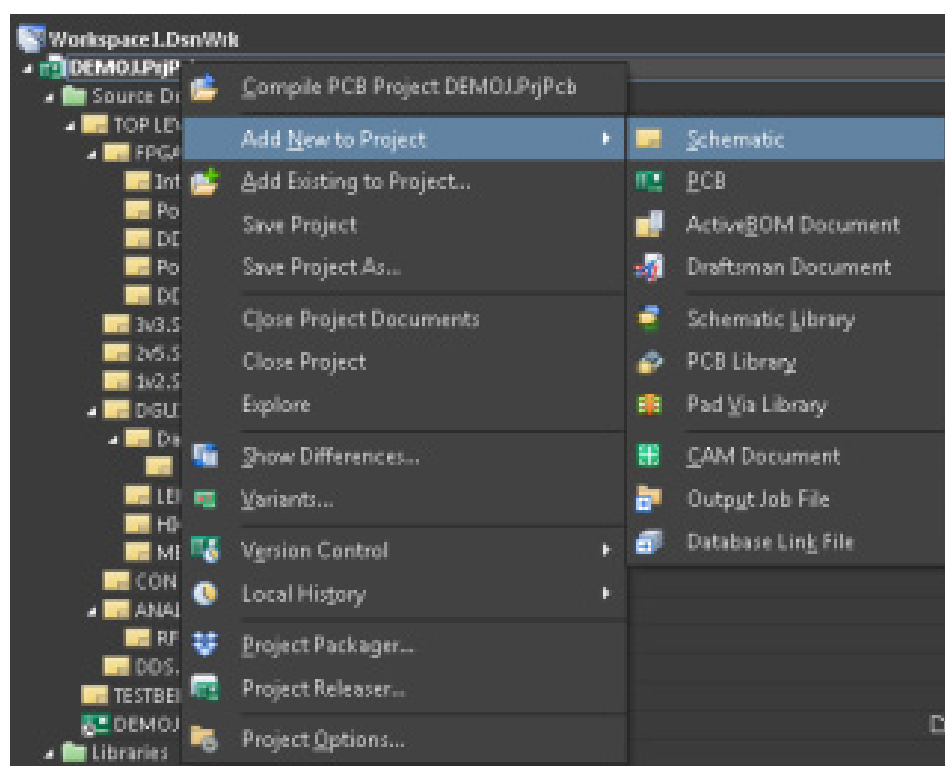
Fig. 3.3 - Creating a new file in PCB project

After creating a new file save it in the project folder, as mentioned above.

Note that if you cannot quickly find a desired document, component, net, etc. you can use the **Search** string in the Projects panel. This will speed up your work significantly.

4. Design environment configuration

This section covers basic configuration Altium Designer's SCH Editor, including current document settings and applying schematic templates.

SCH Editor basic configuration

In OrCAD Capture schematic editor, you configure all basic settings before creating a new project in the Design Template dialog, accessed with the Options » Design Template command from the main menu. On its Page Size tab, you can choose units, grid step (which defined by Pin-to-Pin Spacing option) and sheet size (Fig. 4.1).
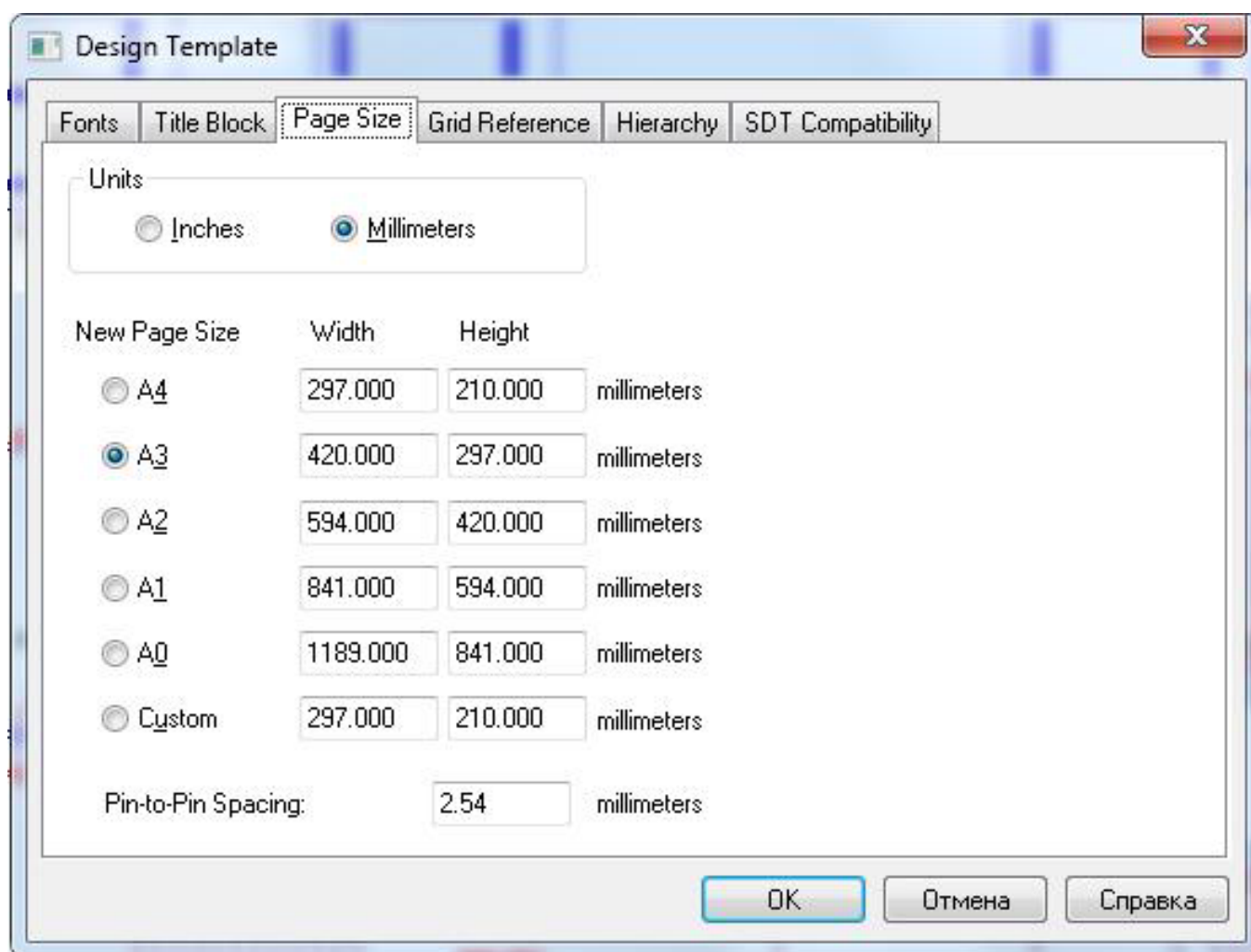
Fig. 4.1 - Unit and grid step settings in OrCAD Capture

Grid style and color are defined in the Preferences dialog (**Options » Preferences**) in appropriate **Grid Display** and **Color/Prints** tabs.

All Altium Designer's editors are configured in the single *Preferences* dialog, which accessed by clicking on the button located in the upper-right corner of the workspace or by right-clicking on the document workspace (Fig. 4.2).
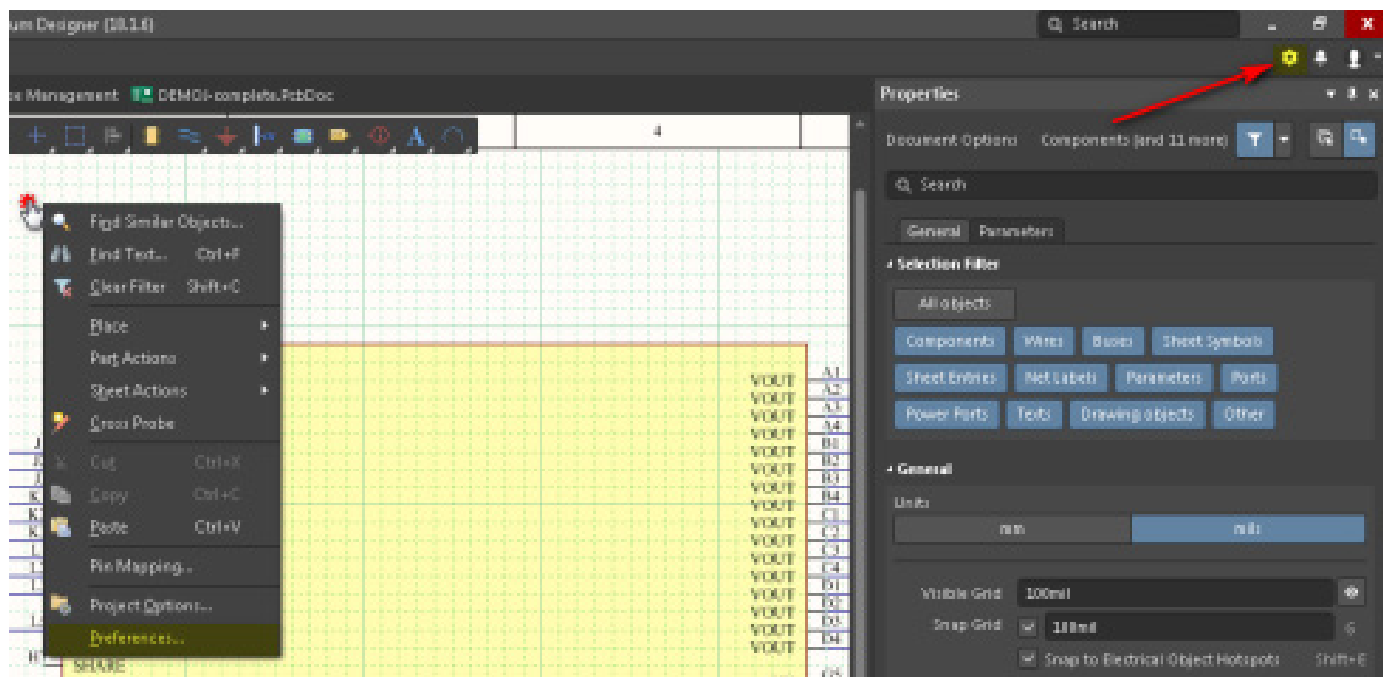
Fig. 4.2 - Access to Altium Designer preferences

In the **Schematic** section you will find all settings for SCH Editor. To set up units go to the **Schematic - General** page (Fig. 4.3).
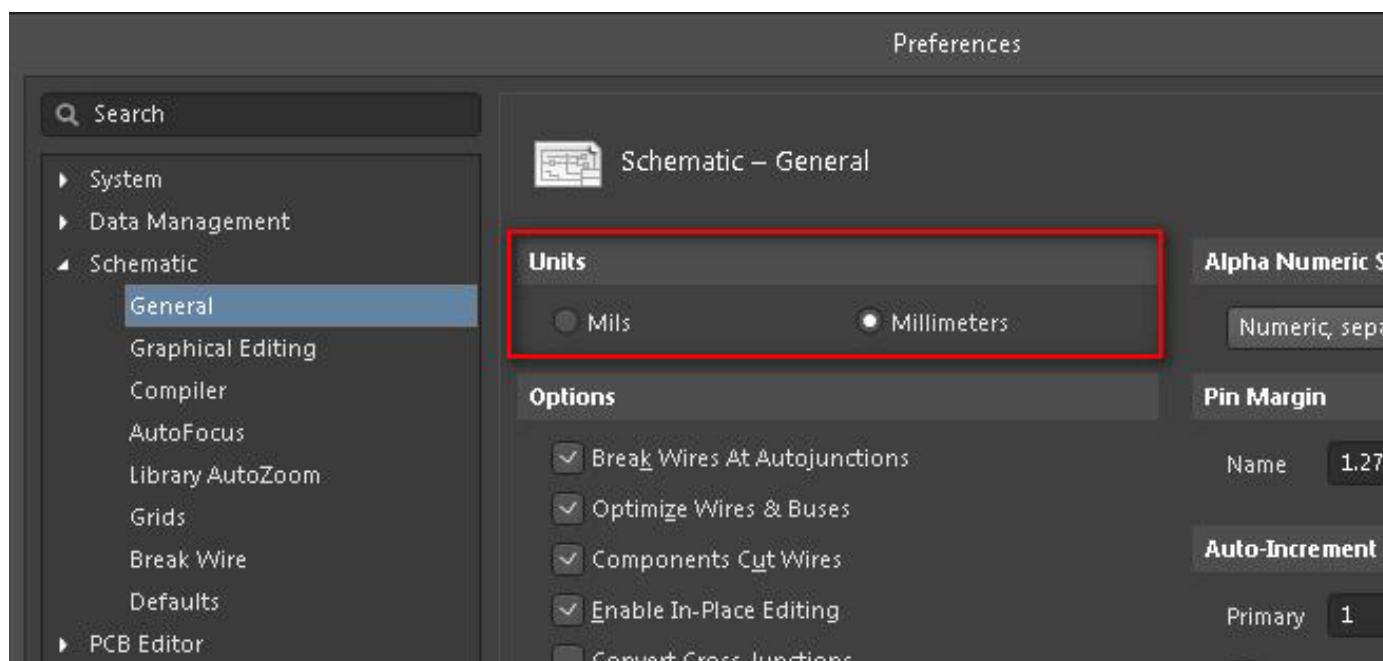
Fig. 4.3 - Configuring units in Altium Designer

To set the grid, go to the **Schematic - Grids** page. Here, unlike OrCAD Capture, you define all options at once – the style, the color and the grid step (Fig. 4.4).
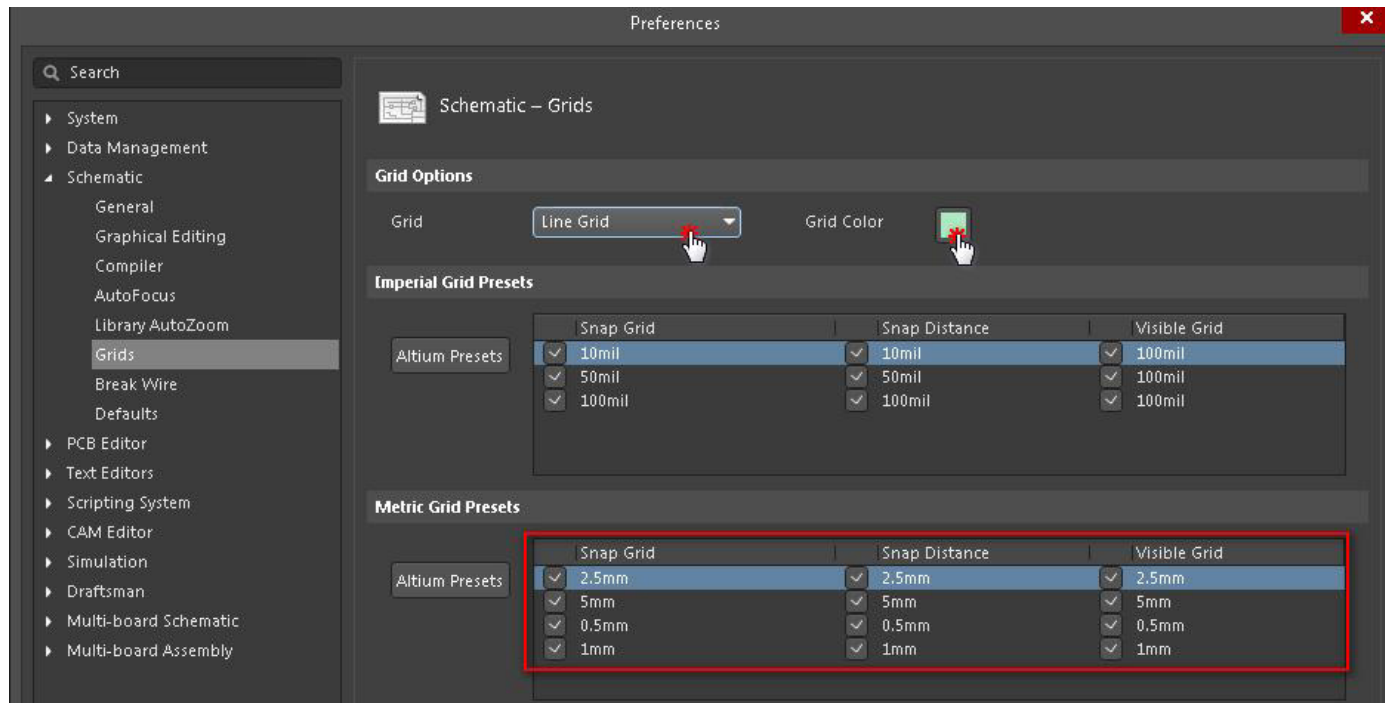
Fig. 4.4 - Configuring grids in Altium Designer

As you can see, a grid has three parameters – **Snap Grid, Snap Distance** and **Visible Grid**. It is recommended to set the same values for all three grids in order not to be confused when working in a schematic or a component library. For adding an additional step value simply right-click and choose the **Add GridSetting** command. You can also change the step values manually.

In OrCAD Capture, any schematic object fonts are configured in the **Fonts** tab of the Design Template dialog (**Options » Design Template**). Here you can specify fonts for all objects, such as **Part Reference, Text, Pin Name, Pin Number** and so on (Fig. 4.5).
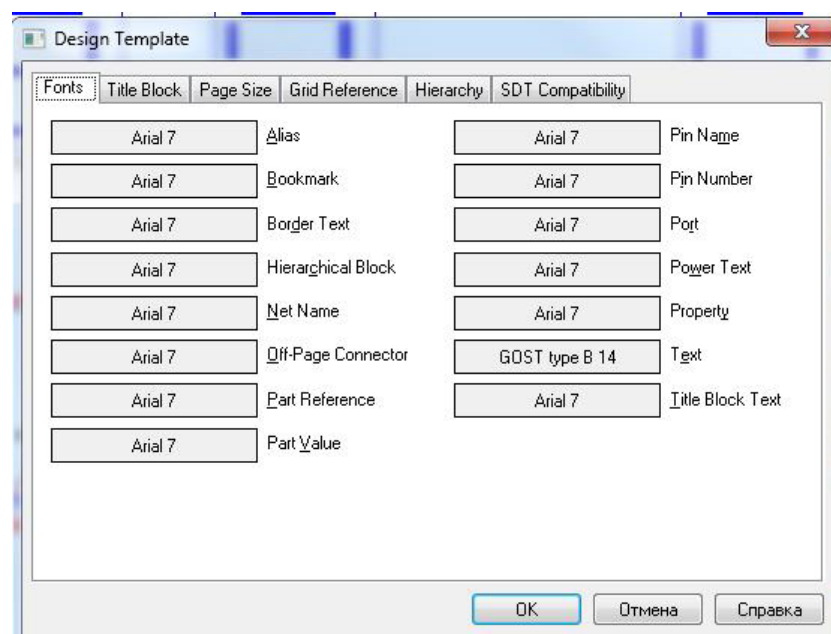
Fig. 4.5 - Font configuring dialog in OrCAD Capture

OrCAD Capture object colors are configured in the **Color/Prints** tab of the *Preferences* dialog.

In Altium Designer, you can go to the **Schematic - Defaults** page of the *Preferences* dialog, select desired units and configure fonts and colors for any schematic object, as it is done in OrCAD. For example, to set the desired font and color for Designator simply select **Designator** object in the primitive list, and its font properties will appear to the right of the list (Fig. 4.6).
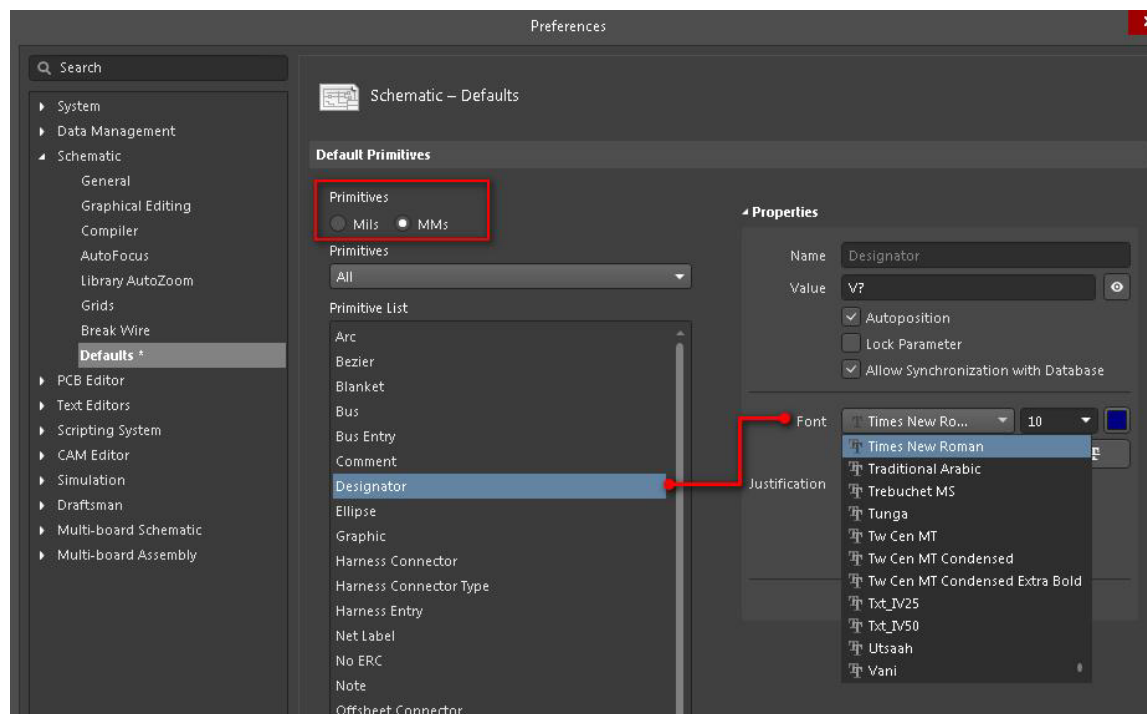
Fig. 4.6 - Configuring object colors and fonts in Altium Designer

Basic configuration of current schematic sheet

In OrCAD Capture, you can define current sheet properties, such as sheet size and units, in the *Schematic Page Properties* dialog (**Options » Schematic Page Properties**). These settings are differ from those discussed above only in that it is not possible to change main grid step for an already created project. You can specify a grid step that is a multiple of the current one with the **Grid Spacing** option in the **Grid Display** tab of the *Preferences* dialog.

In Altium Designer, you can change current sheet settings via the *Properties* panel in the **Document Options** mode (active when no object is selected in the workspace). Here you can configure units, grids, sheet size, etc. You can also change grid with G shortcut (Fig. 4.7).
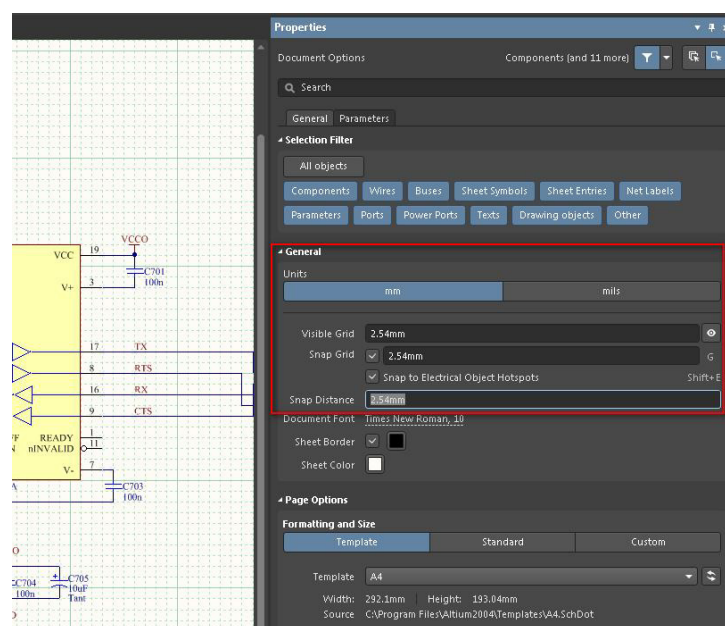
Fig. 4.7 - Basic configuration of current schematic sheet in Altium Designer

Configuring schematic sheet templates

Before starting work in OrCAD Capture you can set the sheet size and its default template. To fill the title block use settings in the Title Block tab of the *Design Template* dialog. Here you can specify basic text options, such as document author, document number, and so on. Changing title block position occurs automatically when you change the sheet size. If you need to change title block you can use the **Place » Title Block** command from the main menu. In this case, you can select sheet elements from the built-in library capsym.olb or from any of your own libraries, where such title blocks are stored.

In Altium Designer, you can set default template on the **Schematic - General** page of the *Preferences* dialog using the **Template** option (Fig. 4.8).
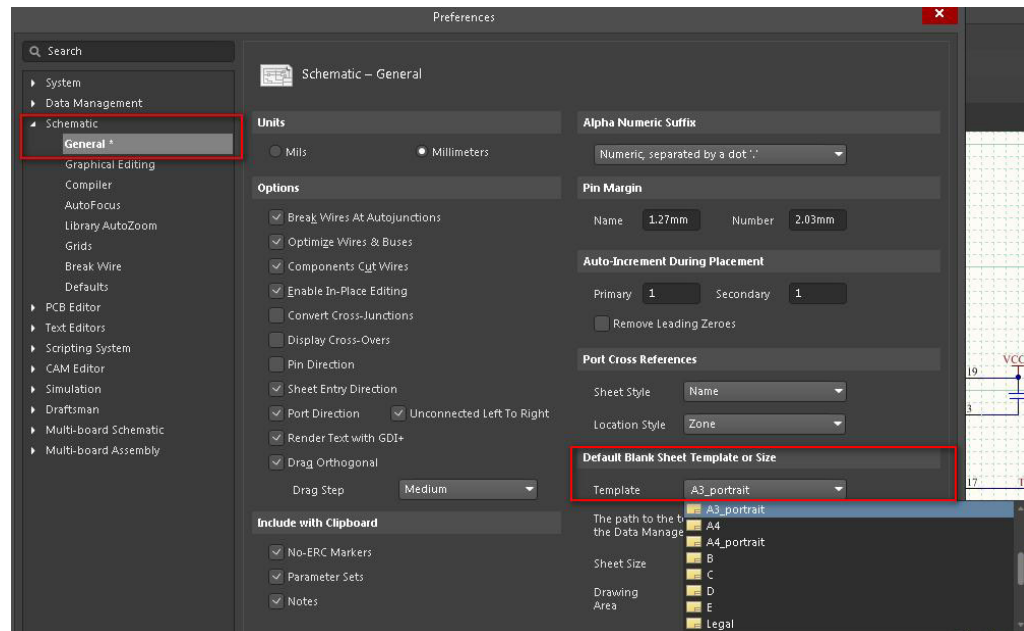
Fig. 4.8 - Select default template for new schematic sheets in Altium Designer

You can change template for a current sheet in the **Page Options** section of the *Properties* panel (Fig. 4.9).
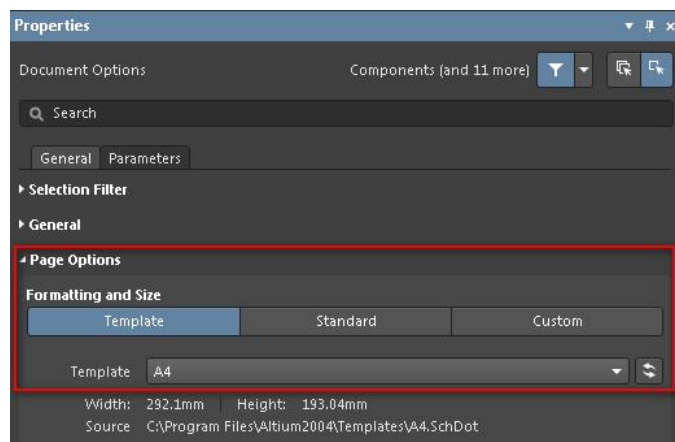
Fig. 4.9 - Changing template for a current sheet in the Properties panel

You can also change template using the **Design » Templates** submenu commands.

Filling the title block is performed on the **Parameters** tab of the *Properties* panel. When a title block is being created, Altium Designer uses document parameters that can has any value. You can change these values in the *Parameters* tab.

5. Understanding libraries

This section includes description of Altium Designer libraries idea and their comparison with OrCAD libraries.

Offline libraries

OrCAD includes a large number of basic libraries that are bundled with the software and can be used for both layout design and circuit simulation with PSpice. Schematic libraries have *.olb extension, e.g. discrete device library Discrete. olb, microcontroller library Microcontroller.olb, operational amplifiers library OPAmp.olb, and so on. Spice models libraries are also quite extensive and have *.lib extension, e.g. bipolar transistor library bjn.lib, Schottky diode library chottky.lib and so on (Fig. 5.1).
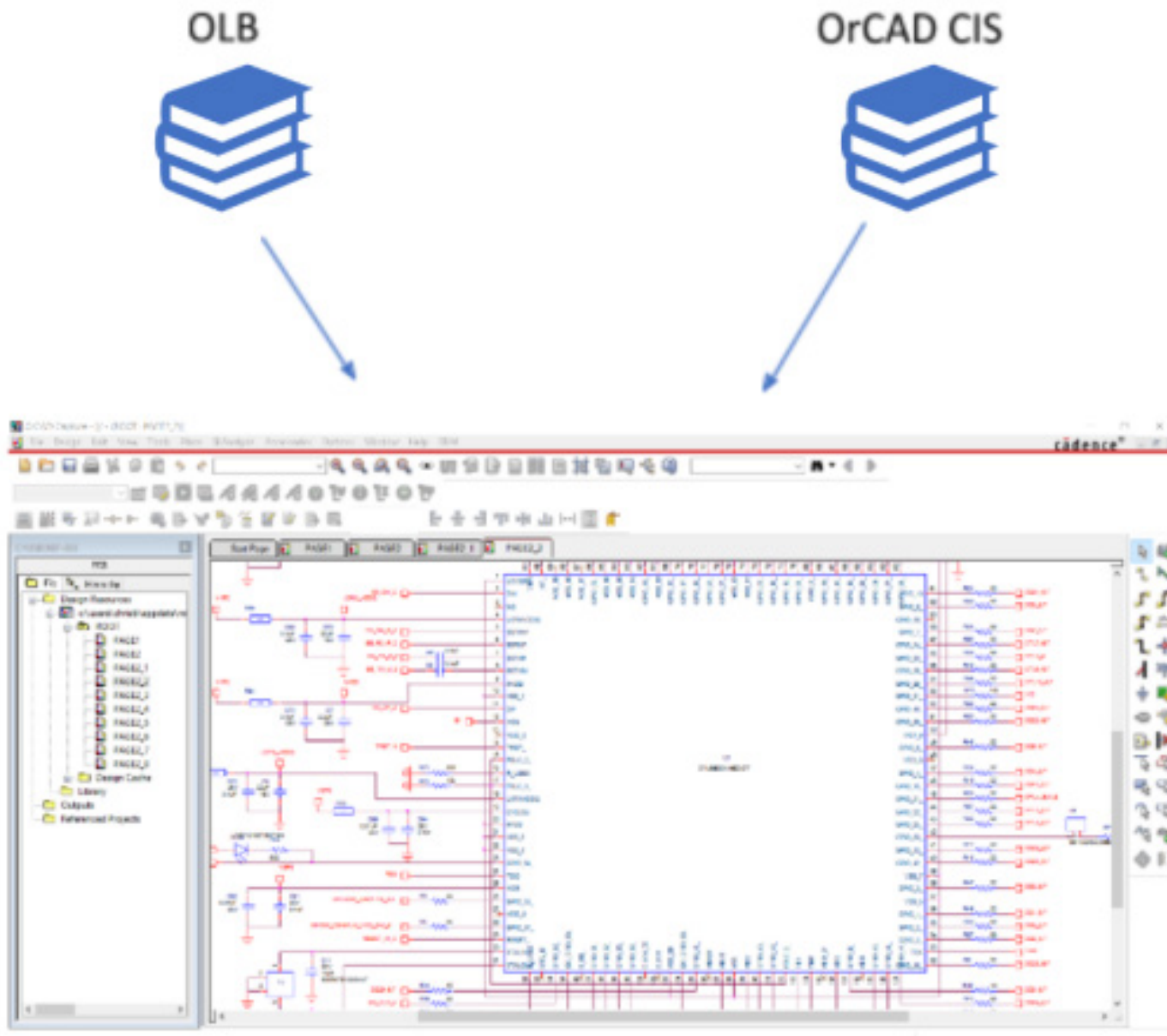
Fig. 5.1 OrCAD Capture libraries

Footprint libraries bundled with OrCAD are represented as *.psm files and complementary *.dra files, where each *.psm and *.dra pair is an individual footprint. Padstack libraries are represented as *.pad files. For a custom pad shape you can create a symbol with *.ssm extension and integrate it into a pad (Fig. 5.2).
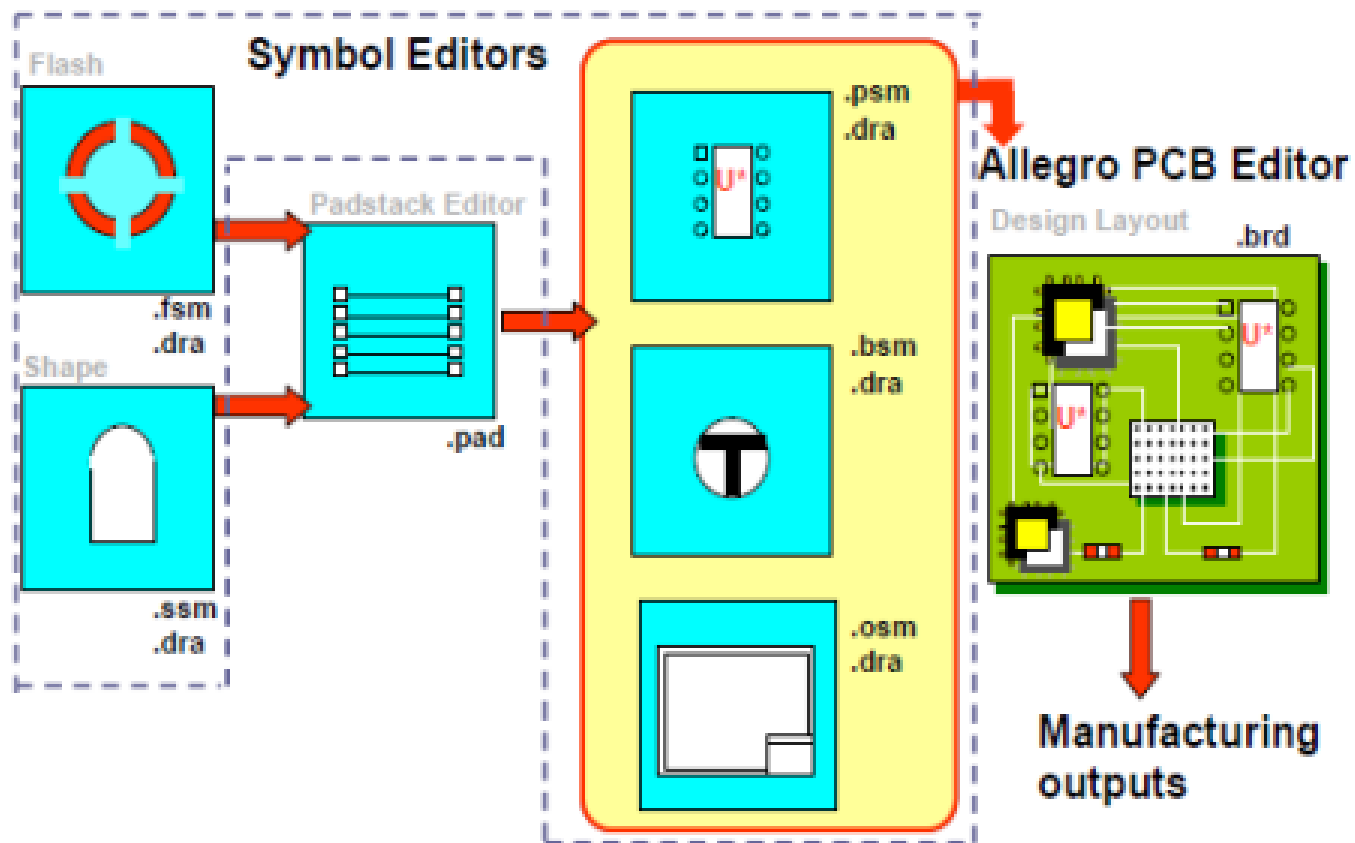
Fig. 5.2 - PCB Editor library structure

There are no integrated libraries in OrCAD. Linking schematic symbols and footprint is performed at component properties level. You can assign a footprint by simple specifying its name in the **PCB Footprint** property at library or schematic level.

You can also download OrCAD libraries from manufacturer websites. There is no a dedicated resource with a ready-made OrCAD component database.

As you can see on Fig. 5.2 you also can use OrCAD CIS for parametric search, verification and component placement using ODBC driver, that allows you to integrate OrCAD to a company workflow system.
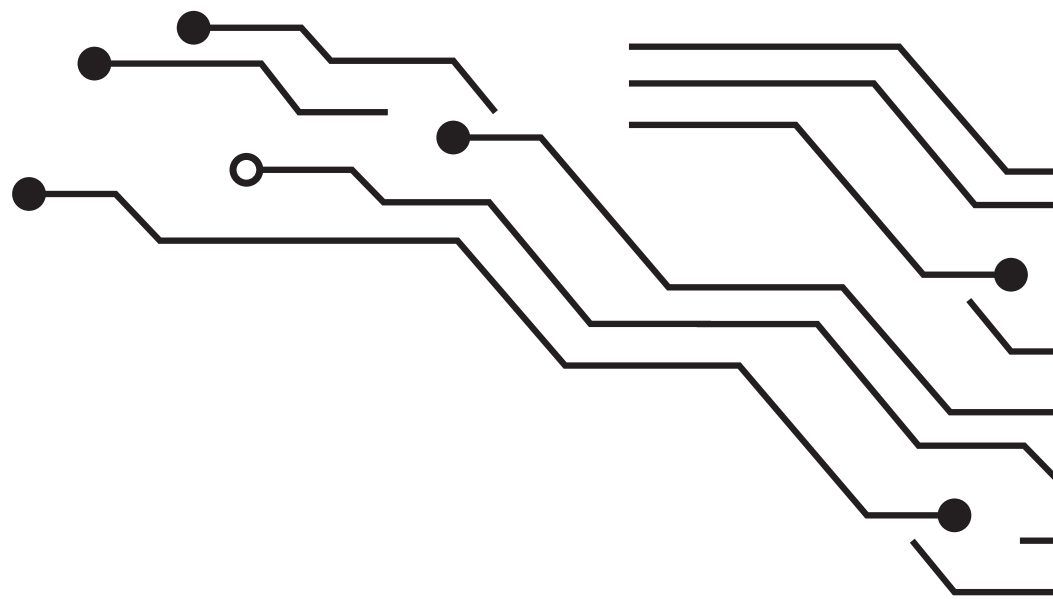
Altium Designer offers several types of file-based libraries. Standard *.SchLib component libraries have the same meaning as *.olb libraries in OrCAD. You can use them when creating a schematic. There is a fundamental difference in footprint libraries, which have *.PcbLib extension. A single *.pcblib file may contain many footprints. You can combine *.SchLib and *.PcbLib into a single integrated library (*.IntLib). In addition, you can use CIS system as in OrCAD that is called database library (*.DbLib) in Altium Designer. Such libraries can be built with Excel, Access, SQL description file or other spreadsheet/database system. Database connection in Altium Designer is performed with standard ODBC interface. You can also use *.CmpLib libraries intended for using in Altium Vault (Fig. 5.3).
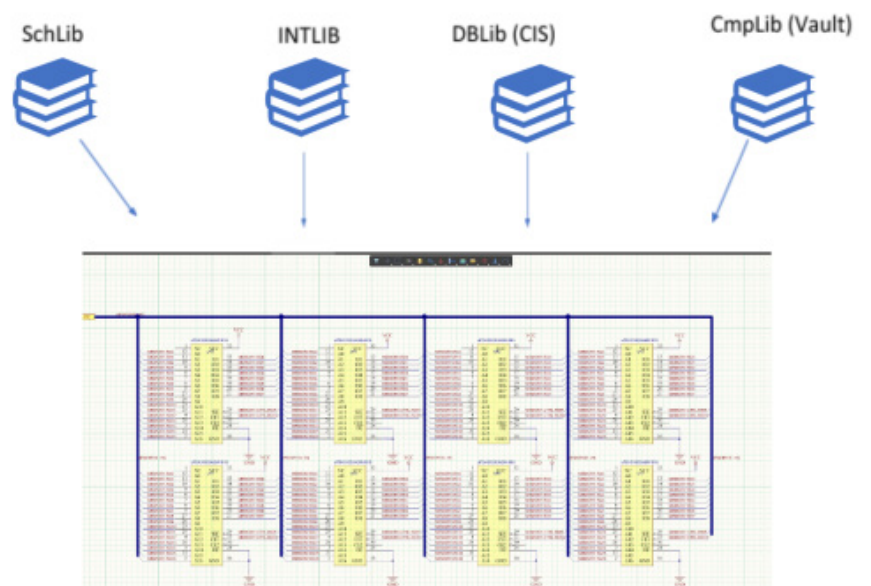
Fig. 5.3 - Altium Designer libraries

Online libraries

OrCAD PCB Designer does not contain an online library as such, which can be connected via a panel. You can download ready-made libraries from component manufacturer websites or with the purchasable Ultra Librarian extension.

In addition to file-based libraries Altium offers access to the global Altium Content Vault library. This resource is available for all licensed Altium users and contains a large number of ready-made component libraries with parametric data for quick search and document creation. The library is accessed directly from the Altium Designer's SCH Editor via the Explorer panel (Fig. 5.4).
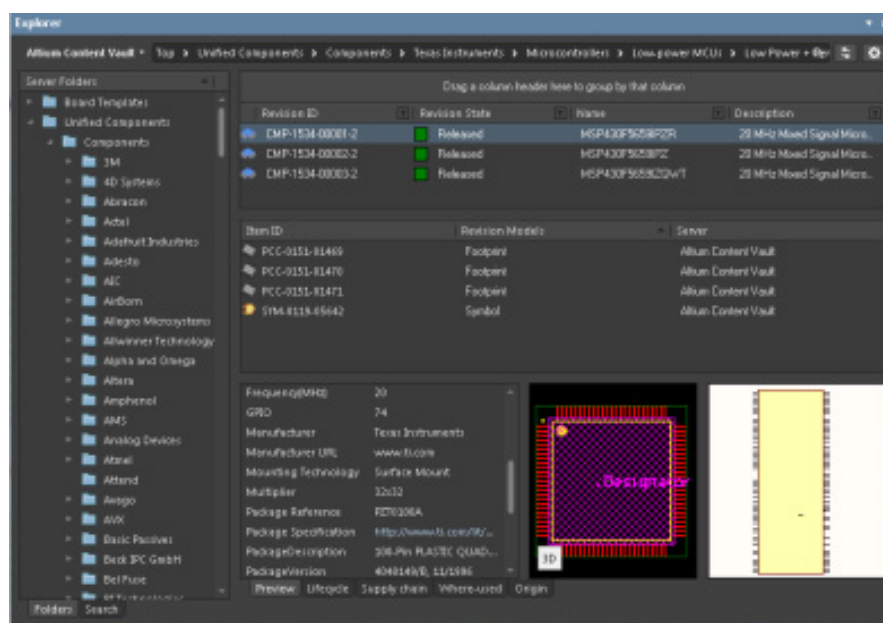
Fig. 5.4 - Global Altium Content Vault library in the Explorer panel

You can generate your own libraries based on Vault components and use them together with the standard SchLib, IntLib, DbLib libraries.

6. Component search and placing in design

This section covers searching and placing components on an Altium Designer schematic sheet and comparison this process with OrCAD Capture.

Libraries panel and component search

There are several ways of placing components on a schematic in OrCAD Capture. The simplest way is using the Place Part panel (accessed with P shortcut). Here you can connect desired library and select a component for placing it on a schematic sheet (Fig. 6.1).
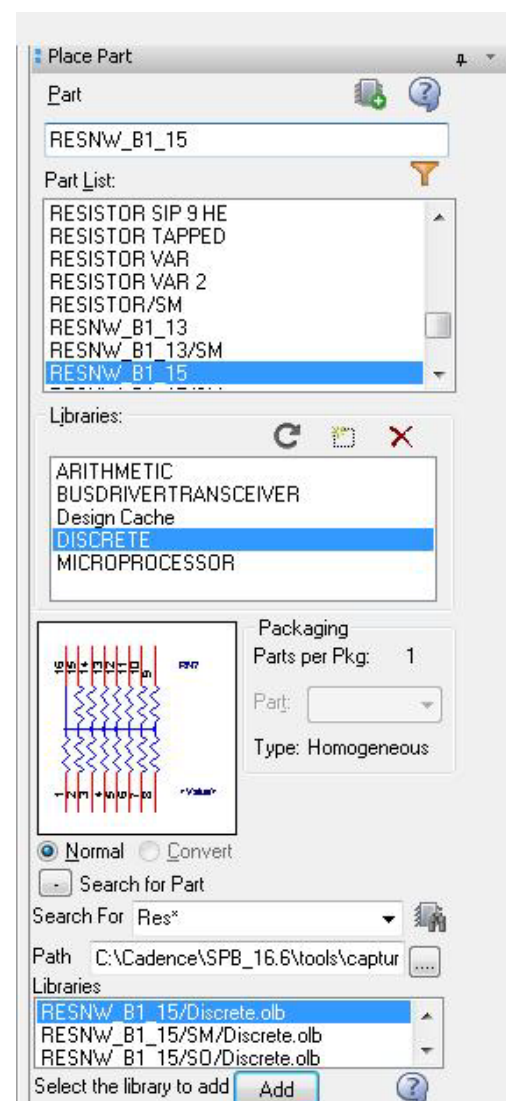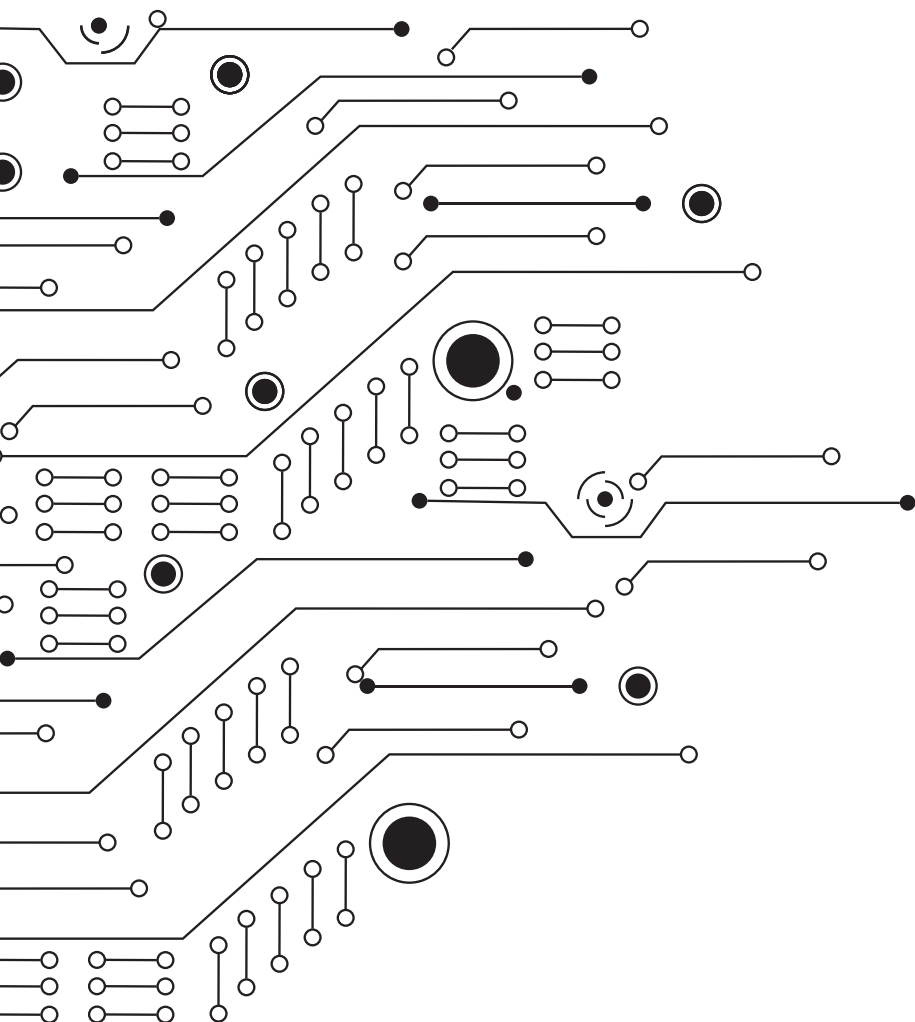
Fig. 6.1 - The Place Part panel in OrCAD Capture

You can use the **Search For Part** section of the panel for searching a component. The search criteria can only be the library component name.

The main panel for searching and placing components on a schematic sheet in Altium Designer is *Libraries* panel. This panel is minimized by default in the right part of the editor window. If it is not opened activate it with the **Panels** button at the bottom right of application window. Use the **Libraries** button at the top of the panel for installing a component library, including database library, Vault library, and others. Use the **Search** button to search components in specified paths based on user-defined criteria (component name, manufacturer, part number and so on), as shown on Fig. 6.2.

Standard libraries are located in C:\Users\Public\ Documents\Altium\AD18\Library directory. If additional libraries should be connected click o the **Libraries** button. On the **Project** tab, the list of project libraries added to current project is shown. The **Library Path Relative** To field on the **Installed** tab points to common library folder. If libraries from other folders should be installed click on the **Install » Install From file/Vault** button.

Select one of available libraries from the drop-down list at the top of *Libraries* panel and type name mask in the next field in order to speed up searching for existing components in selected library.
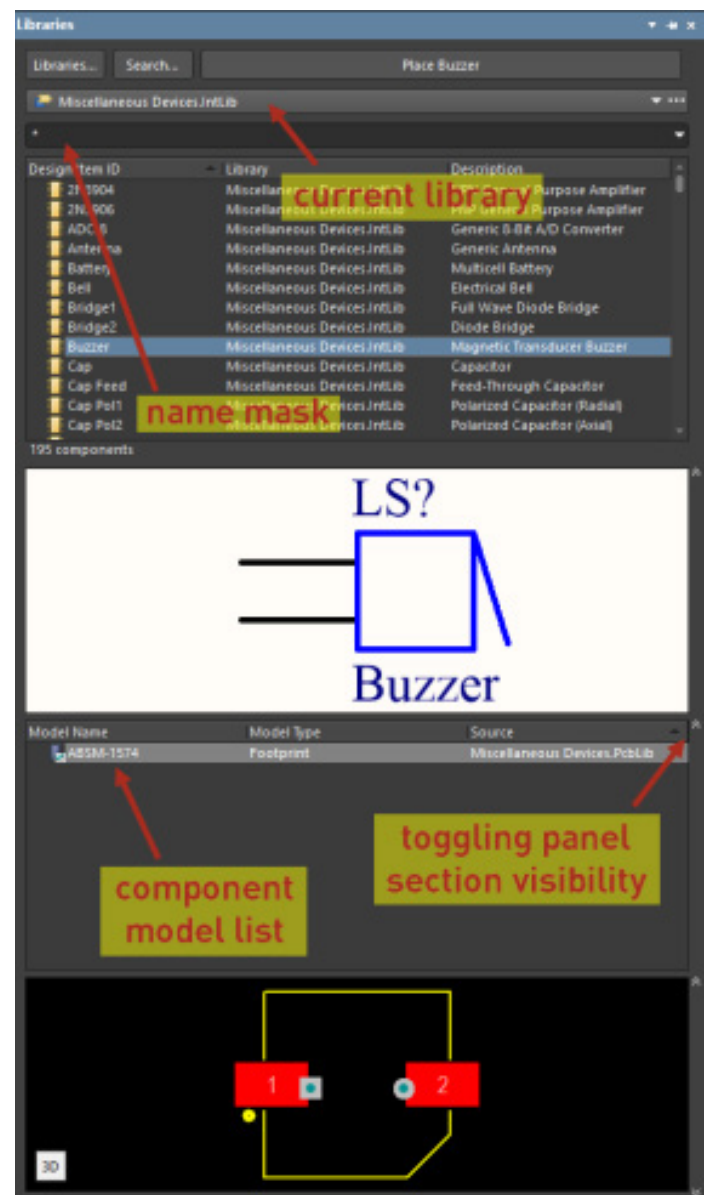
Fig. 6.1 - The Place Part panel in OrCAD Capture

Component placing

Once the desired component is selected in OrCAD Capture's *Place Part* panel double-click on it, and component is placed on schematic sheet. You can use the following shortcuts for standard actions: **R** – rotating, **H/V** – mirror horizontally/vertically. The **Ctrl+E** shortcut opens window for basic component configuration when the component is being placed.

You can place components with auto-numbering enabled. Software will track existing in a design designators and assign next designators for components being placed. You can disable this feature if required. Auto-numbering for all schematic components is also available.

A component is placed on Altium Designer's schematic sheet with the **Place** button at the top of the *Libraries* panel or by drag-and-drop action. In the placement mode you can display available shortcut list with the ~ key. The most frequently shortcuts are: **Spacebar** (rotate component), **X** (mirror horizontally), **Y** (mirror vertically).

Altium Designer does not include a feature for component auto-numbering when placing it on a schematic sheet. You can use powerful auto-numbering features.

Design Cache

As you know, all the components that are placed on OrCAD's schematic are stored in the project "directory" called Design Cache. With the Design Cache you can reuse components within a project, replace a component and update it. You can also create a new *.olb library and save all components from the Design Cache to it (Fig. 6.3).
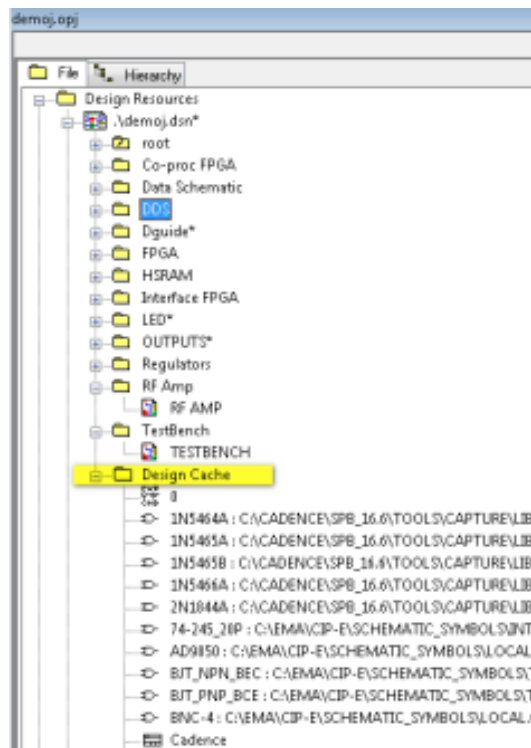
Fig. 6.3 - Design Cache in the Project Explorer panel in OrCAD Capture

There is no Design Cache in Altium Designer, however components are stored in the schematic sheet memory as well. You can also create a library from a project – use the Design » Make Schematic Library/Make Integrated Library command. There is a dedicated command for updating and replacing components – Tools » Update From Libraries (Fig. 6.4).
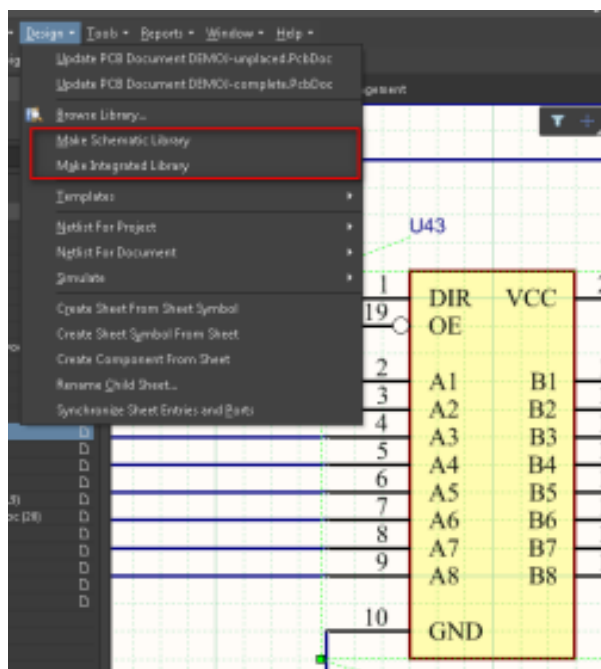
Fig. 6.4 - Commands for creating component library from project

7. Creating connectivity

This section covers description of electrical connectivity objects, net properties and net identifier scope in Altium Designer.

Connection types

OrCAD Capture allows you to create both an individual net (Wire) and a set of nets (BUS, Net Group) that are shown on Fig. 7.1. These objects can be accessed from the Place menu or by using the W, B, U shortcut keys, respectively.
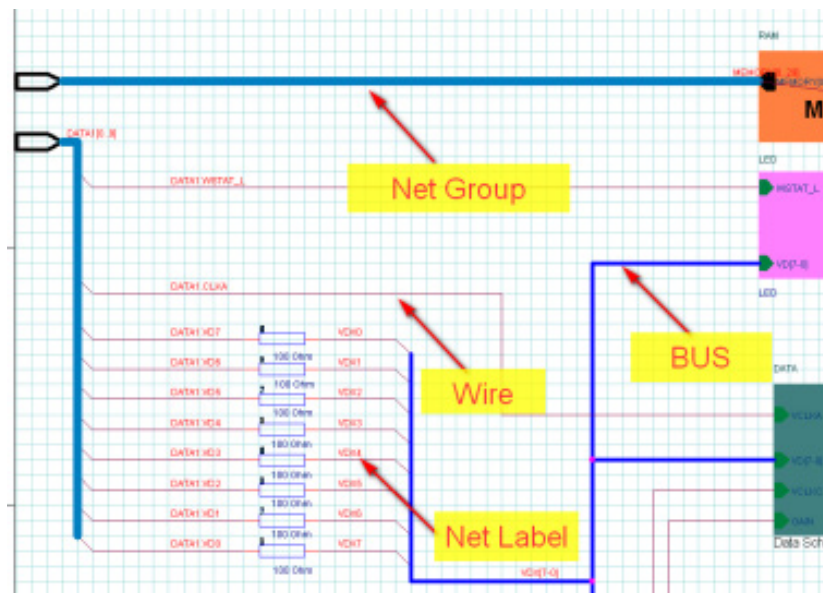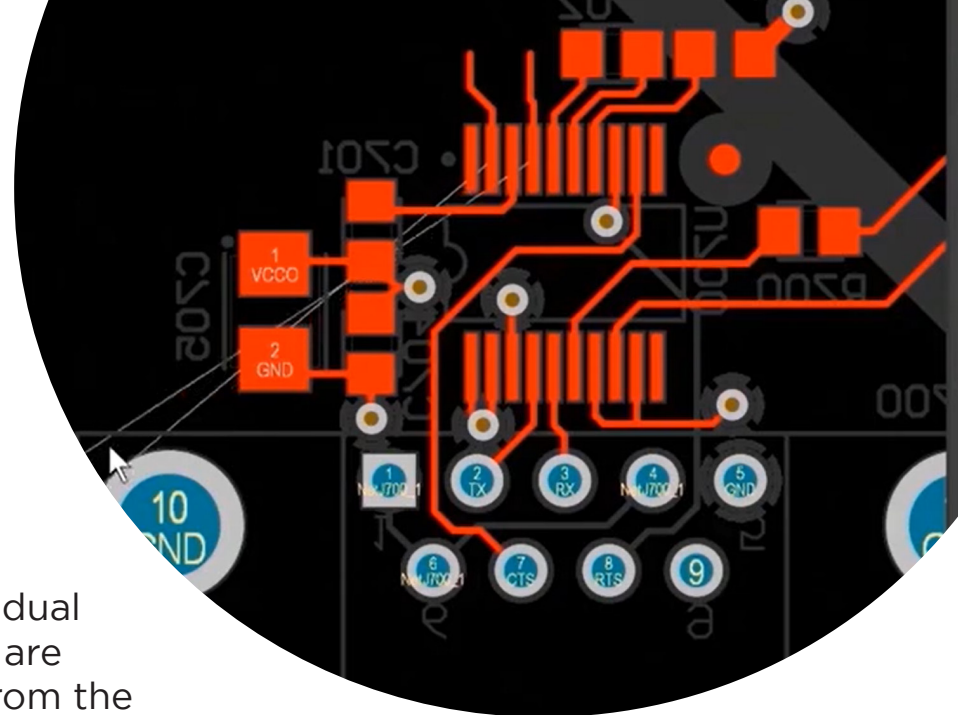
Fig. 7.1 - Connection types in OrCAD Capture

Altium Designer allows you to create similar objects in various ways, including **Active Bar** and shortcut keys (Fig. 7.2).
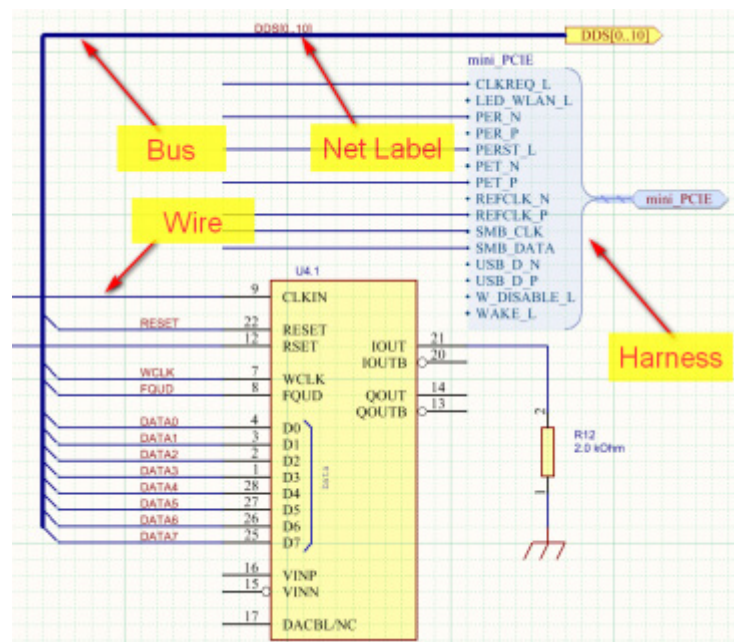
Fig. 7.2 - Connection types in Altium Designer

There is a slight difference in how **Net Groups** in OrCAD and **Harnesses** in Altium Designer are being defined (both are intended to define a set of nets). To create a **Harness**, use commands from the **Place » Harness** menu. You should place **Harness Connector** object first, then place **Harness Entry** objects for input signals. For harness connectivity the dedicated **Signal Harness** object is used (Fig. 7.3).
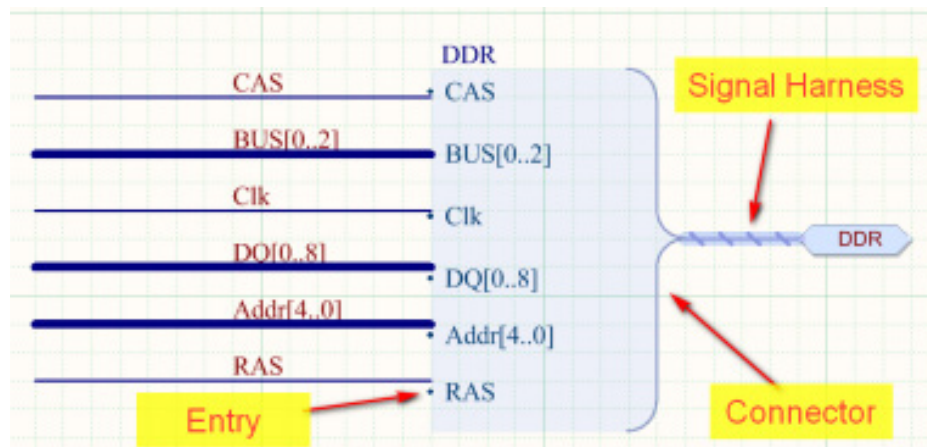
Fig. 7.3 - Harness object in Altium Designer

You can reuse a Harness using the **Place » Harness » Predefined Harness Connector** command. Harness definitions are stored in text files that can be found under the Settings\Harness Definitions Files folder in the *Projects* panel.

Net properties and parameters

In OrCAD Capture you can assign a variety of properties to the nets placed in schematic. For example, you can define the **MIN_LINE_ WIDTH** property in *Property Editor* that defines minimum width for tracks of this net. Any property can be made visible. You can change width and color for nets on schematic (Fig. 7.4).

Fig. 7.4 - Example of net properties in OrCAD Capture

In Altium Designer, you can change only net width and net color on schematic using the Properties panel. There are no additional parameter can be assigned to a net (Fig. 7.5). If a design rule should be specified for a net, you can use the directive objects.
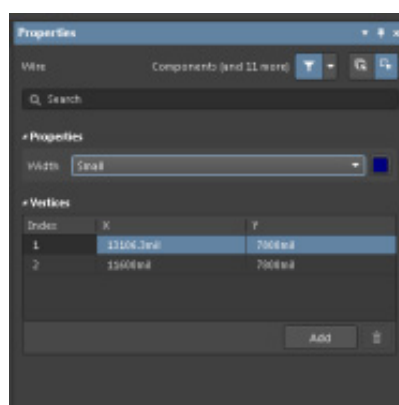
Fig. 7.5 - Example of net properties in Altium Designer

Net identifier scope

In OrCAD Capture, connectivity obeys inter-sheet connection strict hierarchy. You can place the same net label on different sheets using the **Place » Net Label** command but they are connected only if they have Off-Sheet Connector or Port object on their ends. In a hierarchical design, connectivity between parent and child sheets is created through ports only.

In Altium Designer, you can also use off-sheet connectors and ports. However you are able to change net identifier scope with the **Net Identifier Scope** option on the **Options** tab of the *Project Options* dialog (**Project » Project Options**), as shown on Fig. 7.6.
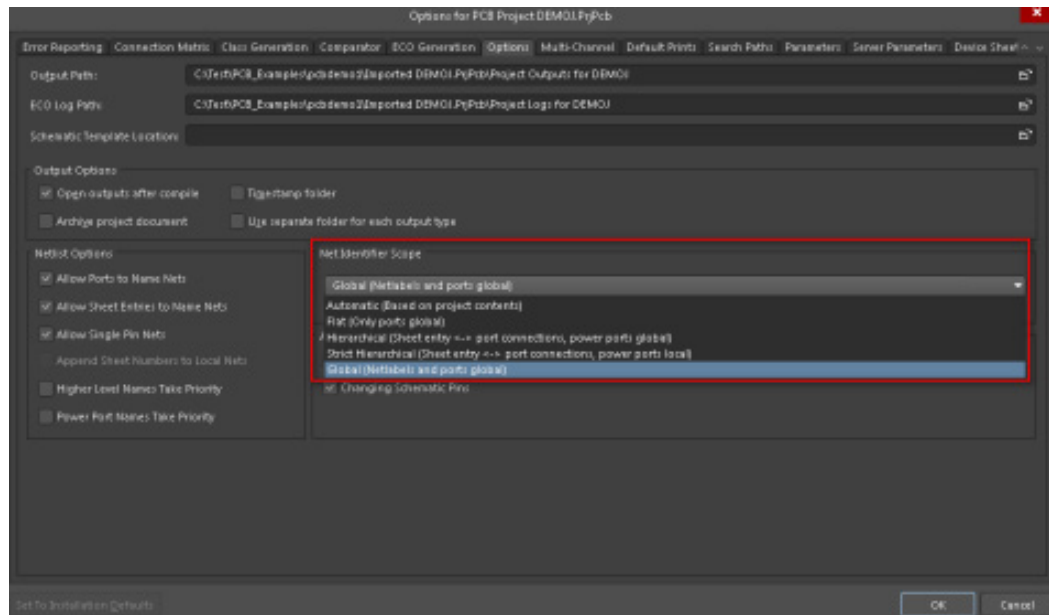
Fig. 7.6 - Change net identifier scope in Altium Designer's project options as required

> **Automatic (Based on project content)** – software automatically selects which of the 3 main net identifier modes to use, based on the following criteria:
>   - if there are sheet entries on the top sheet, then Hierarchical is used;
>   - if there are no sheet entries, but there are ports present, then Flat is used;
>   - if there are no sheet entries and no ports, then Global is used.

> **Flat (Only ports global)** – ports connect globally across all sheets throughout the design, net labels are local to each sheet. This option can be used for flat multi-sheet designs. It is not recommended for large designs as it can be difficult to trace a net through the sheets.

> **Hierarchical (Sheet entry <-> port connections, power ports global)** – connect vertically between a port and the matching sheet entry. This option makes inter-sheet connections only through sheet symbol entries and matching sub-sheet ports. Power ports are global – all power ports with the same name are connected throughout the entire design.

> **Strict Hierarchical (Sheet entry <-> port connections, power ports local)** – this mode of connectivity behaves in the same way as the Hierarchical mode, with the difference being that power ports are kept local to each sheet.

> **Global (Netlabels and ports global)** – with this option, all nets with the same net label will be connected together on all sheets.

8. Hierarchical and Multi-Channel Design

This section covers creation of hierarchical and multi-channel Altium Designer designs. Synchronizing of hierarchical blocks will be discussed as well.

Creating hierarchy

You can build hierarchical design in OrCAD Capture with two different approaches – create a block first, then create a child schematic (top-down fashion), or create a schematic first, then specify it as hierarchical block on the main schematic (bottom-up fashion). A hierarchical block is placed via the **Place » Hierarchical Block** command, then pins are placed on this block via the **Place » Hierarchical Pin** command. Sub-circuit, designator and type should be specified in block properties (Fig. 8.1).
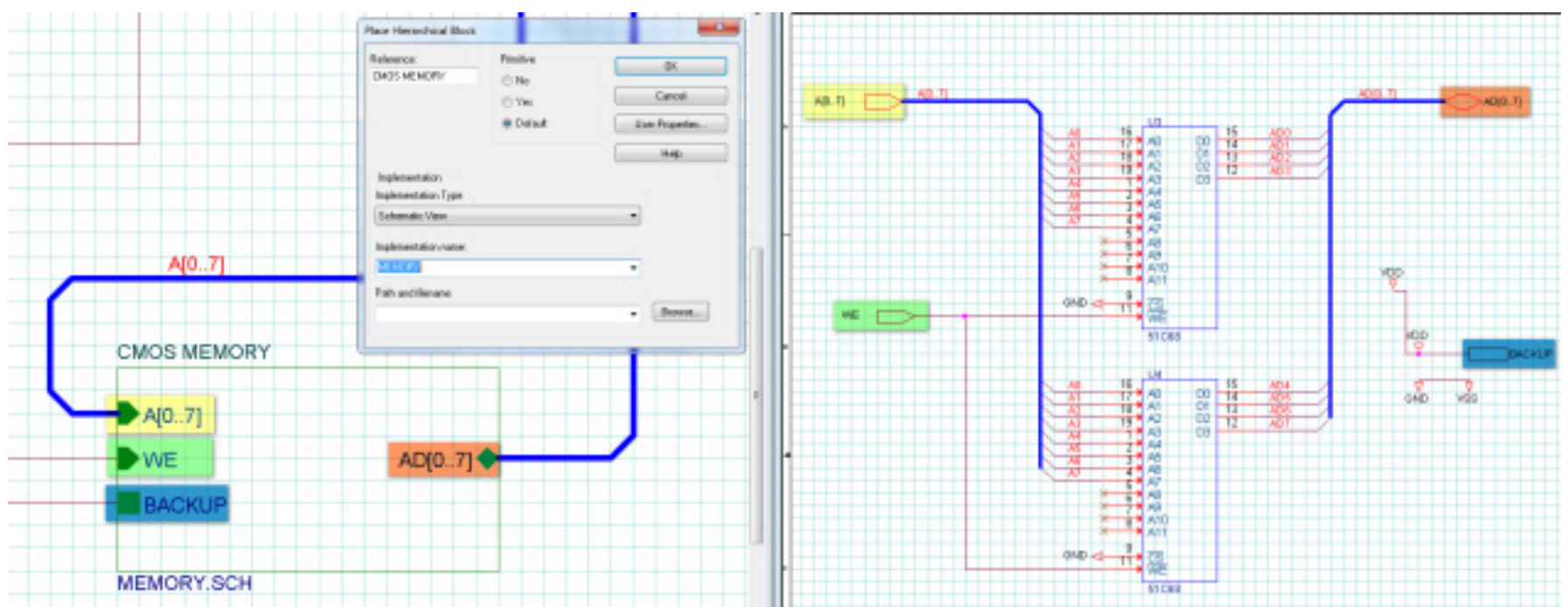
Fig. 8.1 - Hierarchical block and child schematic in OrCAD Capture

Hierarchical block in Altium Designer is called **Sheet Symbol**, and its pin is called **Sheet Entry**. These objects can be placed from the **Place** menu. All properties of a block or a pin can be specified in the *Properties* panel. You can use the same block processing methodology as it is in OrCAD Capture. You can create a schematic first, then a block, and vice versa (Fig. 8.2).
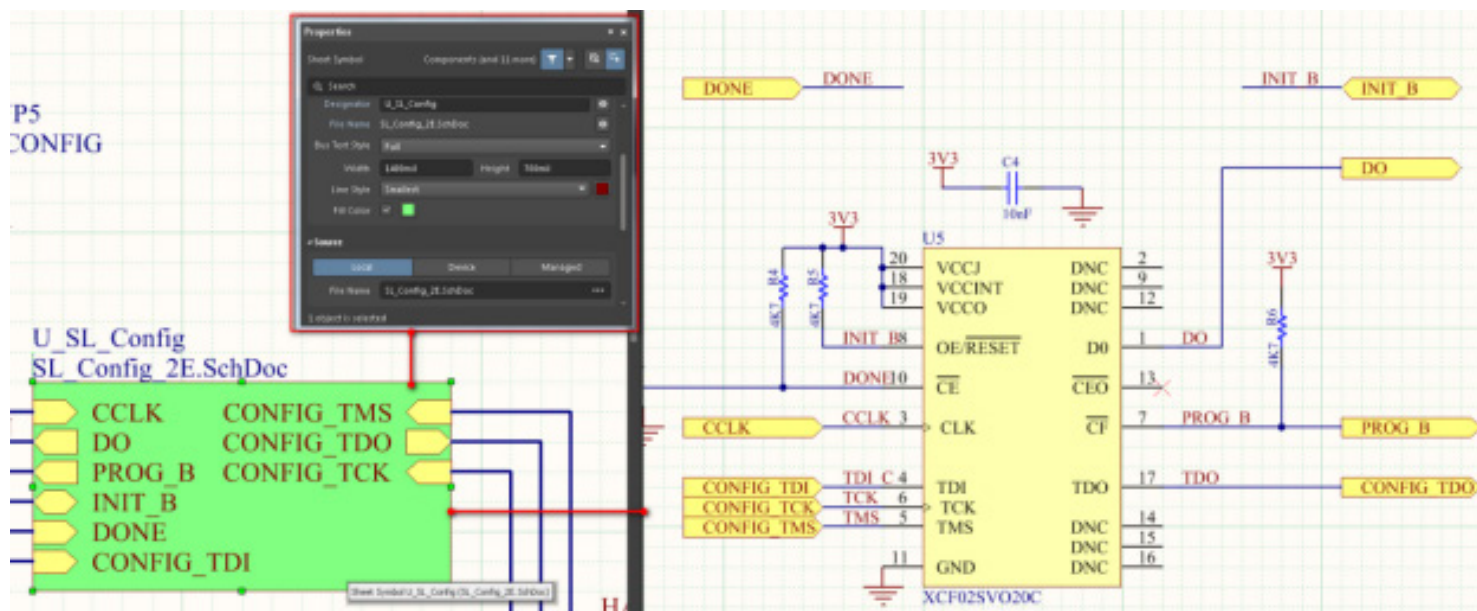
Fig. 8.2 - Hierarchical block and child schematic in Altium Designer

Sheet Symbol related commands are located in the **Design** menu. For example, the **Design » Create Sheet From Sheet Symbol** command is used for translating a block into chematic sheet. The reverse **Design » Create Sheet Symbol From Sheet** command allows you to create the block from a schematic sheet. In this case, you choose the schematic sheet which block will be created from (Fig. 8.3).
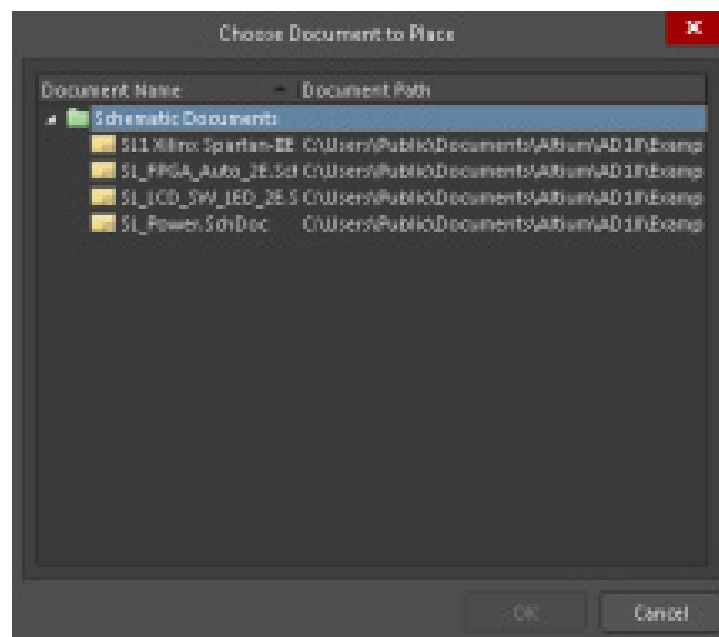
Fig. 8.3 - Choose a schematic sheet for creating block in Altium Designer

Block synchronizing

Block and schematic synchronizing in OrCAD via ports and pins respectively is performed with the **Synchronize Up/Down** from the right-click menu (Fig. 8.4).
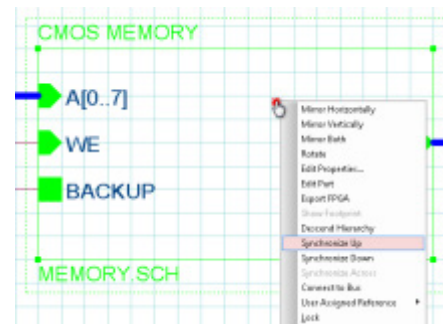
Fig. 8.4 - Schematic and hierarchical block synchronizing in OrCAD

In Altium Designer, you can use the **Design » Synchronize Sheet Entries And Ports** command. Tabs for all sheet symbols are displayed in the *Synchronize Ports To Sheet Entries* dialog, where you can choose pins and ports for synchronizing (Fig. 8.5).
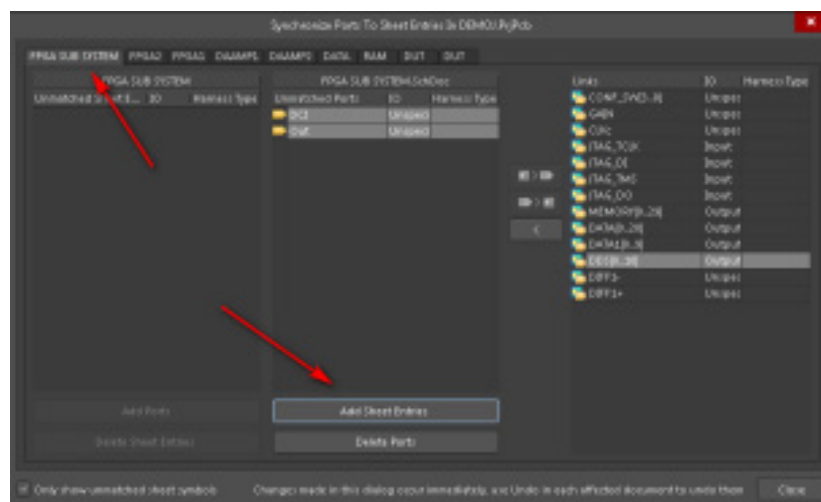
Fig. 8.5 - Sheet Symbol synchronizing in Altium Designer

Multi-channel design

In OrCAD Capture you can create multi-channel design by copying a block on schematic (Fig. 8.6).
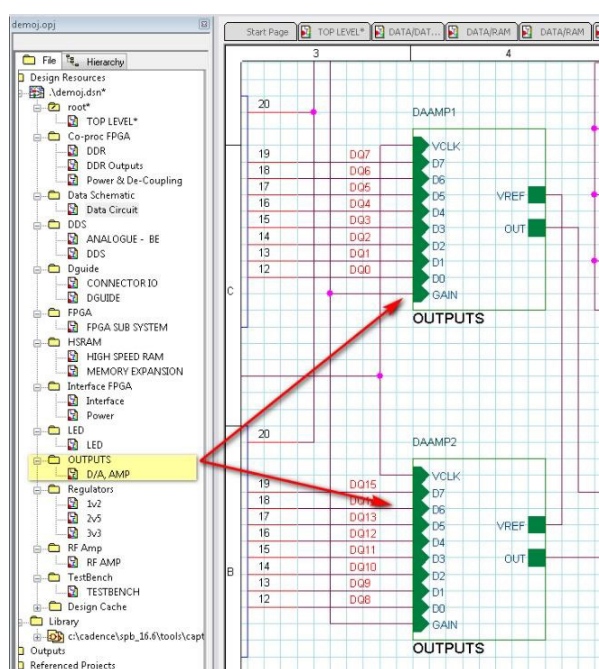
Fig. 8.6 - Multi-channel hierarchical block in OrCAD Capture

Unlike OrCAD Capture, in Altium Designer copying a block multiple times on a schematic to create multi-channel design is optional. The following methodology is applied when working with multi-channel blocks in Altium Designer:

1. Type keyword Repeat and specify channel names and numbers in Reference Designator.

2. As result, each channel is converted to the room on the board. You can place and route one of these room, then get the same layout in other rooms (similar to the **Reuse Module** command).

A multi-channel block with Repeat keyword in Altium Designer is shown on Fig. 8.7.
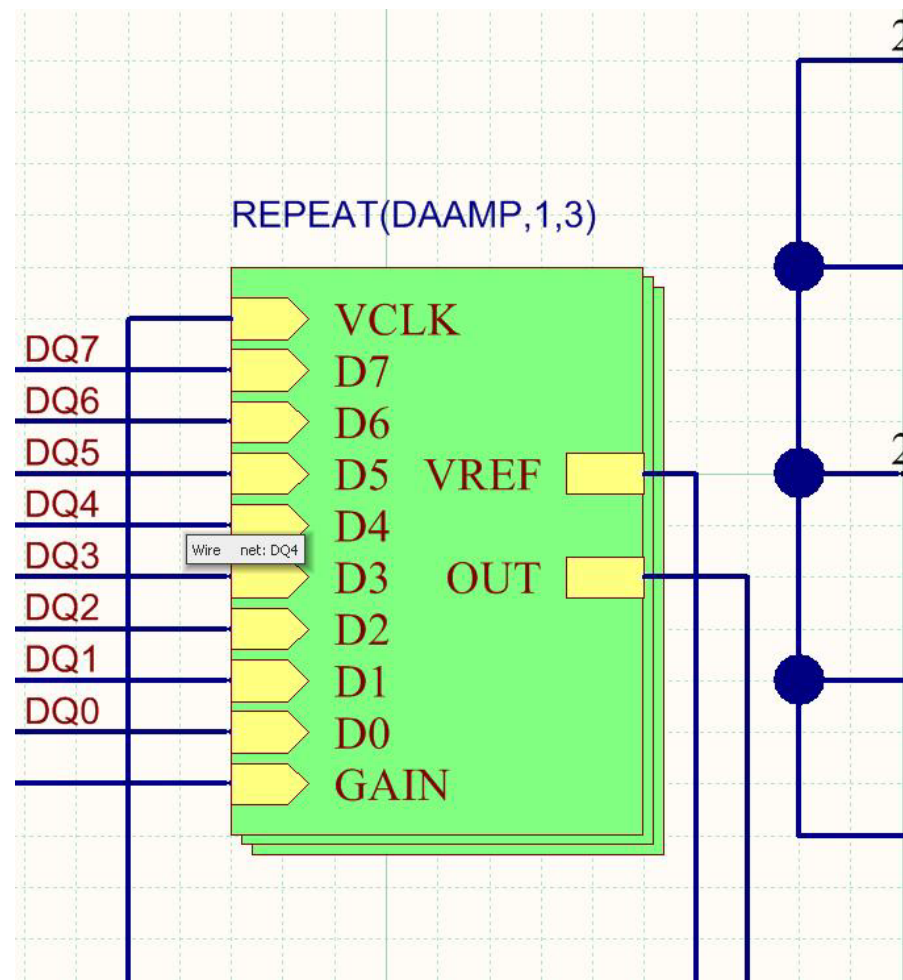
Fig. 8.7 - Multi-channel hierarchical block in Altium Designer
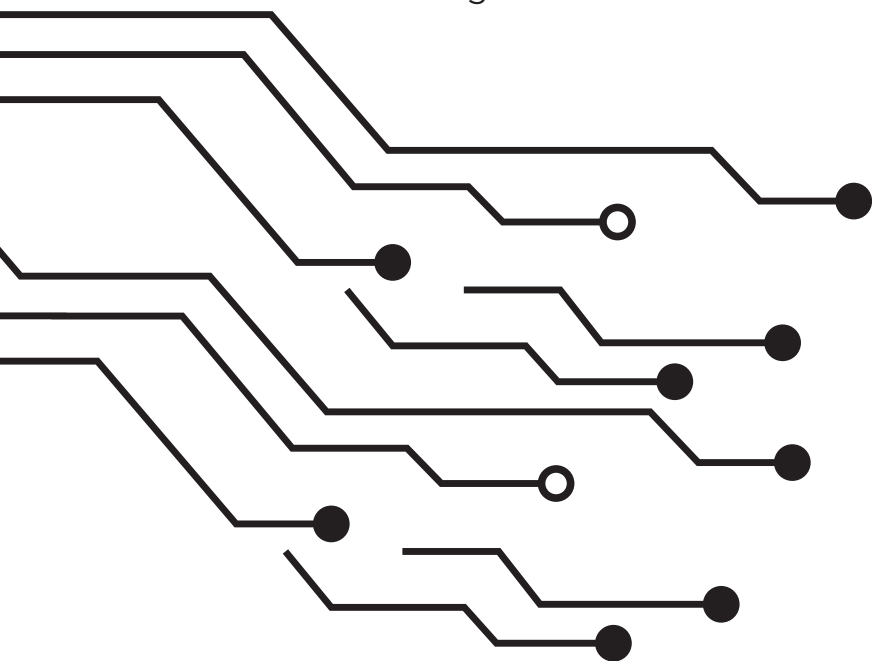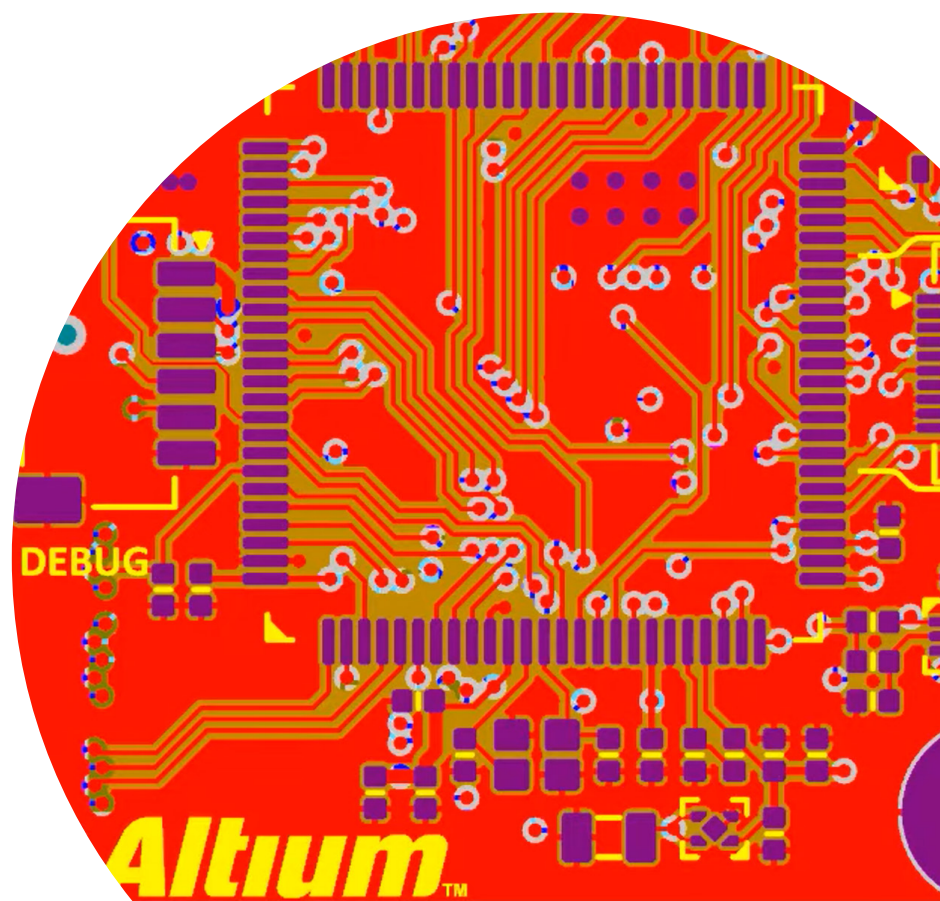
9. Schematic search and navigation

This section covers various options for components and net searching in an Altium Designer's schematic.

Text search

In OrCAD Capture, you can press **Ctrl+F** and specify search criteria. Name of a net or other object is specified in the search field. However you should closely monitor where you are making the search from and whether the corresponding object is selected for search. If you run search on a schematic, then an object will be found within this schematic only. To search the entire project, select it in the Project Manager panel first, and then run search by desired criteria (Fig. 9.1).
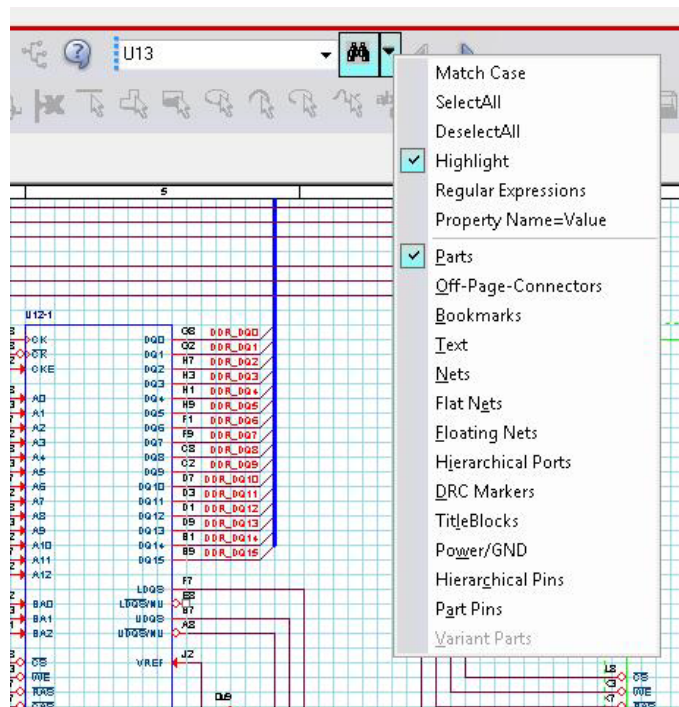
Fig. 9.1 - The Search toolbar in OrCAD Capture

Search results are shown in the Find Window panel (Fig. 9.2).

Fig. 9.2 - The Find Window panel and search results

In Altium Designer, you can also use the **Ctrl+F** shortcut to run standard tool for component and net text-based search. Simply specify search criteria and search scope to find an object (Fig. 9.3).
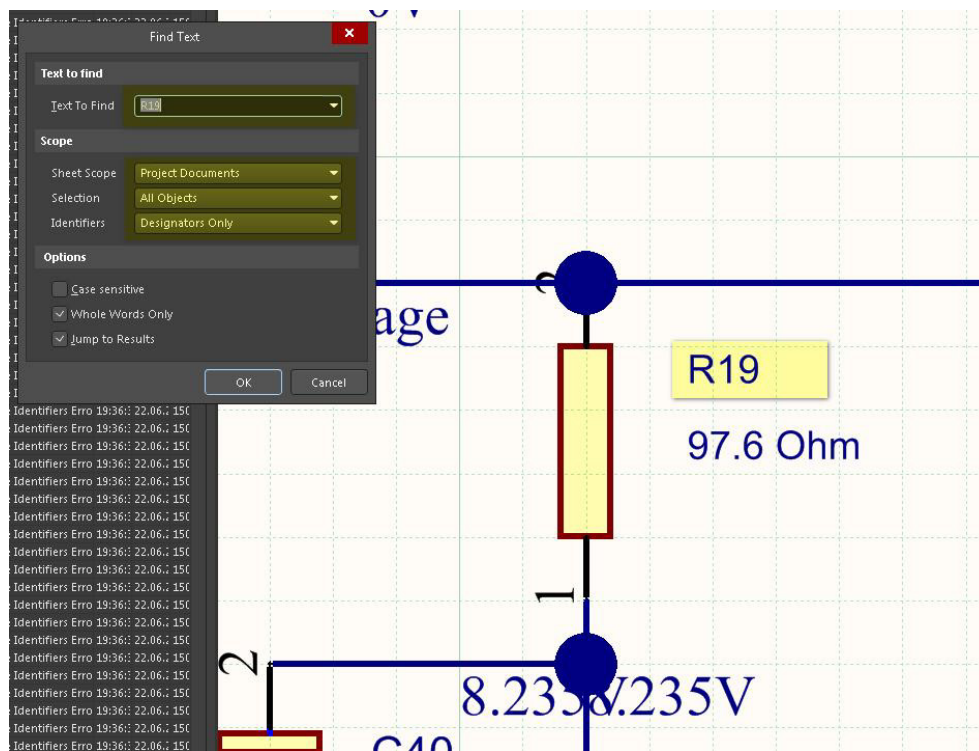
Fig. 9.3 - Component search with Find Text tool

Object navigation

In OrCAD Capture, the *Navigation Window* is navigation tool, which appears when you select a net and choose the **Signals** command from the context menu (Fig. 9.4).
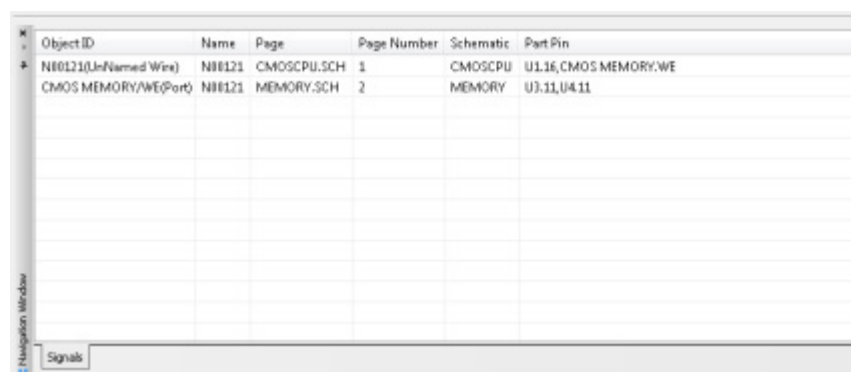
Fig. 9.4 - The Navigation Window panel

When selecting a project in *Project Manager* panel and running the **Edit » Browse** command, the list of available objects with navigation ability appears (Fig. 9.5).
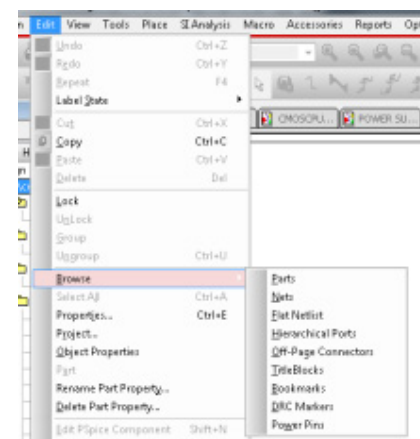
Fig. 9.5 - Running object navigation in OrCAD Capture

The *Navigator* panel is the excellent navigation tool in Altium Designer's schematic. This panel allows you to easily find the desired component or net. Open the panel, right-click on the top of the panel and choose **Compile All** – all components and nets will be listed in the bottom sections of the panel. If you click on an object here it will be zoomed in the workspace. There is the interesting **Connectivity Graph** tool. Run it from the right-click context menu, and select a net – connection points of this net will be shown on the schematic sheet (Fig. 9.6). Use **Ctrl+C** shortcut to disable the graph.
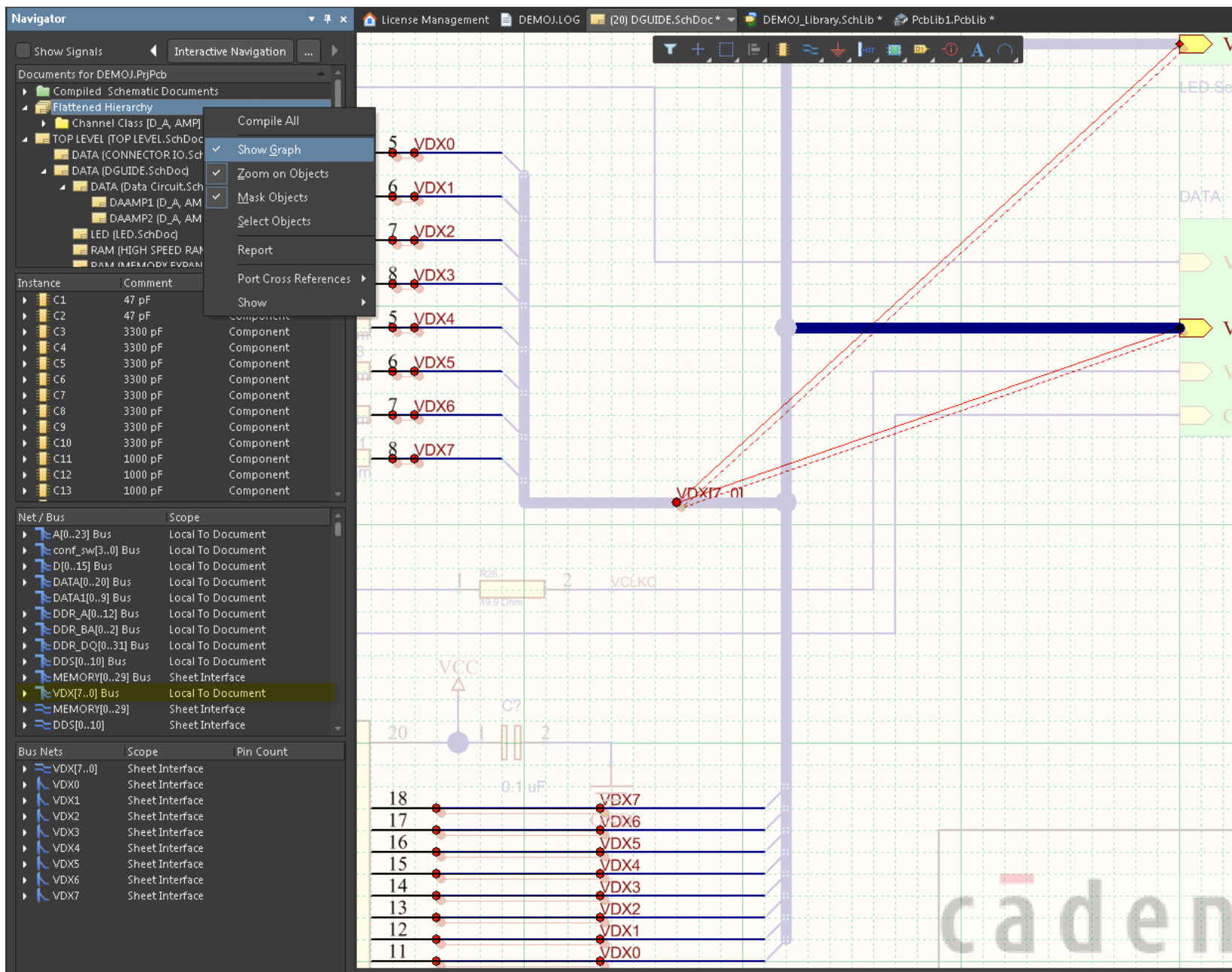
Fig. 9.6 - The Navigator panel and the connectivity graph

The **Interactive Navigation** button at the top of the *Navigator* panel allows you to select a net on schematic manually, and software will show you their connectivity points. In the interactive navigation mode you also can choose a hierarchical block pin or a port, and its counterpart will be shown on the corresponding schematic sheet.

10. Schematic verification and error resolving

This section covers schematic verification system in Altium Designer

Verification rule set

There are several ways for error checks, search and resolving in OrCAD Capture. Use the Design Rule Check dialog (**Tools » Design Rule Check**) to configure the checks. In this dialog, you can specify error types to check (**Run Electrical Rules** and **Run Physical Rules**) and choose rules to check on the **Electrical Rules** and **Physical Rules** tabs (Fig. 10.1).
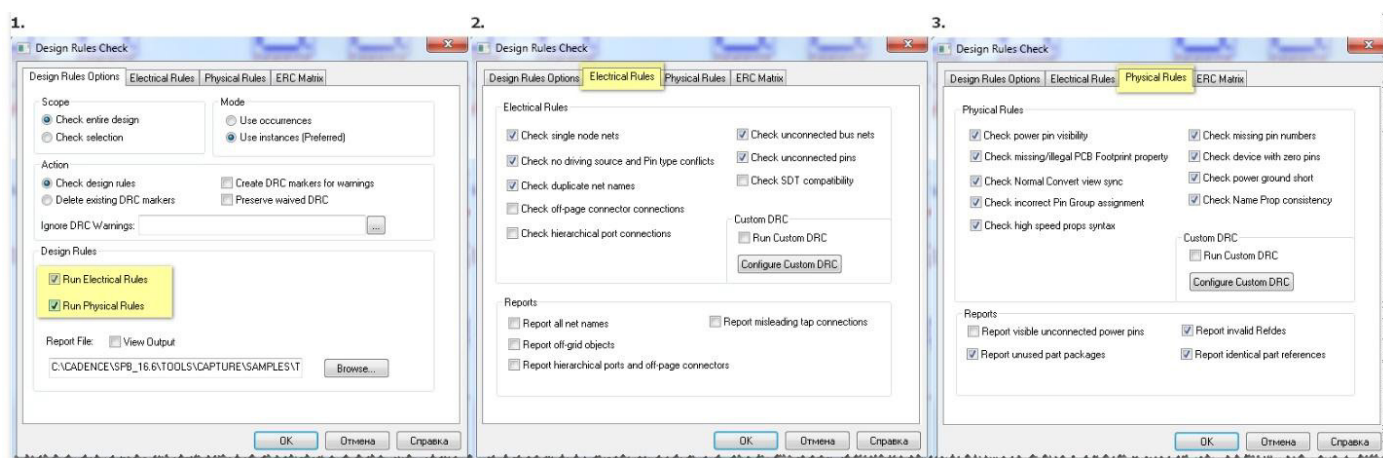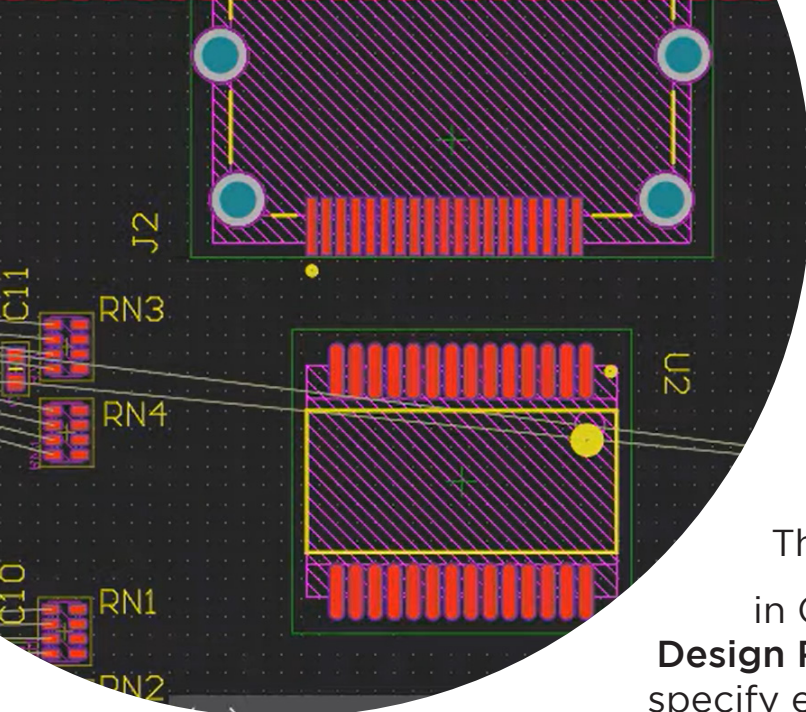
Fig. 10.1 - Rule check configuration in OrCAD Capture

There are two violation levels in OrCAD Capture – Warning and Error.

The following checks should be noted among others:

⊛ Check duplicate net names

⊛ Check no driving source and Pin type conflicts

⊛ Check missing/illegal PCB Footprint Property

Connectivity matrix is shown on the **ERC Matrix** tab, similar to it in Altium Designer (Fig. 10.2).
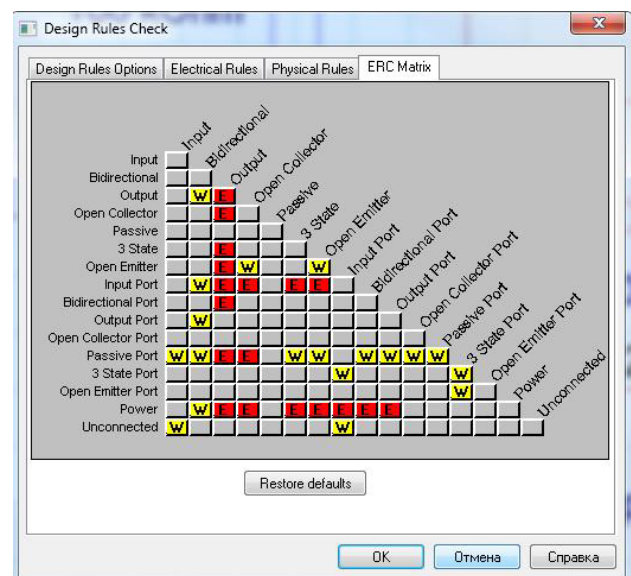
Fig. 10.2 - Connectivity matrix in OrCAD Capture

In Altium Designer, electrical rule check is performed during the compilation process. Errors are checked in accordance with the settings in the **Error Reporting** and the **Connection Matrix** tabs of the *Project Options* dialog (**Project » Project Options**). Error checks on the **Error Reporting** tab are organized in groups, in accordance with object types, for example **Violations Associated with Nets, Violations Associated with Components**, etc. (Fig. 10.3).
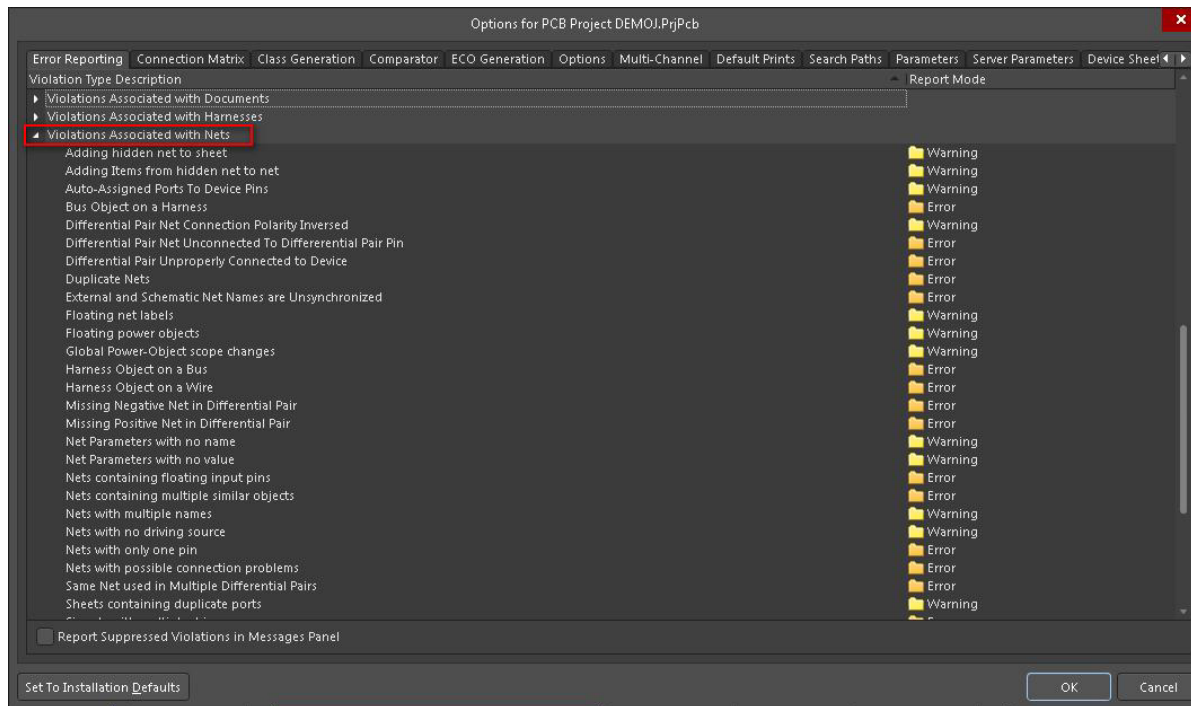
Fig. 10.3 - Electrical rule checks configuration in Altium Designer

You can specify an impact of each violation from one of four values: No Report, Warning, Error, Fatal Error. It will help to resolve the most critical errors in the design and ignore ones of little import.

Mentioned above checks in OrCAD are similar to the following ones in Altium Designer:

> Duplicate Nets

> Net with no driving Source

> Missing Components Models

The electrical connectivity for pins of different types is checked in accordance with the settings in the Connection Matrix (Fig. 10.4).
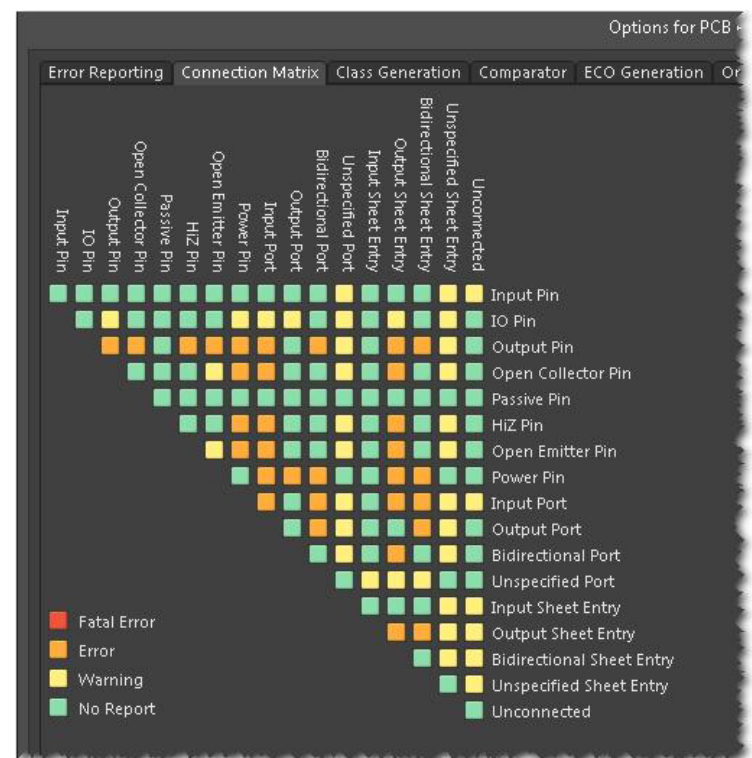
Fig. 10.4 - The Connection Matrix tab of the Project Options dialog

Error search and highlighting

When schematic check is run is OrCAD Capture the special markers (green circles) are shown where a error occurs. For marker navigation run the **Edit » Browse » DRC Markers** command or the **File » Check and Save command**, and the *Online DRCs* panel with **Warning** and **Errors** tabs appears. Double-click an error to navigate it to its location in the design (Fig. 10.5).
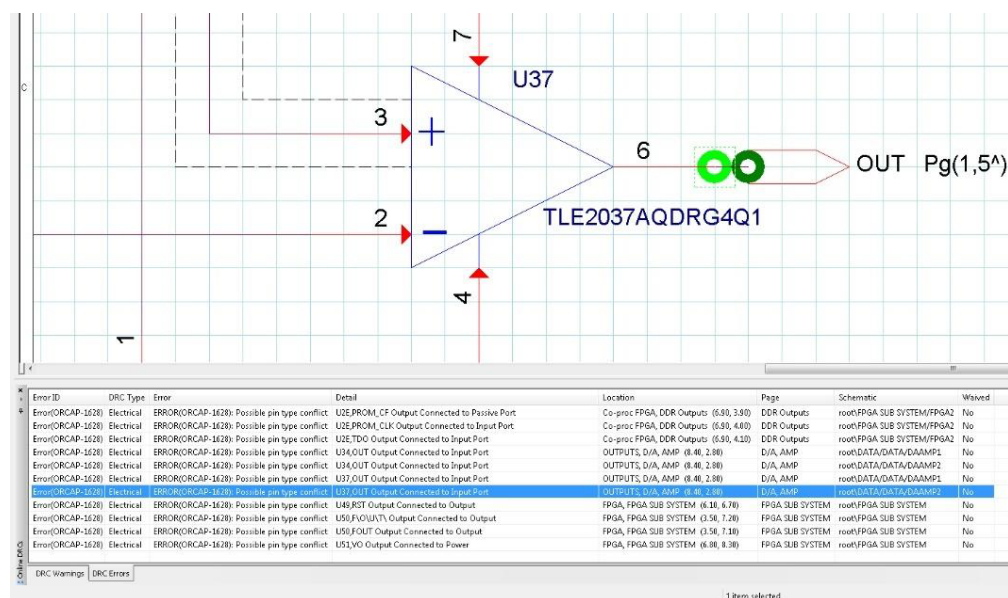
Fig. 10.5 - Error search with the Online DRCs panel in OrCAD Capture

In Altium Designer, electrical rule check is performed with the **Project » Compile PCB Project <ProjectName>** command. The *Messages* panel appears, where all detected violations with their descriptions are listed (Fig. 10.6).
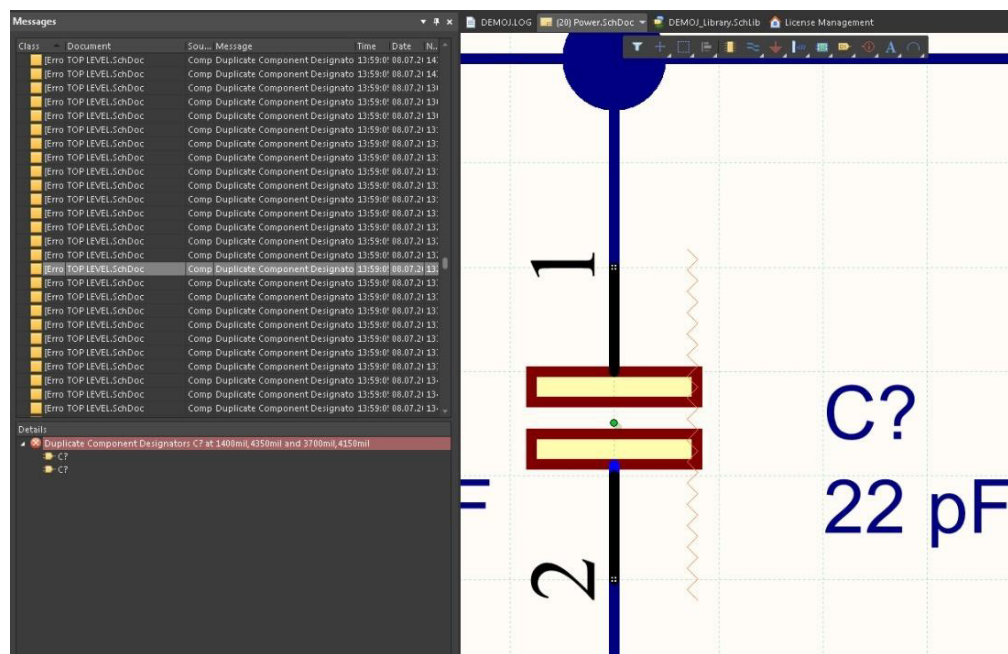
Fig. 10.6 - Violation list in the Messages panel

Double-click a violation in the panel, and its location will be zoomed in the workspace. In the **Details** section details of the violation will be shown. Generally violations with Fatal Error and Error report mode defined are highlighted with squiggly line. Hover the cursor on the violation location, and popup message with its description will be shown.

When all violations are resolved recompile the project to make sure there are no violations left.

11. Bill of Materials preparation

This section covers Bill of Materials (BOM) report preparation and its generation with ActiveBOM feature in Altium Designer.

Configuring properties and templates

For generating a BOM in OrCAD Capture you can use the standard **Tools » Bill of Materials** feature (Fig. 11.1).
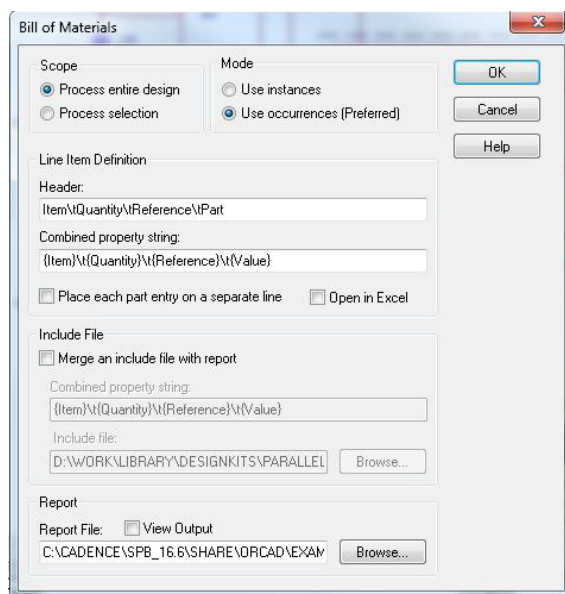
Fig. 11.1 - BOM generating in OrCAD Capture

Here you can specify attribute headers in the **Header** fields and the attributes that should be read from schematic components in the **Combined property string** field. You can use Excel-format report generating.

As you can see, this is very different from what Altium Designer offers. Only one provided template is available for generating a BOM in OrCAD Capture in the classic fashion.

If you have the CIS option available and a connected database, which the schematic is created from, you can use alternative way for BOM generating using the *Standard Bill of Materials* dialog (**Reports » CIS Bill of Materials » Standard**). Here you can select a property in the **Select Properties** list and move it to the **Output Format** list, and therefore create a list of properties to include in the BOM report to generate. You can use both schematic component properties and CIS database component properties if the database is connected and match to the schematic components (Fig. 11.2).
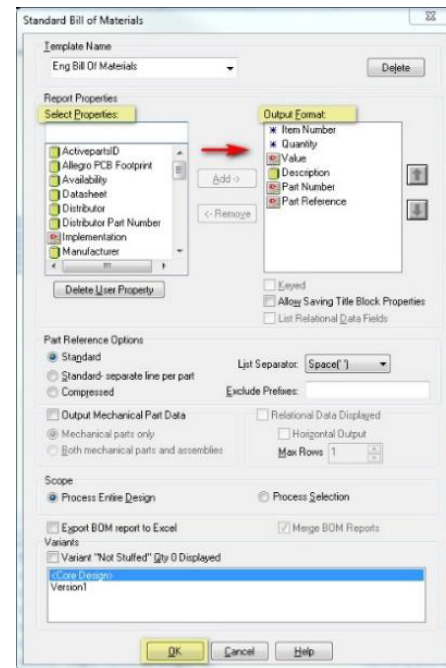
Fig. 11.2 - BOM generating with CIS option in OrCAD Capture

Additional way in OrCAD Capture is BOM generation based on Crystal Report templates. The software has a number of basic templates embedded, however you must have Crystal Report license to create a new template, or use a free alternative that supports the Crystal Report format. Use the *Crystal Reports Bill of Materials* dialog (**Reports » CIS Bill of Materials » Crystal Reports**) to work with reports of this type in OrCAD Capture.

BOM generation in Altium Designer is performed with the *Bill of Materials* dialog (**Reports » Bill of Materials**). Here you can specify component properties to include in the generated BOM report, select its format and template. All properties are migrating from schematic to OrCAD Capture perfectly, and they can be included to the generated BOM file. Simply select them in the **All Columns** section and check the desired properties in the **Show** column. Checked properties appear in the main grid section of the dialog. Select output format in the **Export Options** section. The **Add to Project** and the **Open Exported** options allow you to add BOM report to the project and open it after generating respectively. Select desired Excel template from the template folder (C:\Users\Public\Documents\Altium\AD18\Templates by default) in the **Excel Options** section. Press **Export** in the bottom of the dialog to proceed with the BOM generating (Fig. 11.3).
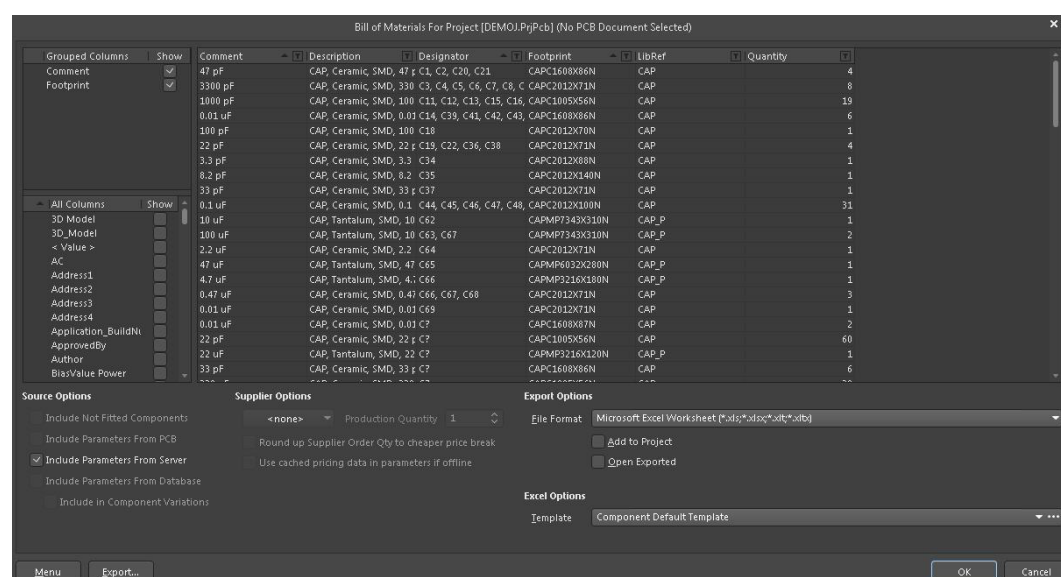
Fig. 11.3 - The dialog for Bill of Materials configuring and generating in Altium Designer

ActiveBOM

One of the most powerful and efficient features in Altium Designer is ActiveBOM. This is the dedicated type document with the .BomDoc extension, which can be added to a project in a standard fashion, for example with the **File » New » ActiveBOM Document** command. This is the interactive list of components of the project and at the same time bill of materials for your project that is able to interact with supplier catalogs via Internet. Component databases from suppliers such as Farnell, Digi-Key, Future Electronics etc. includes millions of items with up-to-date parameters and pricing information. ActiveBOM allows you to automatically load key component properties that are required for the bill of materials, and find optimal pricing and supplier. In addition to this, ActiveBOM checks accuracy of current schematic component properties and shows expired components or items which no supplier can be found for.

To search component information for your project automatically press the **Edit** button next to the **Manufacturer Link** field in the **Solutions** section of the *Properties* panel. In the *Define Manufacturer Link Fields* dialog that appears specify parameter name for the manufacturer and the manufacturer part number (Fig. 11.4).
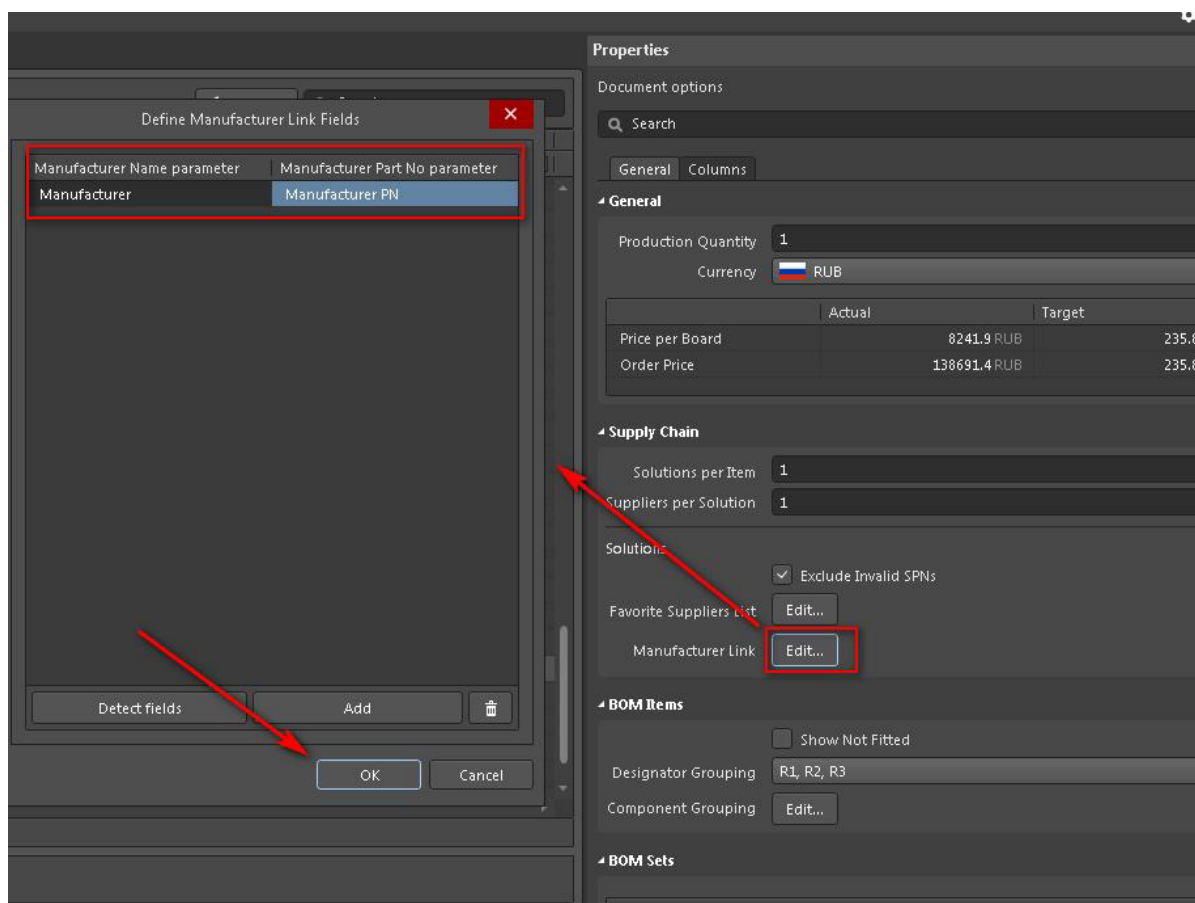
Fig. 11.4 - Select key component parameters from Properties panel when working with ActiveBOM

Then simply click on the **Refresh** button above the component list, and all solutions for each component will be shown at the bottom of the ActiveBOM document, if these solutions are available (Fig. 11.5).
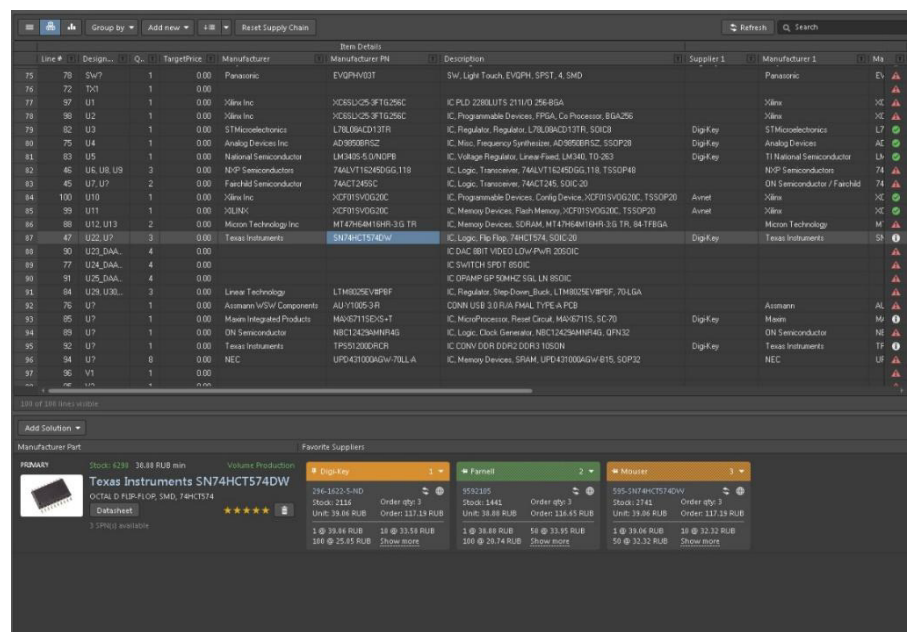
Fig. 11.5 - The ActiveBOM document when component solutions are loaded

Note the **BOM Checks** section at the bottom of the *Properties* panel, where you can configure checks of the BOM and see violations. Each BOM item has a status report resulted from the check (Fig. 11.6).
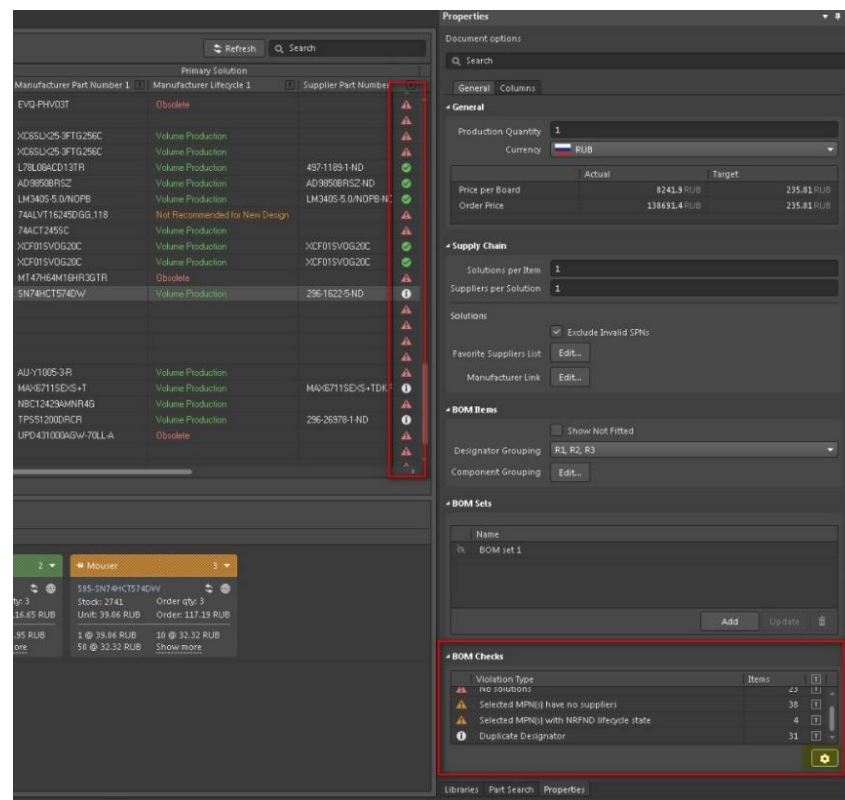
Fig. 11.6 - BOM Checks for detecting violations

There are a lot of things that can be told about this tool but the basic information given in this Migration Guide outlines a concept of this excellent feature. You can generate output BOM in the same way as it was described above, with the **Reports » Bill of Materials** command, when ActiveBOM file is opened as an active document in the main editing window.

12. Basic configuration of PCB Editor

This section covers settings for units, grids, origin and colors.

Units and grids configuration

Units in OrCAD PCB Editor is configured on the **Design** tab of the *Design Parameters Editor* dialog (**Setup » Design Parameters**). Here you also can define design accuracy, workspace sizes and other parameters (Fig. 12.1).
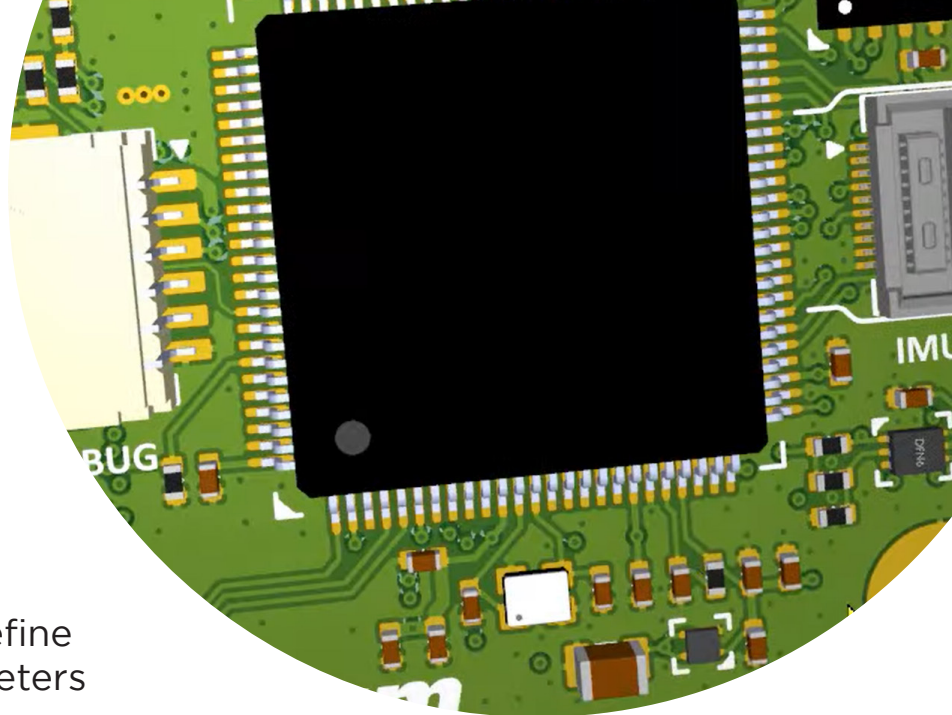
Fig. 12.1 - Units configuration in OrCAD PCB Editor

Grids can be configured with the **Grids** button on the **Display** tab of the dialog or with the **Setup » Grids** command. In the *Define Grid* dialog that appears you can define grid for each layer separately or for all layer simultaneously. There are two grids: **Etch** and **Non-Etch** – for all signal layers and for all non-signal layers respectively (Fig. 12.2).
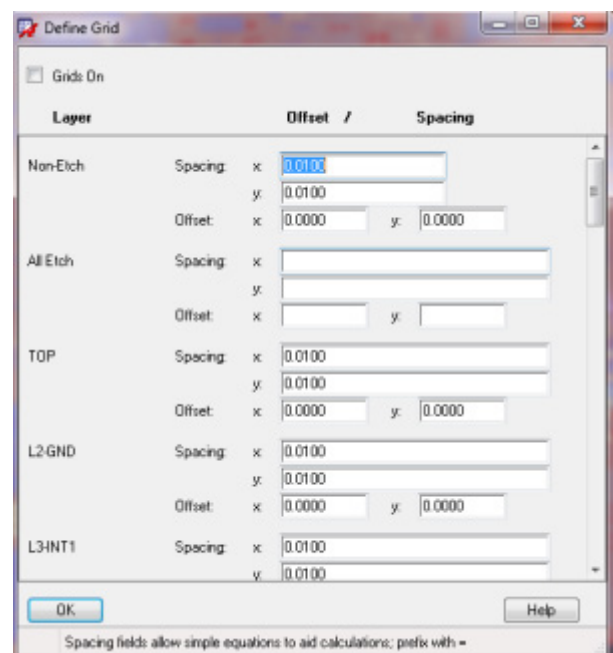
Fig. 12.2 - Grid configuring in OrCAD PCB Editor

In Altium Designer, configuration of units is performed in the **Grid Manager** section of the *Properties* panel in the **Board** mode (active when there are no selected objects in the PCB Editor workspace), as shown on Fig. 12.3.
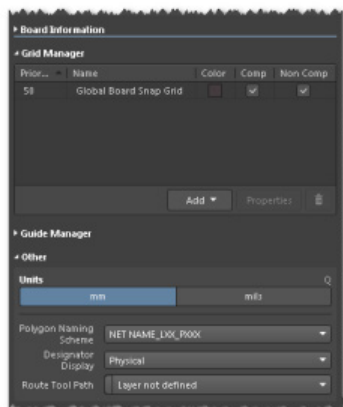
Fig. 12.3 - Grid configuring in Altium Designer's Properties panel
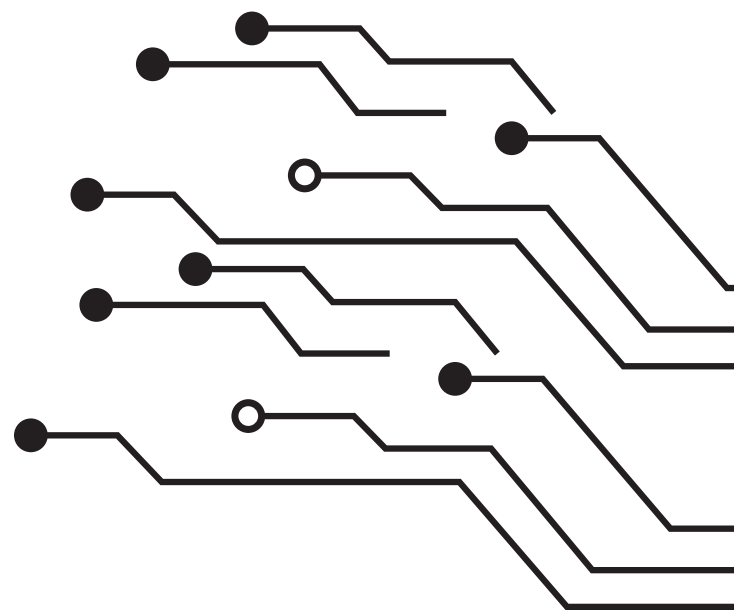
In the **Grid Manager** section you can leave the defined global grid and change its step with the **G** shortcut. You can add a cartesian or a polar grid by clicking on the **Add** button. Grid application can be changed, for example for using it for component placement only.

The origin

In OrCAD PCB Editor, you can move the origin using the **Setup » Change Drawing Origin** command. You can hide the origin in the *Design Parameter Editor* dialog (**Setup » Design Parameters**) – disable the **Display Origin** option on the **Display** tab. You also can change color or disable corresponding Drawing_Origin layer in the **Drawing Format** settings (**Display » Color/Visibility**), as shown on Fig. 12.4.
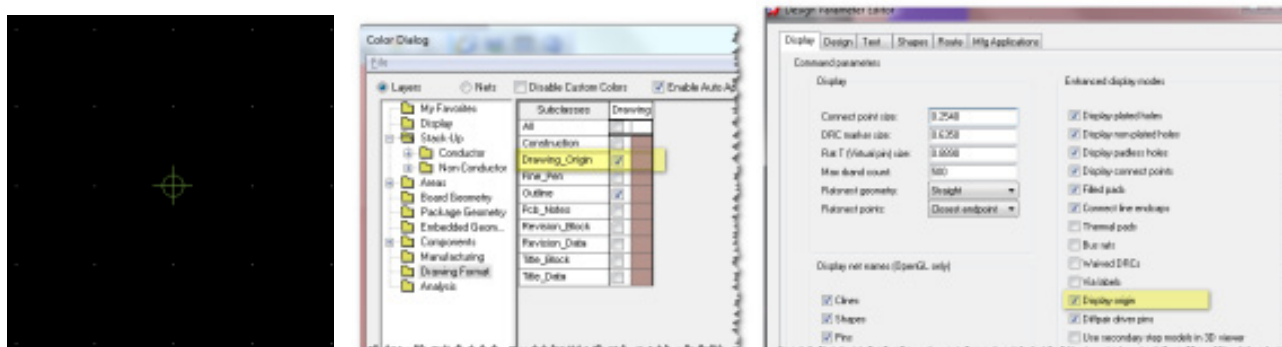
Fig. 12.4 - View and configuration of the origin in OrCAD PCB Editor

In Altium Designer, you can move the origin using the **Edit » Origin » Set** command. Use the **Reset** command from the same menu to move the origin to the default location. Change origin visibility and its color as required via the *View Configuration* panel (Fig. 12.5).
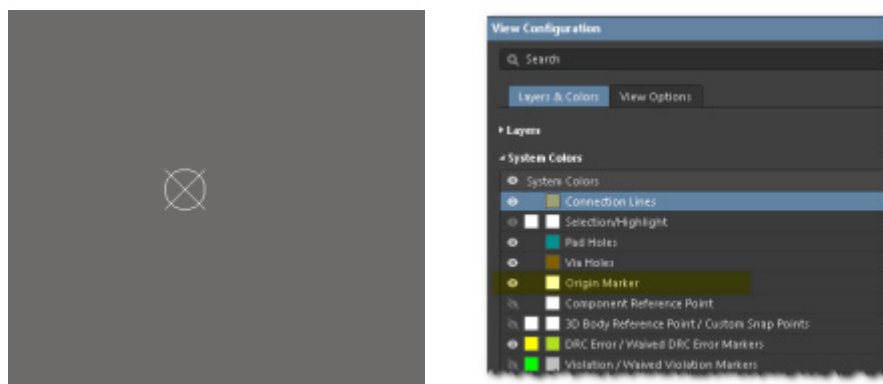
Fig. 12.5 - View and configuration of the origin in Altium Designer

In Altium Designer, you can configure system and layer colors in the *View Configuration* panel. You also can change colors quickly using layer tabs below the PCB workspace. Simply right-click on a tab and choose required command (Fig. 12.8).
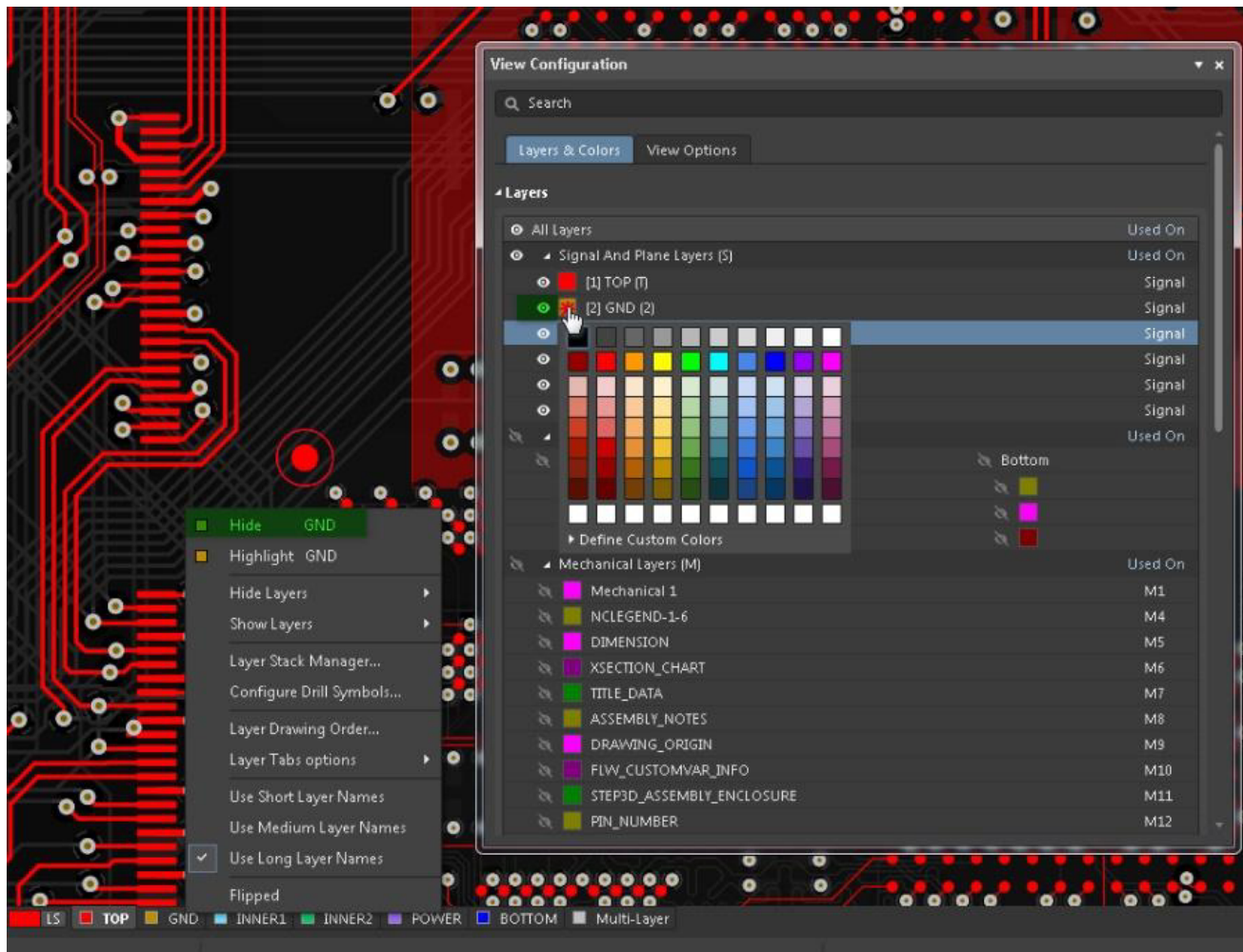
Fig. 12.8 - Configuration of layer color and visibility in Altium Designer

You also can enable a Single Layer Mode by using **Shift+S** shortcut, where outlines of other layers (besides the active one) are visible. Use **Shift+S** again to hide this outlines. You can also change this option on the **View Options** tab of the *View Configuration* panel (Fig. 12.9).
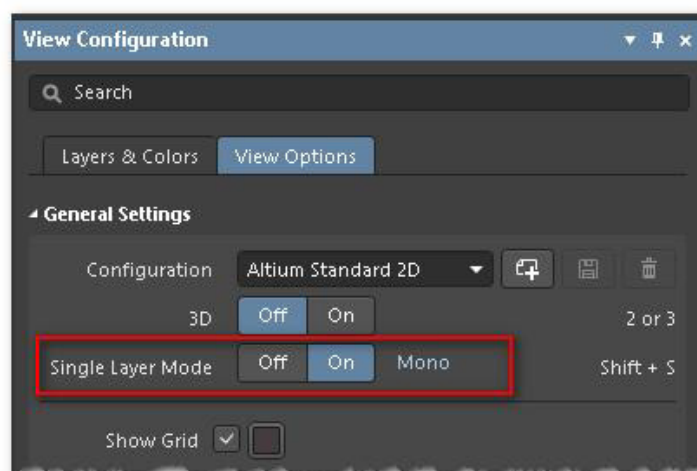
Fig. 12.9 - Enabling Single Layer Mode in Altium Designer

Object and layer color/visibility controls

In OrCAD, color and visibility configuration is performed in the *Color Dialog* (**Display » Color/Visibility**). In its **Display** section you can configure system color, for example for B**ackground, Via, Grids**, etc. (Fig. 12.6).
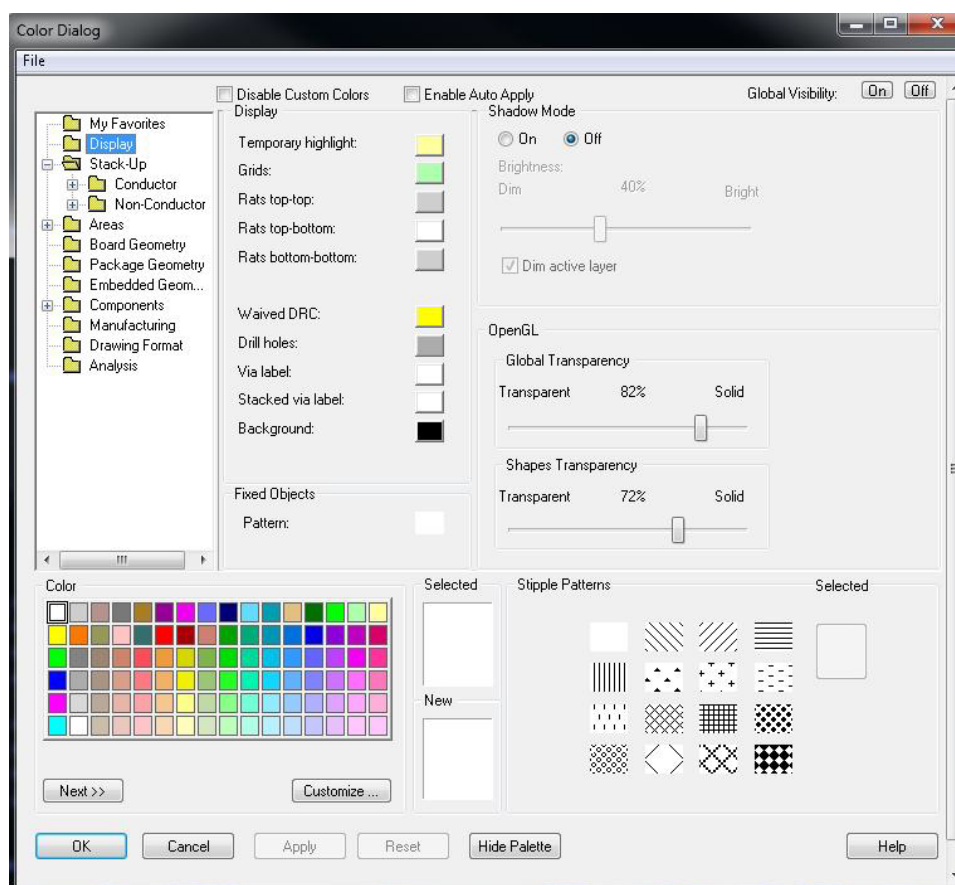
Fig. 12.6 - The Color Dialog in OrCAD PCB Editor

In addition, the *Visibility* panel is available, where you can quickly change the color and visibility for signal and plane layers. In the **Views** drop-down list, you can select set of visible layer that are currently required (Fig. 12.7).
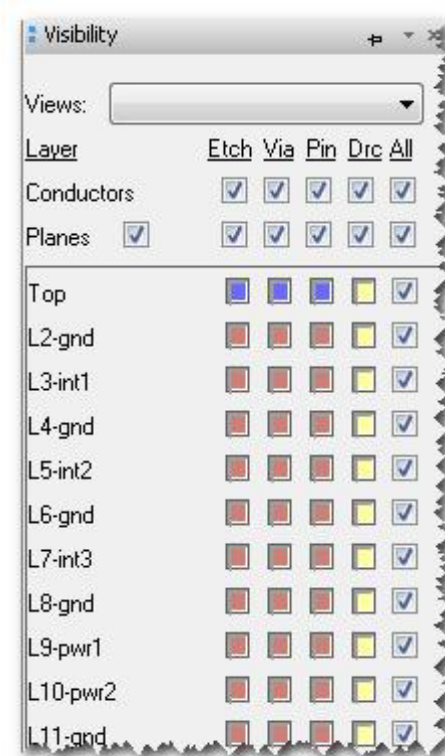
Fig. 12.7 - The Visibility in OrCAD PCB Editor
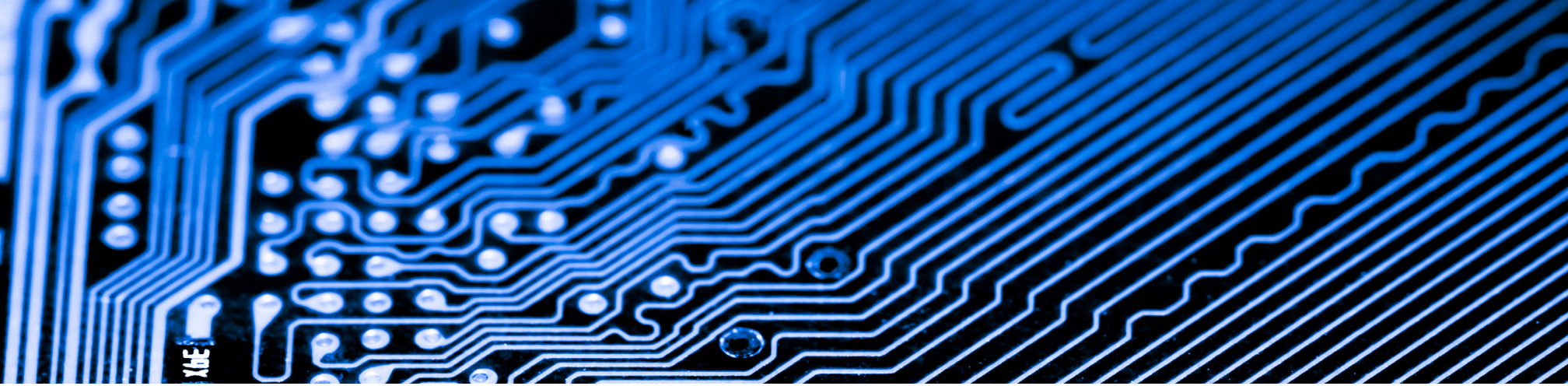
13. Creating layout template

This section covers creating board outline, mounting hole placement, keepout areas for routing and component placement

Creating board outline

In OrCAD PCB Designer, the board outline is placed on the dedicated reserved layer (subclass) **Board_Geometry\Outline**. You can draw it with lines and polygons. There is no special color to distinguish the board area from the rest of the workspace (Fig. 13.1).
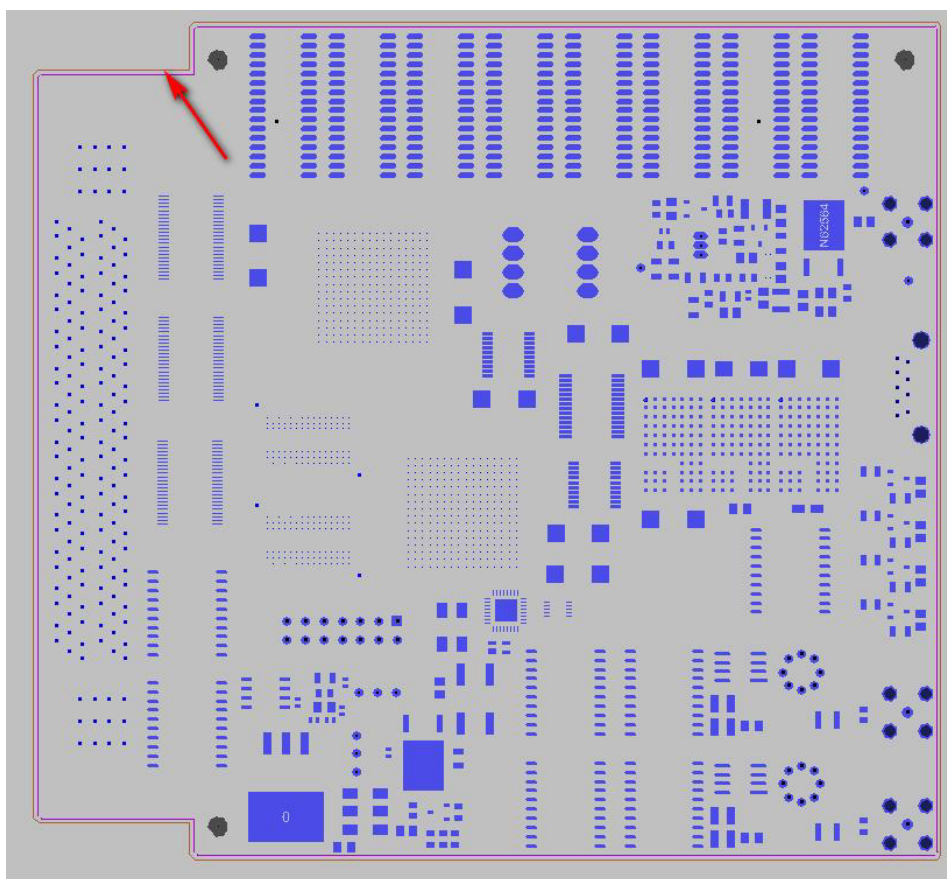
Fig. 13.1 - Board outline in OrCAD PCB Designer
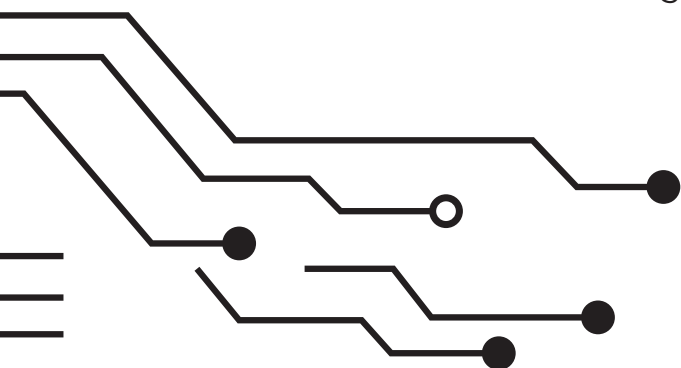
In Altium Designer, the outline is optional. The board area is displayed with dedicated color that is configurable in the **System Colors** section of the *View Configuration* panel. Outline layer can be changed easily in *Properties* panel. Even if this outline was removed you can restore it with the **Design » Board Shape » Create Primitives From Board Shape** command. Generally designers choose a mechanical layer for board outline (Fig. 13.2).
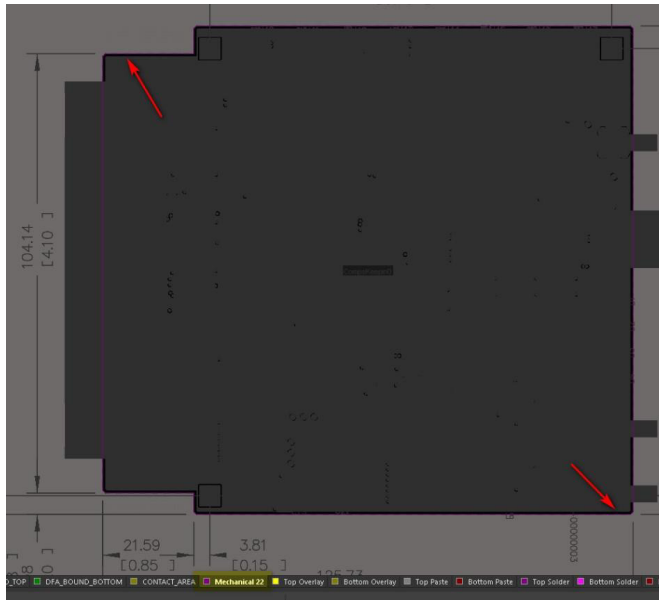
Fig. 13.2 - Outline of board's physical shape on the Mechanical 22 layer

Board shape can be defined in Altium Designer as following:

1. Manually, from lines or other primitives (**Place** menu)

Set the origin with the **Edit » Origin » Set** command. Select line tool (**Place » Line**) and start to draw outline on a mechanical layer, for example Mechanical 1 (additional mechanical layers can be activated in the *View Configuration* panel). You can use **J, L** keys to define outline vertices by their coordinates. Draw a closed outline, select it, and use the **Design » Board Shape » Define From Selected Objects** command. New board shape is ready.

2. From AutoCAD (DXF, DWG) imported data

Choose the **File » Import DXF/DWG** command. Specify units and map AutoCAD layers to Altium Designer's mechanical layers. Once the outline is imported use the **Board Shape » Define From Selected Objects** command.

3. From 3D Body (STEP, SolidWorks, or Parasolid)

Press **3** to switch to the 3D View mode, then choose the **Place » 3D Body** command. In the *Properties* panel select the **Generic** model type. Press the **Choose** button to import an external model file. Once the 3D model is imported choose the **Place » 3D Body** command, which is performed in three steps:

- ❯ Select the imported model
- ❯ Select face
- ❯ Press **Close** in the face choosing dialog with default options

New board shape is ready. The main difference of this method from the second one is that 3D Body can include holes and cutouts that are detected in Altium Designer automatically.

Board mounting holes

Mounting holes are placed as symbols of dedicated types from a *.bsm file, which library placement the psmpath variable is responsible for. You should create a padstack in Pad Designer, then create a mechanical symbol in OrCAD PCB Editor with the **File » New » Mechanical Symbol** command. A mechanical hole is defined by two *.dra и *.bsm files. It is easiest to place a mechanical hole in the **Placementedit** mode of the *Options* panel. You should select placement of those symbols in the **Place » Manually** menu (Fig. 13.3).
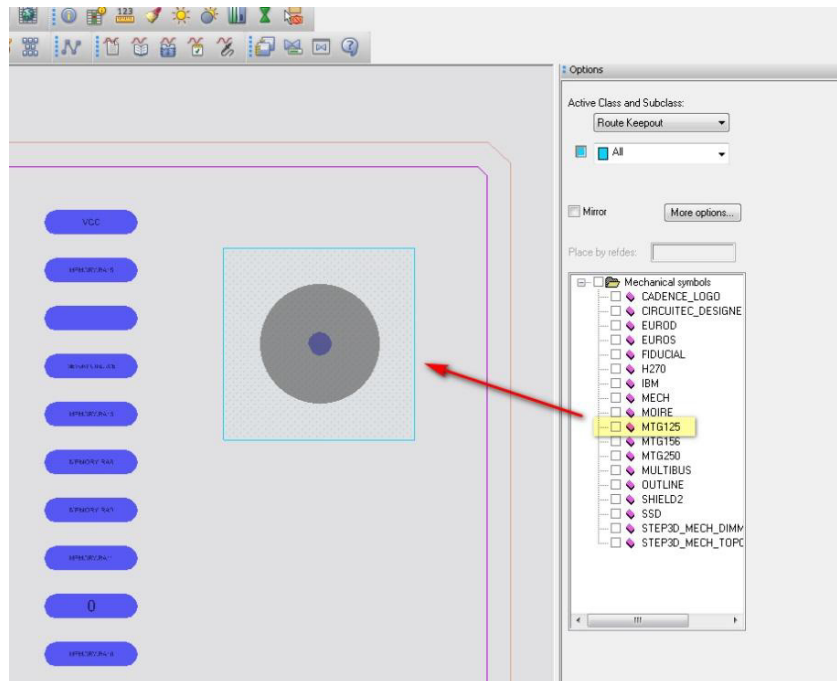
Fig. 13.3 - Mounting hole placement in OrCAD PCB Editor

In Altium Designer, padstack with holes are located in the dedicated Multi-Layer layer. Mounting hole placement is performed with the **Place » Pad** command. All hole properties are located in the *Properties* panel. You can place a hole by its coordinates with **J, L** keys (Fig. 13.4).
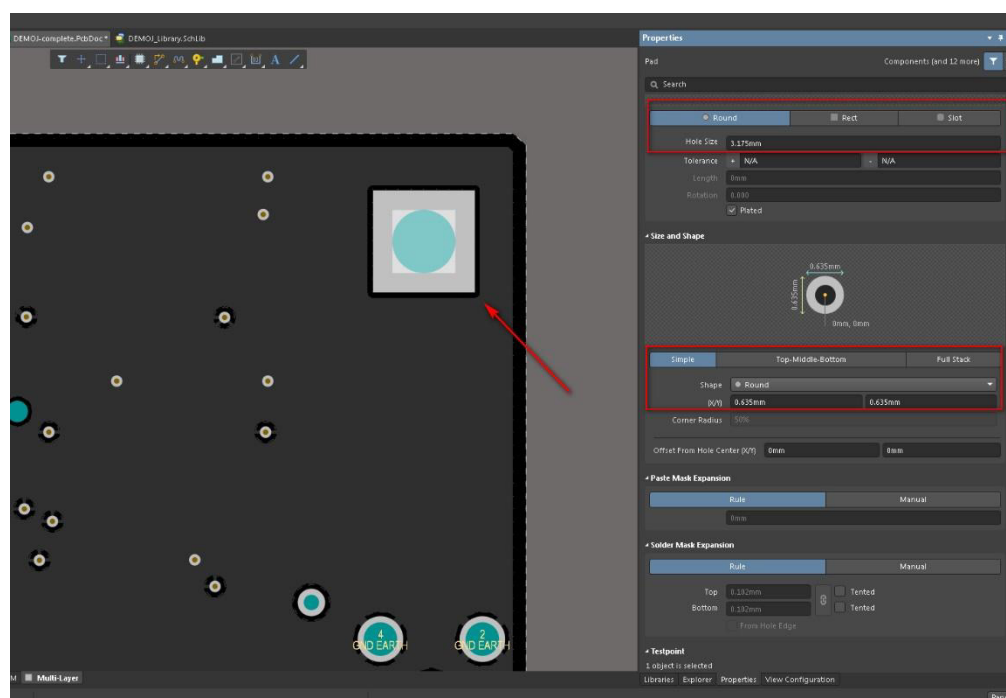
Fig. 13.4 - Mounting holes and their properties in the Properties panel

Keepout areas for component placement and routing

In OrCAD, keepout and keepin areas are placed in different layers. To create a routing/component placement keepin area Route_Keepin and Package_Keepin are used respectively. The Route_Keepout and Package_Keepout are classes for routing/component placement of keepout areas. You can draw those areas using lines or polygons. For this purpose, you can also use the **Setup » Areas** menu commands. To simplify the placement of these areas, you can use creation of equidistant outline along the board border or a cutout using the **Edit » Z Copy** command (Fig. 13.5).
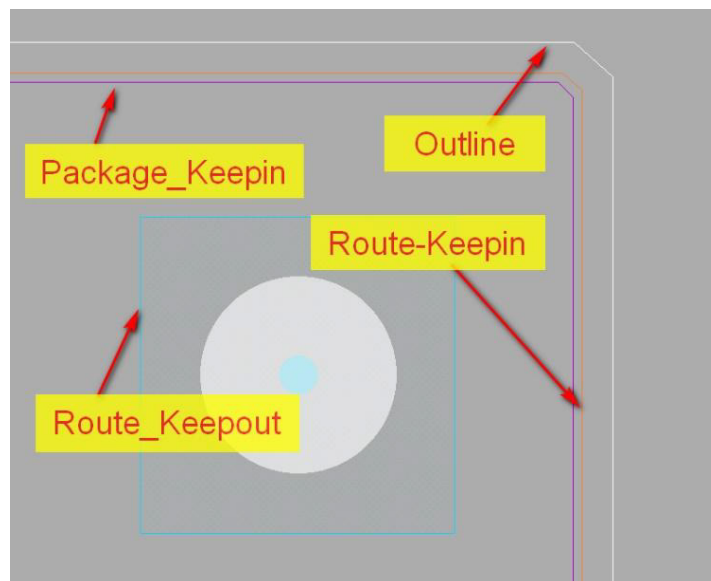
Fig. 13.5 - Keepin and keepout areas for routing and component placement in OrCAD PCB Editor

In Altium Designer, you can use the dedicated Keep-Out layer for keepout area placement. You can draw such area using the **Place » Keepout** command or **Active Bar**. Keepout area creation commands are available only when the Keep-Out layer is active. Also designers can make a keep-out area based on board outline.

Note that you can define keepout restrictions of a keepout area for specific object types in the *Properties* panel (Fig. 13.6).
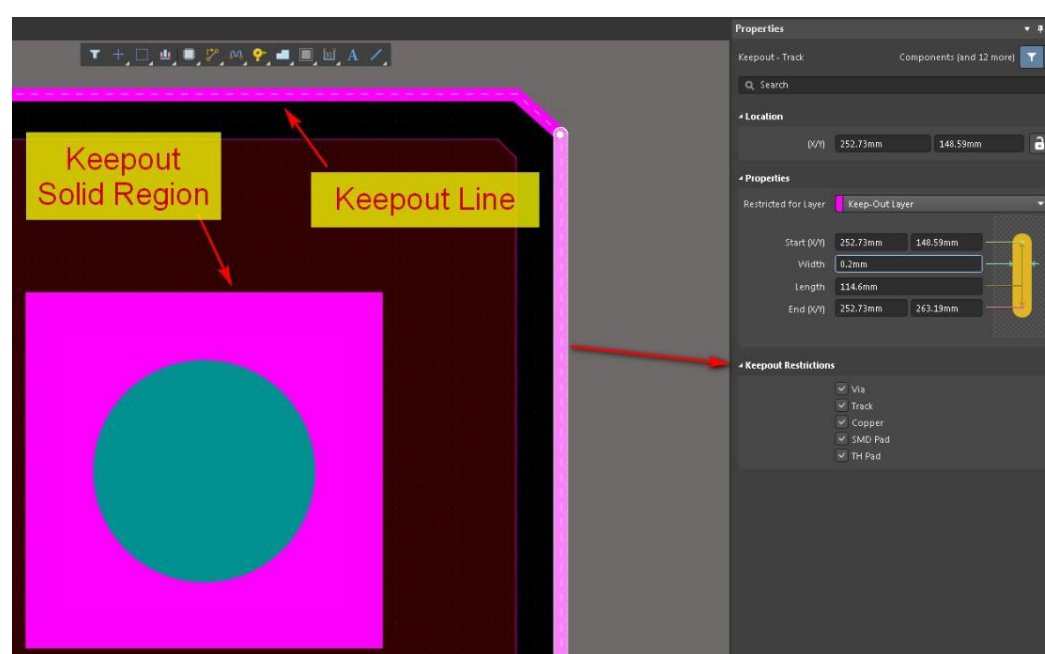
Fig. 13.6 - Keepout line and keepout region in Altium Designer

14. Defining the layer stack and via types

This section covers defining board layer stack and via types

Defining the layer stack

You can define the layer stack in OrCAD PCB Editor using the *Layout Cross Selection* dialog (**Setup » Cross-Section**). A new layer can be added via the context menu of the layer grid. There are three layer types available: Conductor, Plane and Dielectric. You can specify all basic electrical properties: thickness, dielectric constant, loss tangent, etc.
For the convenience of data capture of each layer, you can keep layer reference using the **Setup » Materials** command. You can change layer properties when required using the **Update Fields** button at the bottom of the stack editor. The **Show Impedance** and the **Show Diff Impedance** options allow you to use the impedance calculator for single signals and differential pairs, respectively (Fig. 14.1).
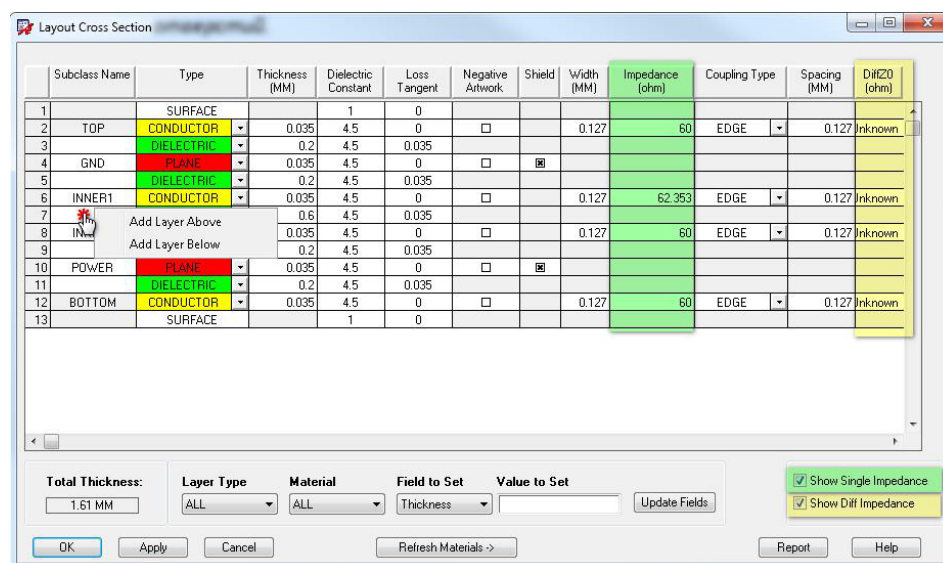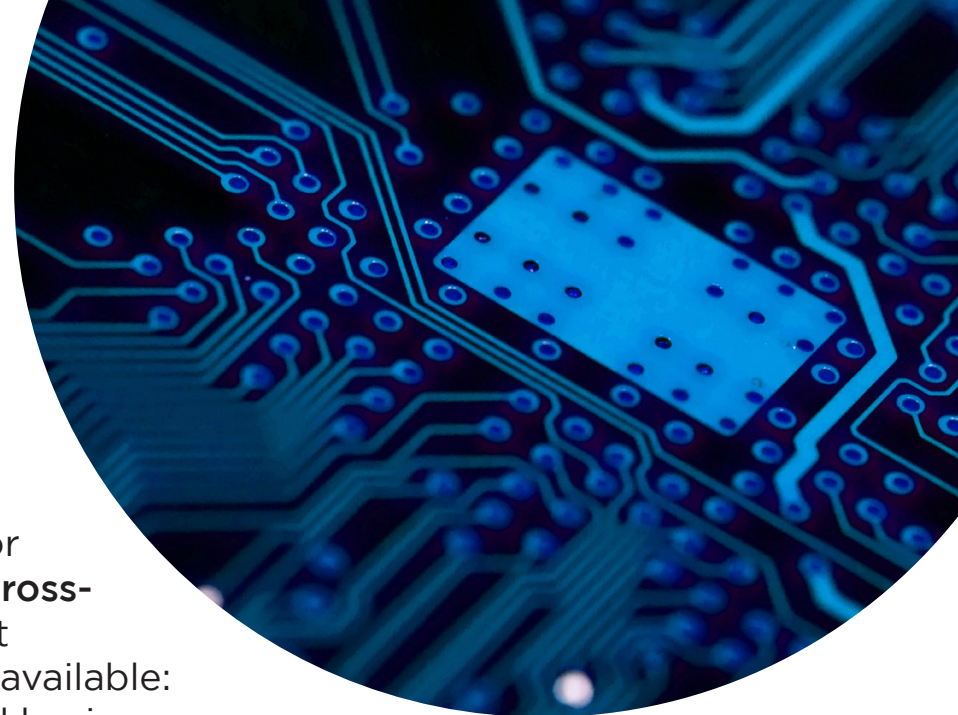
Fig. 14.1 - Layer stack editor in OrCAD PCB Editor

In Altium Designer, layer stack is defined in the *Layer Stack Manager* dialog (**Design » Layer Stack Manager**). The Altium Designer developers made sure that the standard layer sets were always at hand. Using the Presets button you can select a stack template for your design and then modify it as required. You can select any cell in the layer grid and add or remove layers using the right-click menu. The Move Up/Down buttons under the grid region allow you to change the layer position in the stack. Layer definition also depends on the settings at the top right corner of the stack manager – in pairs or one by one (Fig. 14.2).
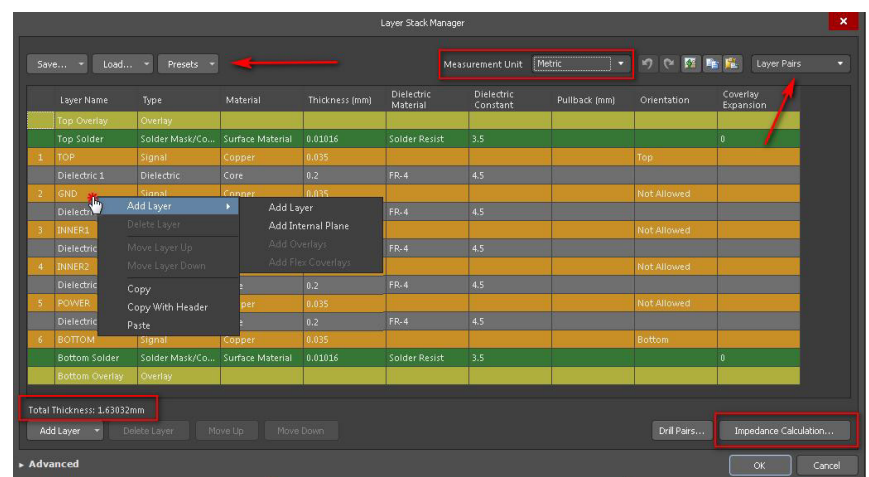
Fig. 14.2 - The Layer Stack Manager dialog in Altium Designer

Defining the via stack

In OrCAD, via stacks are created in Pad Designer. You can form blind and buried vias in OrCAD PCB Editor using the *Blind / Buried Vias* dialog (**Setup » B/B Via Definitions » Define B/B Via**), specifying start and end layers for any stack from library (Fig. 14.3).
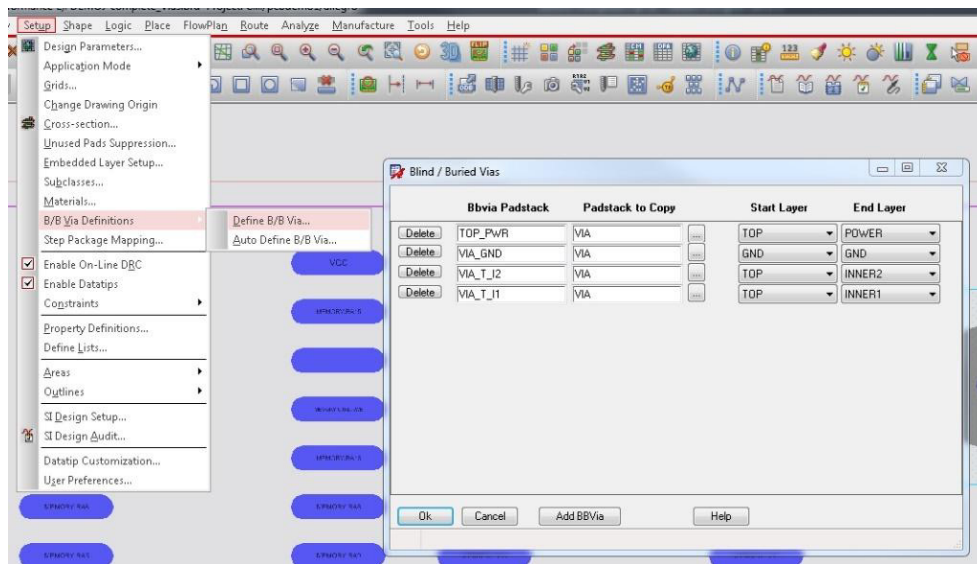
Fig. 14.3 - Defining the stacks for blind and buried vias in OrCAD PCB Editor design

The applicability of these vias should be specified in the **Constraint Manager** in the section of physical requirements.

In Altium Designer, use the **Place » Via** command to place a via. Configuration of a via is performed in *Properties* panel and is similar to padstack configuring. Diameter and hole size are specified here. In the **Drill Pair** drop-down you can select a pair which layers are connected by the via. You can define as many layer pairs as required using the **Drill Pairs** button. The same button is located in Layer Stack Manager. In the *Drill-Pair Manager* dialog you can add each pair individually using the **Add** button or add all possible for the current layer stack options using the **Create Pairs From Layer Stack** button (Fig. 14.4).
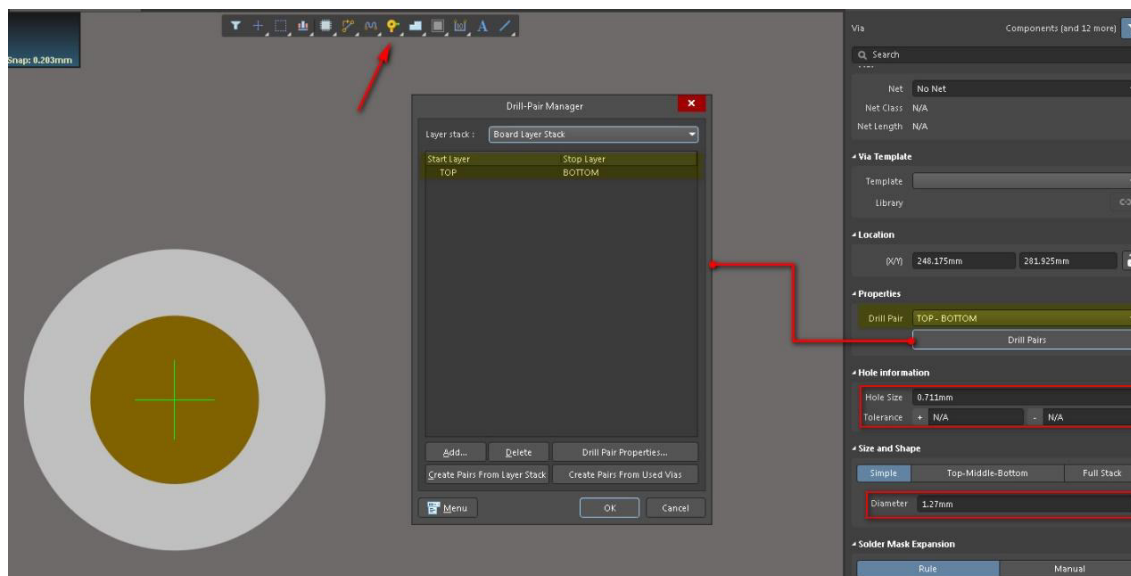
Fig. 14.4 - Configuring vias in Altium Designer

You also can use via templates from a Pad And Via library. A special design rule for controlling via properties for specific nets and net classes is discussed later in this guide.
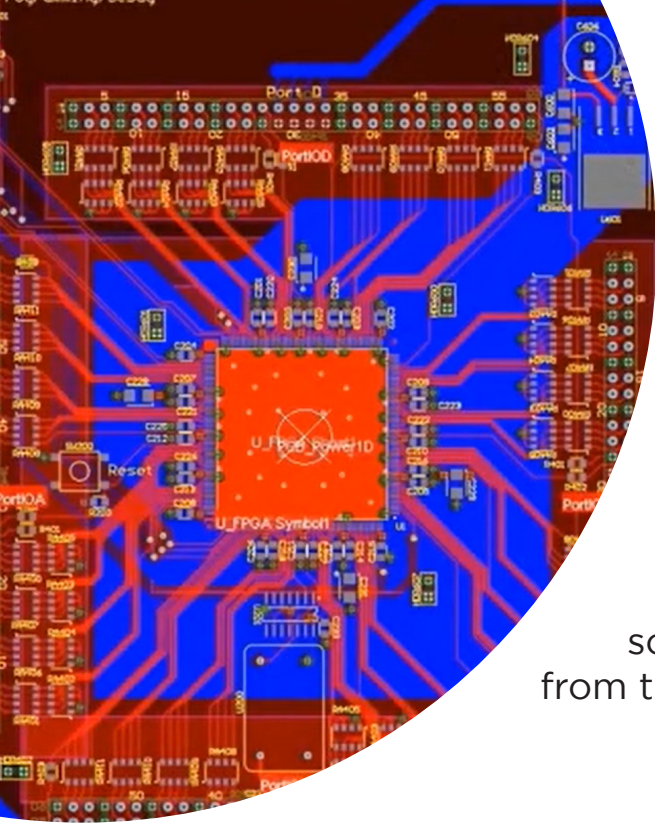
15. Schematic packaging and transition to layout design

This section covers transferring data into the PCB Editor to start placement and routing. In Altium Designer, you can precisely control the changes that are passed between editors.

The netlist and error messages

In OrCAD PCB Designer, the transferring of changes between schematic and PCB is performed either from the schematic editor or from the PCB editor. In the former case, the *Create Netlist* dialog (**Tools » Create Netlist**) is used (Fig. 15.1).
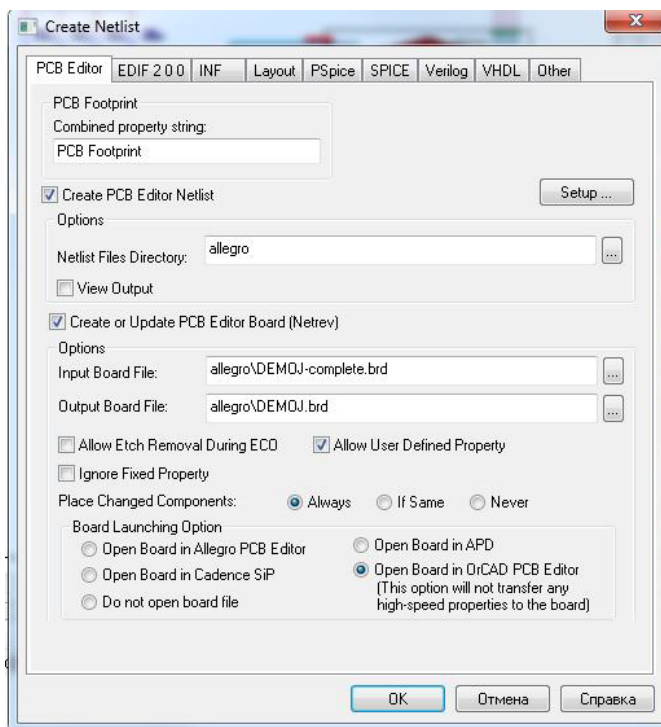
Fig. 15.1 - Netlist generating dialog in OrCAD Capture

There are no additional options, and all errors and warnings are recorded in the Session Log, which is available via the **Windows » Session Log** command (Fig. 15.2).

Fig. 15.2 - Schematic packaging data in the OrCAD Capture's Session Log

In OrCAD, three netlist files are created in the allegro subfolder of the project folder: pstxner.dat, pstxprt.dat, pstchip.dat. These files are coupled closely, and if one of them is lost, it is necessary to regenerate the netlist. If the system detects fatal errors during the generation, the netlist generation is interrupted and user must correct all errors based on the message log.

In OrCAD PCB Editor, the transferring changes from the schematic is performed via the **File » Import » Logic** command (Fig. 15.3).
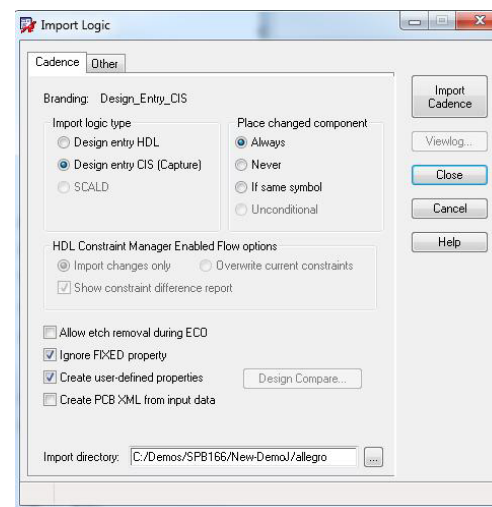
Fig. 15.3 - Schematic change transferring dialog in OrCAD PCB Editor

Specify the path to the netlist folder in the **Import directory** field. To view error log use the **File » Viewlog** command and the netlist.log file.

In Altium Designer, the **Design » Update PCB Document <PCB_file_name>** command is used to transfer data from the SCH Editor to the PCB. A similar function is provided by the **Design » Import Changes From <PCB_project_name>** command in the PCB Editor. As a result of the any of these commands running, a list of Engineering Change Order appears. Unlike OrCAD, in Altium Designer you do not work with a text netlist, and all changes and error messages are listed in single window.

To validate and make a final decision to apply changes the **Validate Changes** and the **Execute Changes** buttons are used, respectively (Fig. 15.4).
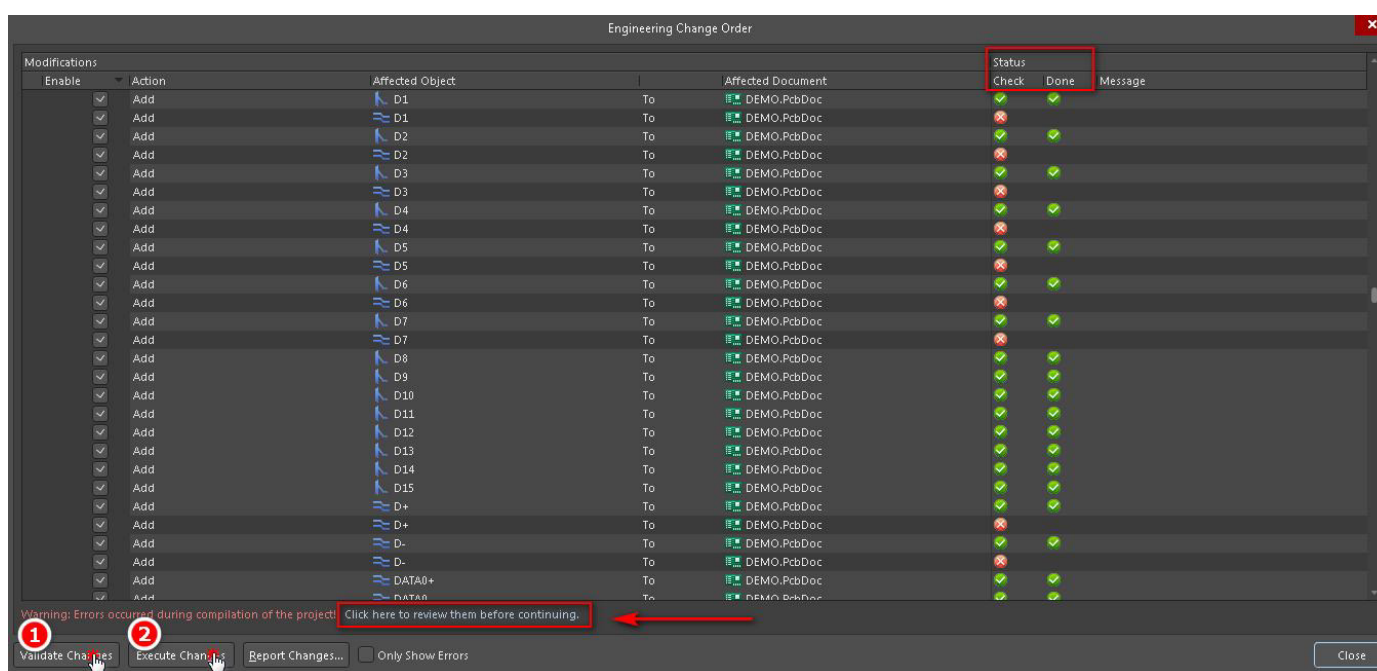
Fig. 15.4 - Schematic to PCB Editor changes transferring dialog in Altium Designer

All changes are grouped into collapsible categories. For example, in the Add Nets category, all the net that will be transferred to the PCB are listed, and there is a checkbox associated with each net that permits its transferring. Select any change from the list and right-click on it to access to the context menu, where you can choose various actions, for example, refuse to apply changes of a specific category.

If a change cannot be applyed and it does not pass the check, a red mark appears in the **Check** column. A green label indicates that the change has been verified and submitted to the PCB Editor.

Notice the warning message that may appear at the bottom of the *Engineering Change Order* dialog: Warning: Errors Occurred during compilation of the project. In this case, you can click the Click to to read them before continuing link to compile the project and display schematic violation list so that you can fix the most critical errors.

PCB Editor objects review after initial transfer

In OrCAD PCB Editor, components and net do not appear in the workspace automatically. You can view a list of transferred components, rooms, nets, and differential pairs in various ways. For example, a full list of components can be displayed via the *Options* panel in the **Placement Edit** mode (Fig. 15.5).
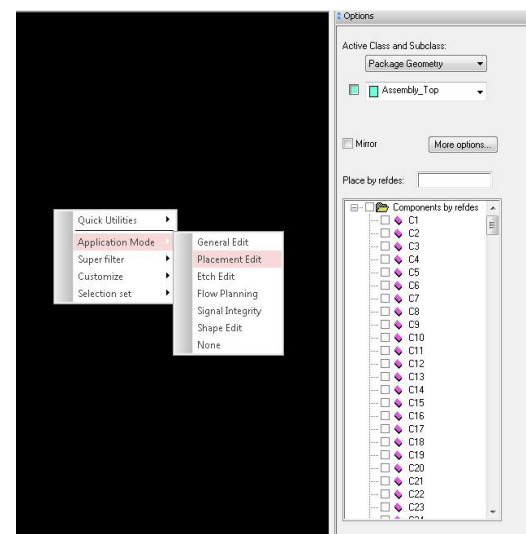
Fig. 15.5 - Component list in the Options panel in OrCAD PCB Editor

Also, a complete component and net list can be displayed in the *Constraint Manager*, via the Find panel, reports, etc.

In Altium Designer, once the data are transferred to the PCB, all components and connections are placed to the right of the board. The list of all components, nets, classes is available in the *PCB* panel (Fig. 15.6).
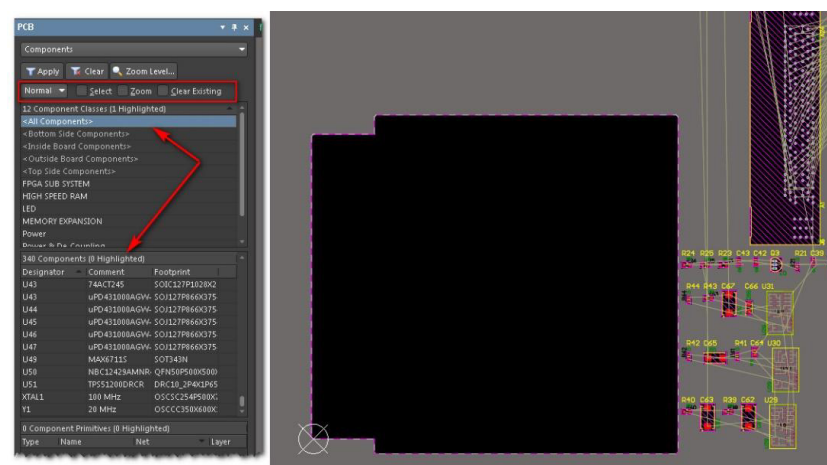
Fig. 15.6 - Components and nets after their transferring to Altium Designer's PCB Editor

In the following sections, we will discuss board layout, and component placement.
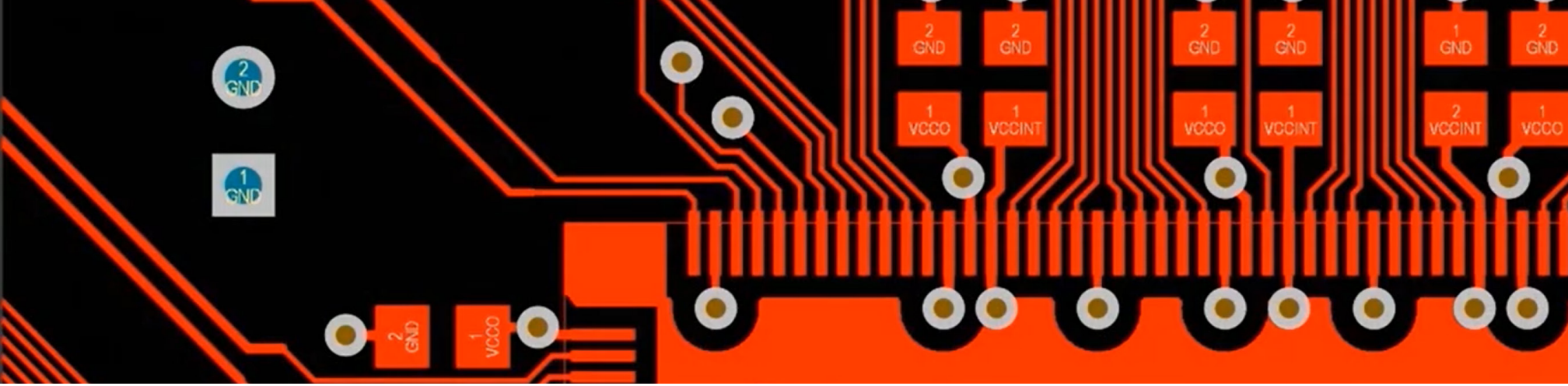
16. Component placement

This section covers component placement tools.

Manual placement

In OrCAD PCB Editor, you can place components manually using the **Placement Edit** mode. In this mode, the list of components is displayed in the *Options* panel. Simply select components and place them in the board area (Fig. 16.1).
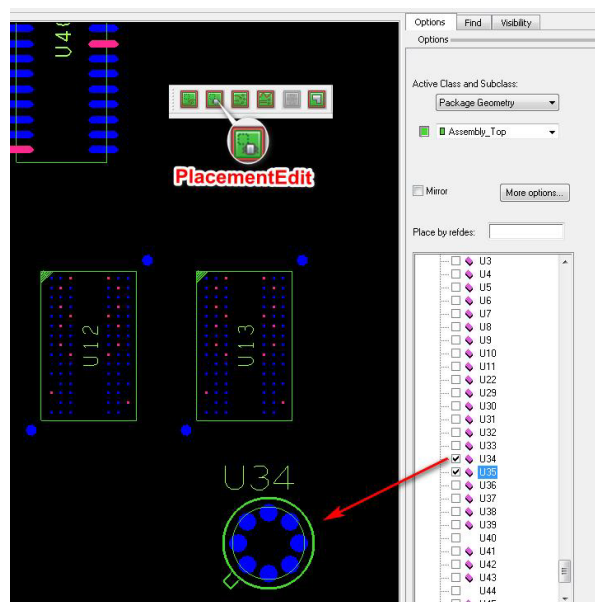
Fig. 16.1 - Manual component placement

You can perform placement using the **Place » Manual** command as well (Fig. 16.2).
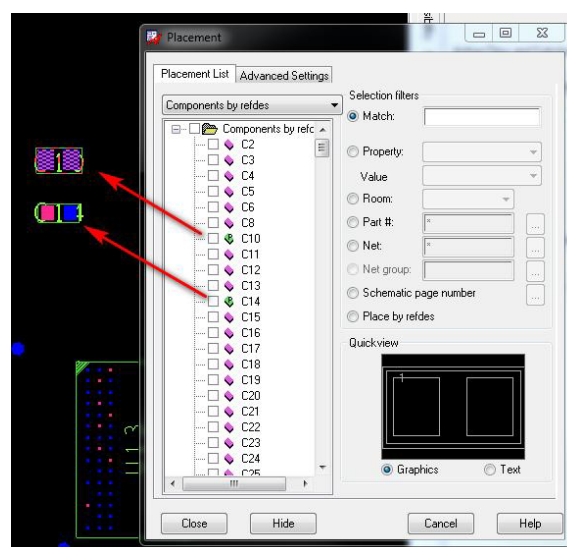
Fig. 16.2 - Component placement using the Place » Manual command

In Altium Designer, the manual component placement can be performed in a manner similar to this in OrCAD. Since all the components transferred from the schematic are placed in the PCB Editor workspace, you can manually move them in the board area one by one or as a group. In addition, you can display the component list in the *PCB* panel and, like in OrCAD, transfer them from the list to the board (Fig. 16.3).
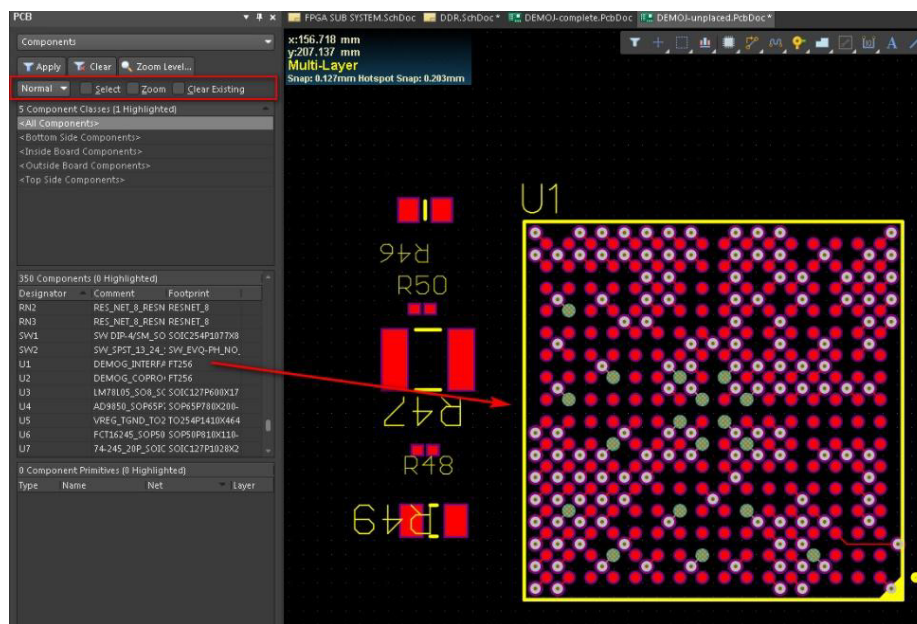
Fig. 16.3 - Component placement using the PCB panel in Altium Designer

Quick (semi-automatic) component placement

To quickly place components in OrCAD, you can use the *Quickplace* dialog (**Place » Quickplace**), which offers quite extensive set of options. You can choose placement based on the component properties, connected nets, schematic numbers, designators, numbers of leads, etc. (Fig. 16.4).
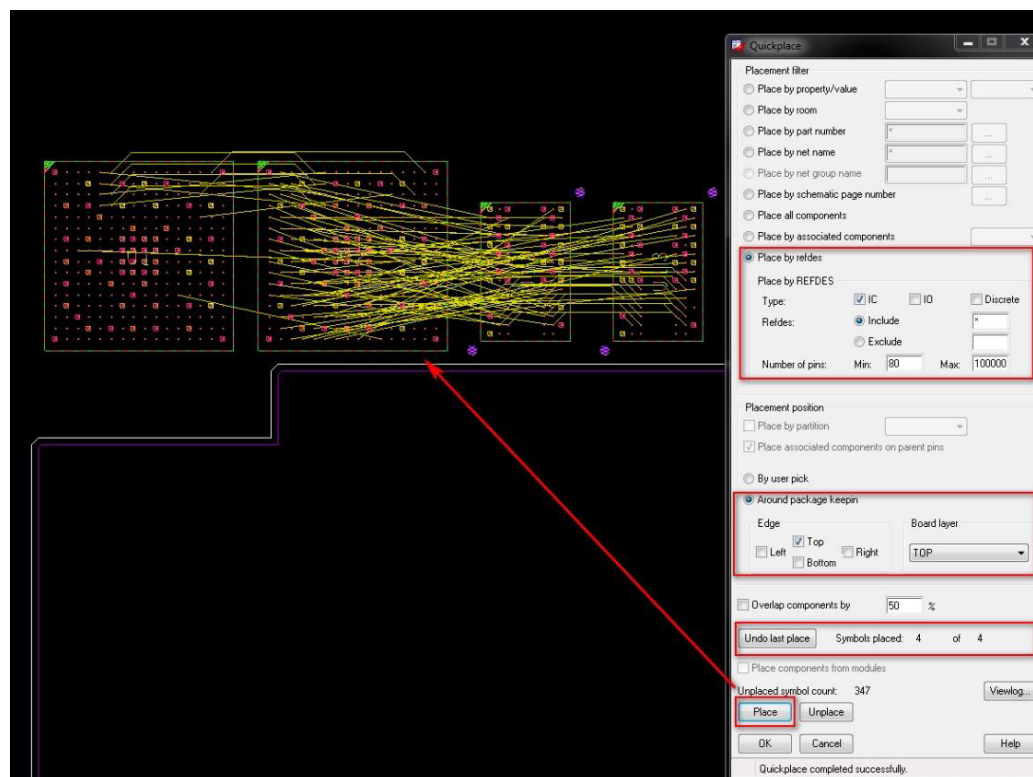
Fig. 16.4 - The Quickplace dialog in OrCAD PCB Editor

In Altium Designer, you can use commands from the **Tools » Component Placement** submenu to quickly place components. Most of these commands are available only if you selected the components you want to place beforehand. This can be performed, for example, using the *PCB* panel. Check the **Select** and the **Clear Existing** options, select the components you want to place, in the All Components class and use the **Tools » Component Placement » Arrange Within Rectangle** command. Then simply click and drag a rectangle in the board area where the components should be placed (Fig. 16.5).
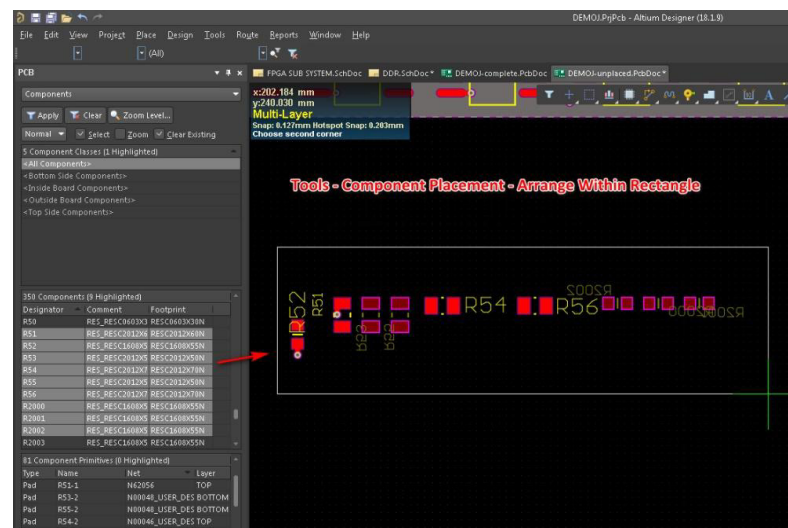
Fig. 16.5 - Component arrangement within rectangle in Altium Designer

Component placement by the schematic

You can select components on the schematic and place their footprint on the board. To do this in OrCAD PCB Editor, you need to run the manual placement command or click on the special button on the toolbar that launches the manual placement without a footprint symbol list window displaying. Then you select one or more components on the schematic and place them on the PCB (Fig. 16.6).
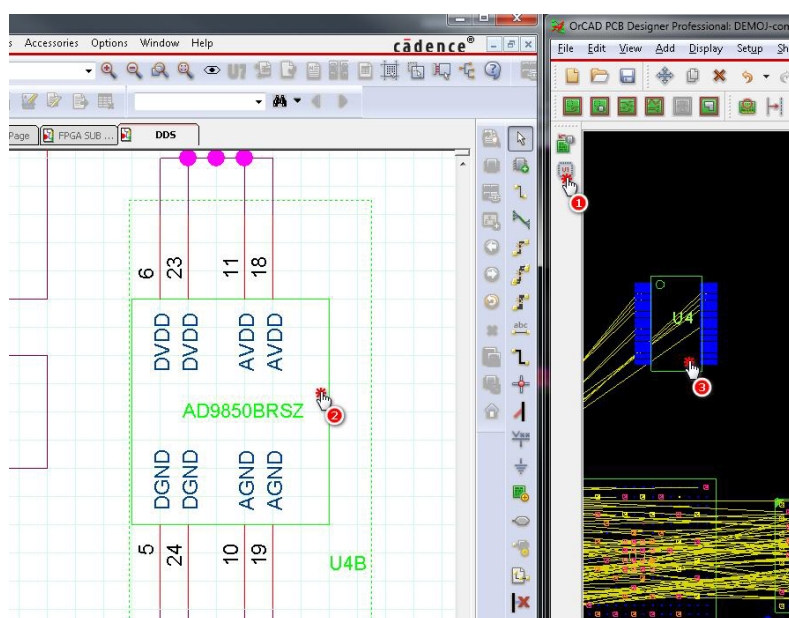
Fig. 16.6 - Component placement by the schematic in OrCAD

In Altium Designer, to place components by the schamic, you should activate Cross Select Mode first (**Tools » Cross Select Mode**). Once the mode is activated, you can select components in the SCH Editor. Then switch to the PCB Editor and use the **Tools » Component Placement » Arrange Within Rectangle** command (Fig. 16.5).

Of course, there are a number of other tools in Altium Designer that you can use for component placement. Depending on the specific situation, you can use any of them to speed up your design process.

17. Object classes generation

This section covers net, component, differential pair and xSignals object classes

Generation net and component classes

In OrCAD PCB Editor, classes are formed using the Constraint Manager (**Setup » Constraint Manager**) only. You can select a netlist in any of the rule categories and add a new class from the context menu (Fig. 17.1).
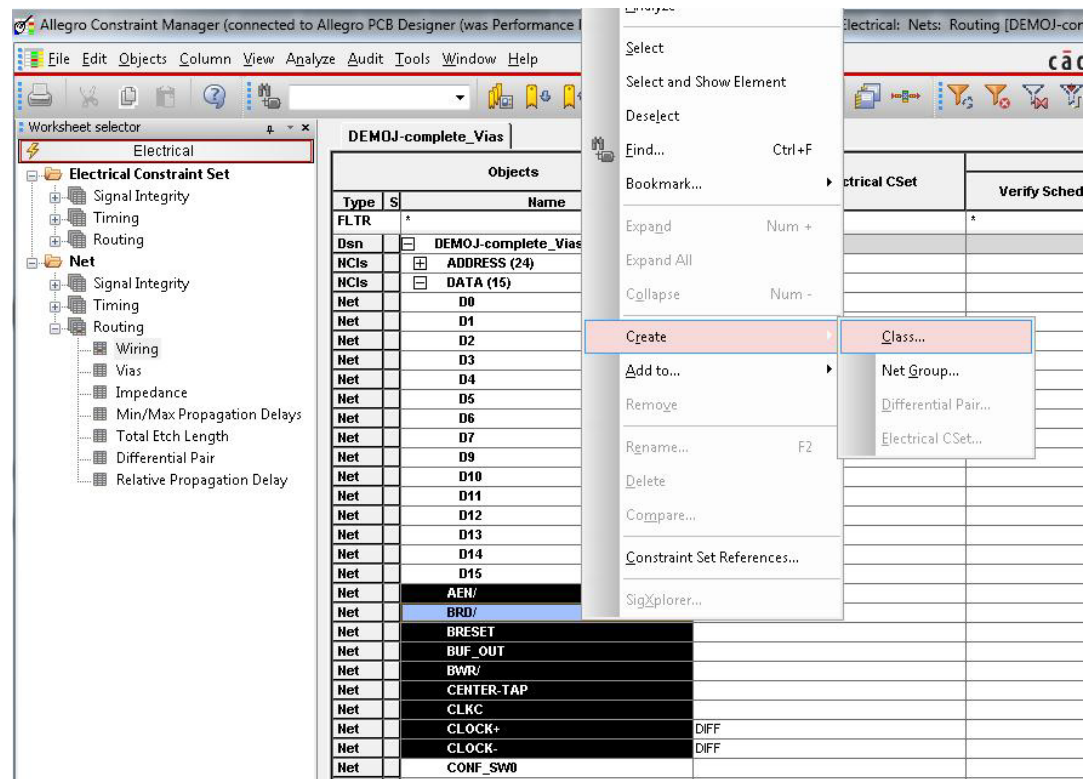
Fig. 17.1 - Creating net class in OrCAD PCB Editor

Class structure can be viewed in *Constraint Manager* only, which can work concurrently with OrCAD PCB Editor.

There is no concept of component class in OrCAD. You can create a room using the special **ROOM** property, which can be assigned to selected components in OrCAD Capture or in the *Constraint Manager*, in the **Properties** section (Fig. 17.2).

Fig. 17.2 - The Room property in Constraint Manager

In Altium Designer, creating net and component classes is important. Iit simplifies management and visualization of the large groups of objects, and it allows you to assign design rules more selectively, taking into account logical division of circuit stages and interfaces. The easiest way to create a net class in Altium Designer's PCB Editor is using the **Add Class** command from the context menu of the **Classes** section in the *PCB* panel. In the dialog that appears specify the name of class, then move nets from the left list to the right one to include them in this class (Fig. 17.3).
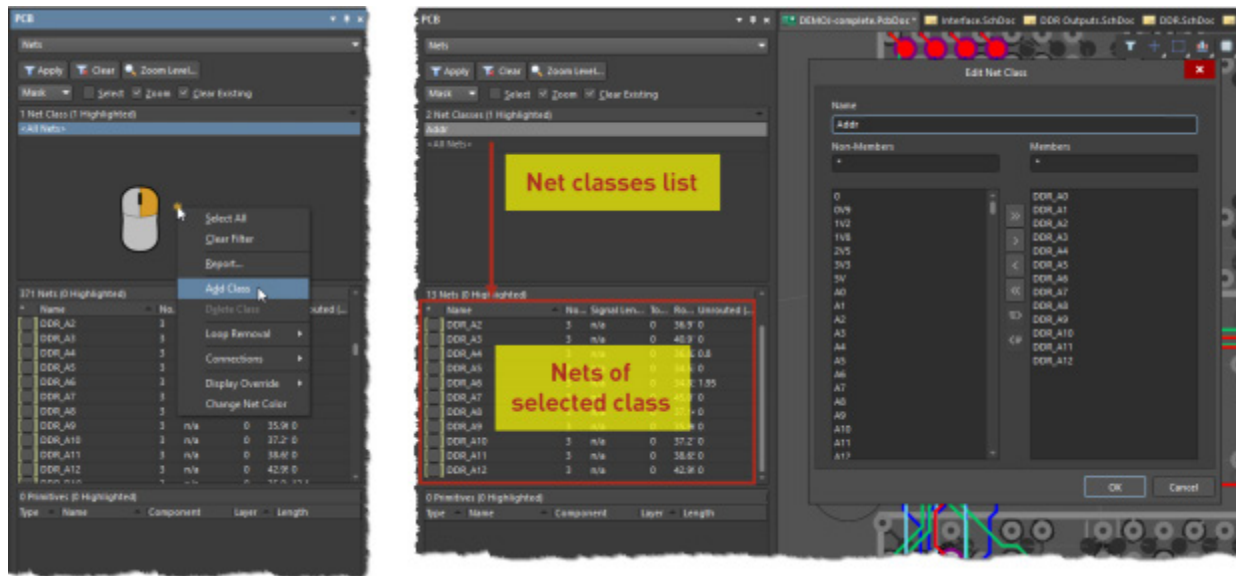
Fig. 17.3 - Creating a net class via the PCB panel in Altium Designer

Component and other object classes are created via the PCB panel in similar way. Altium Designer has the special *Object Class Explorer* dialog (**Design » Classes**) for class management. Here you can create a class for any object type. Once created, it is available in the *PCB* panel (Fig. 17.4).
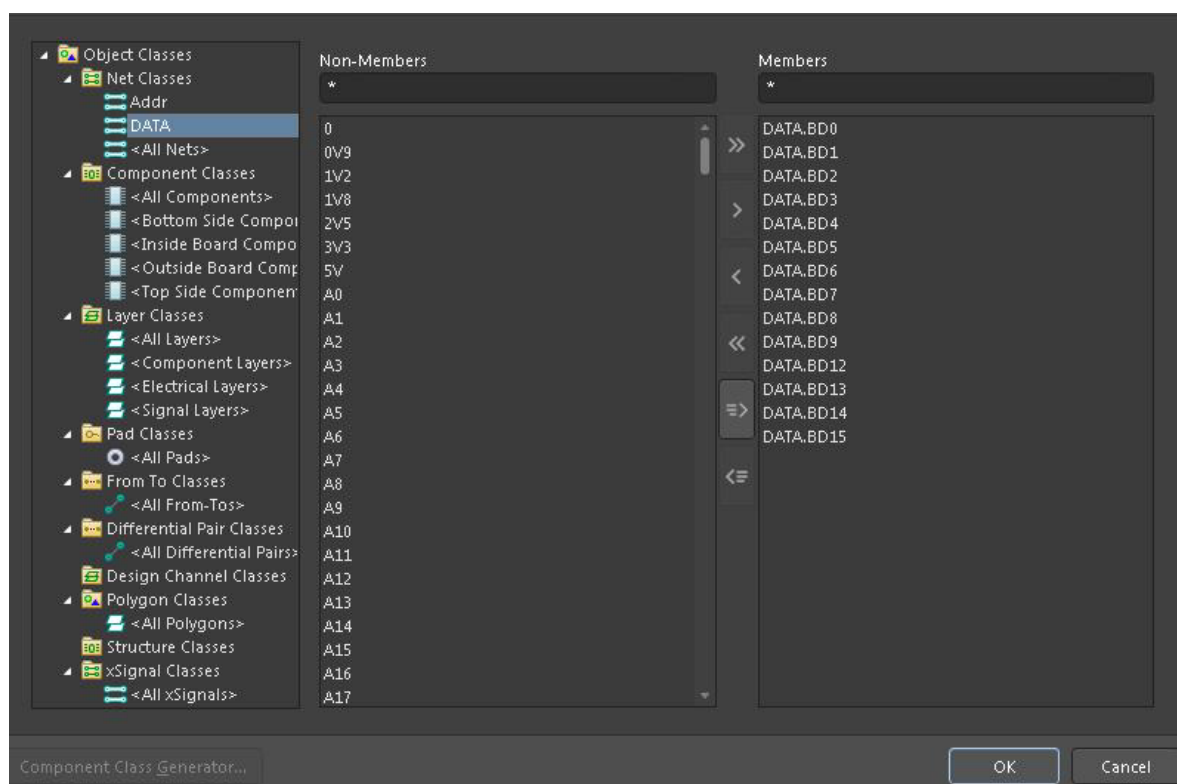
Fig. 17.4 - The Object Class Explorer dialog

Differential pair creation

In OrCAD PCB Editor, a differential pair can be created either in the editor itself using the **Logic » Assign Differential Pair** command, or via the *Constraint Manager.* Simply select two nets and run the **Create » Differential Pair** command from the context menu (Fig. 17.5).
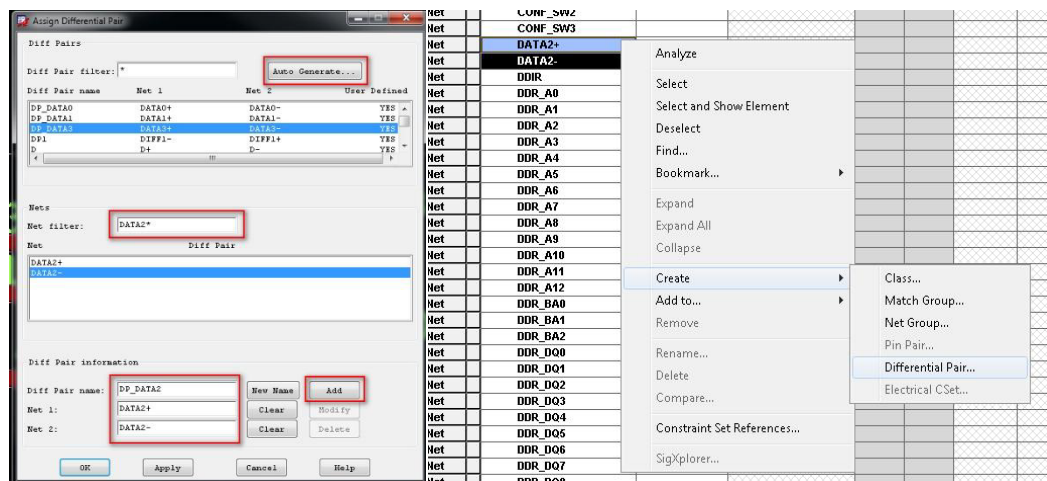
Fig. 17.5 - Creating a differential pair in OrCAD PCB Editor and in the Constraint Manager

You can create multiple differential pairs using the **Objects » Create Differential Pair** command in the *Constraint Manager* and the **Auto Setup** button (Fig. 17.6).
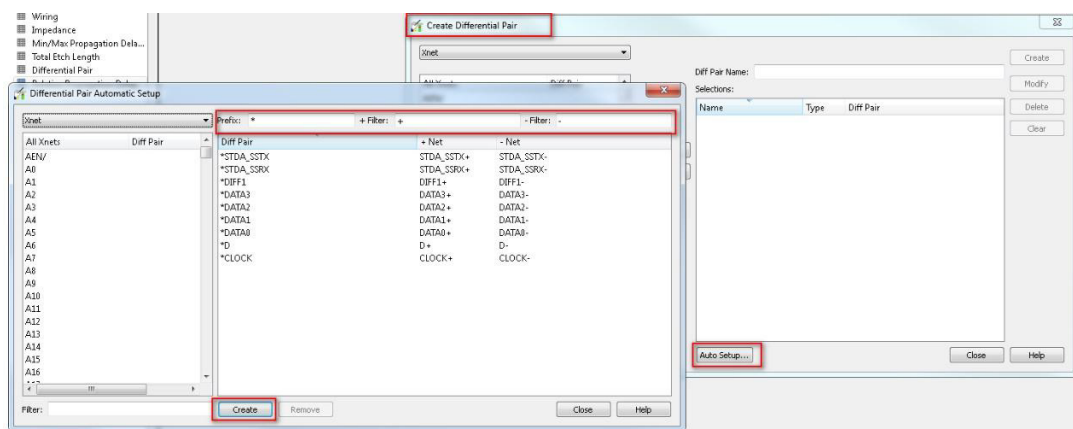
Fig. 17.6 - Creating differential pairs automatically in the Constraint Manager

You can do this in a similar manner in Altium Designer. The easiest way to create differential pairs is using the *PCB* panel. At the top of the panel, select **Differential Pair Editor** mode. You can automatically create all differential pairs using the **Create From Nets** button and the corresponding filter for nets (Fig. 17.7).
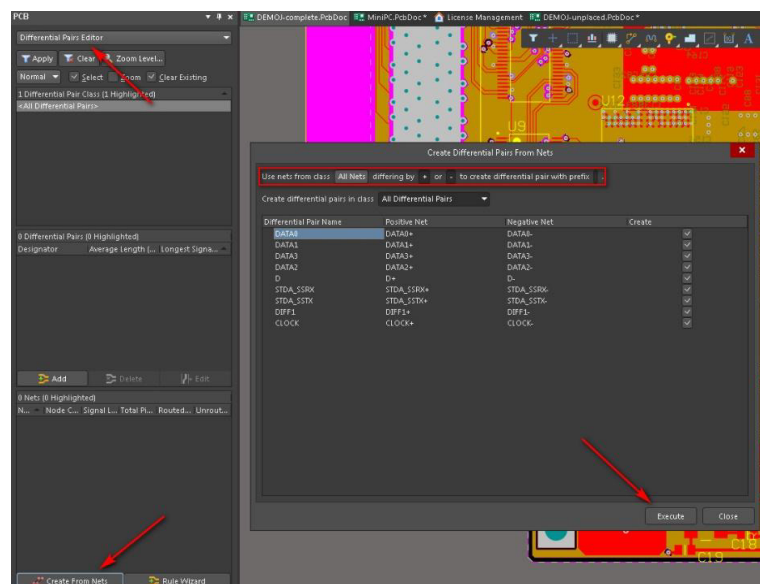
Fig. 17.7 - Creating differential pairs automatically in Altium Designer

From *PCB* panel you can create a differential pair manually using the **Add** button (Fig. 17.8).
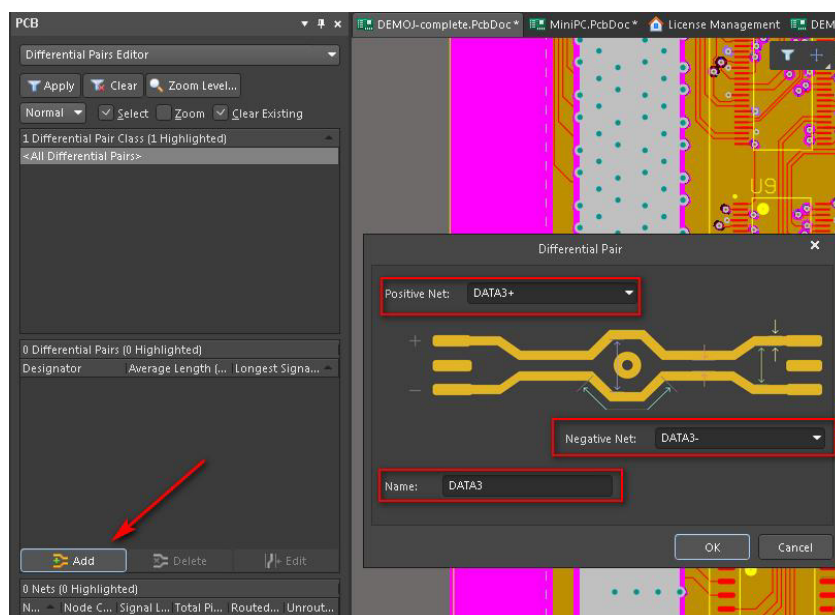
Fig. 17.8 - Creating a differential pair manually in Altium Designer

Extended nets creation

An extended net is a net that includes the entire signal path from a source IC through discrete passive termination components to a destination IC. For example, the address bus of the DDRx interface can act as a set of extended nets.

To create such nets in OrCAD you need to assign so-called ESpice models to the passive components. This can be done in several ways. The simplest way is using the *Signal Model Assignment* dialog (**Analyze » Model Assignment**), where you can select passive components on the board, click on the **Create Model** button and select the **Create ESpiceDevice** model option (Fig. 17.9).
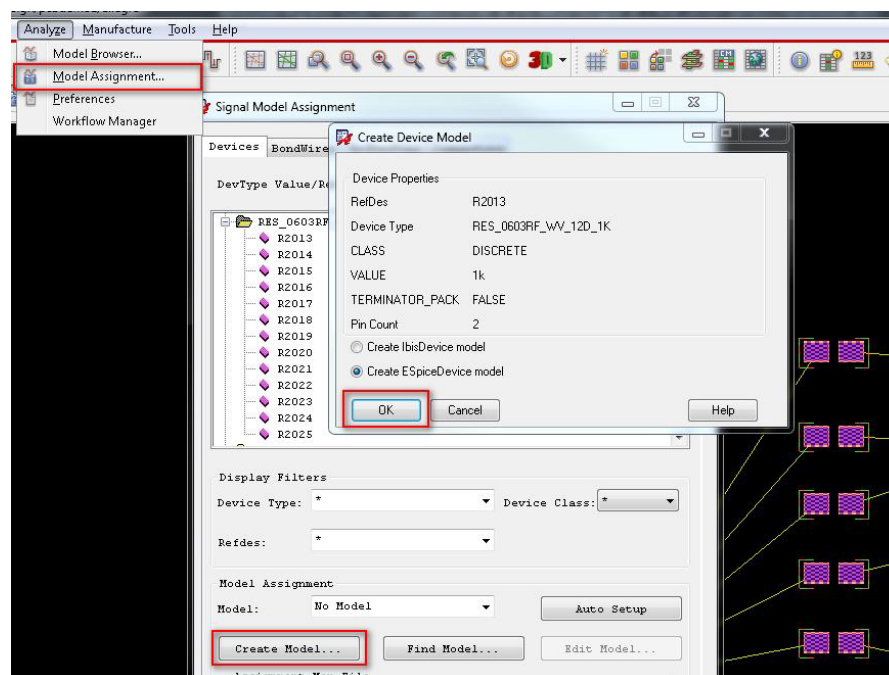
Fig. 17.9 - Creation of ESpice models in OrCAD PCB Editor

Nets that connect those passive components are automatically considered by the software as XNet. The XNet list can be displayed in the Constraint Manager (Fig. 17.10).
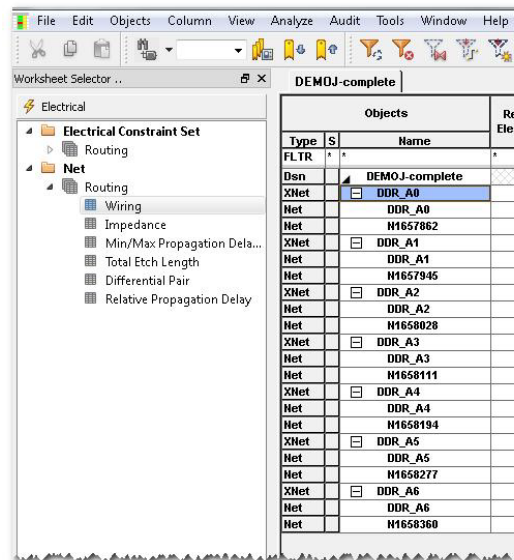
Fig. 17.10 - The XNet list in the Constraint Manager

You can also use the OrCAD PCB Editor's design setup wizard, which includes XNet configuration, using the **Setup » SI Design Setup** menu (Fig. 17.11).
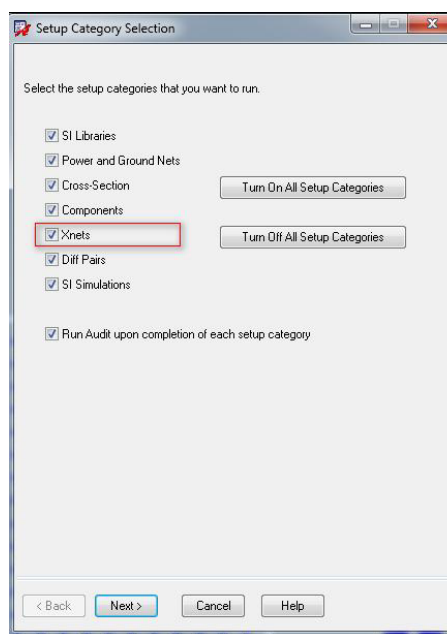
Fig. 17.11 - Design setup wizard in Setup » SI Design Setup menu in OrCAD PCB Editor

In Altium Designer, such extended nets are called xSignals. You can use several methods for creating these nets. Using the *PCB* panel, you can also manage xSignals and their classes (Fig. 17.12).
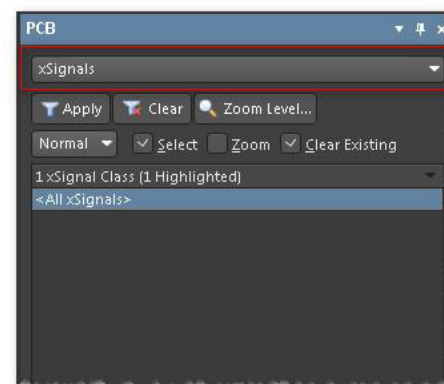
Fig. 17.12 - The xSignals mode of PCB panel in Altium Designer

To create xSignals you can select a pad, press **Tab** to select the entire net, right-click and choose the **xSignals » Create xSignal from selected pins** command from the context menu (Fig. 17.13).
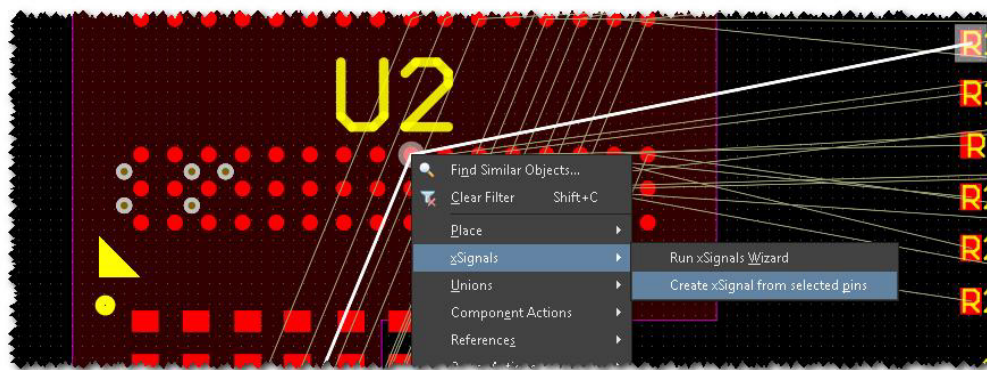
Fig. 17.13 - Manual xSignals creation in Altium Designer

To create a group of extended nets, you can use the *Create xSignals Between Components* dialog (**Design » xSignals**), where you can select signal source ICs, nets and destination IC. Press the **Analyze** button to proceed with creating xSignals based on the selected criteria, for example, the **Through 1 series component** (Fig. 17.14).
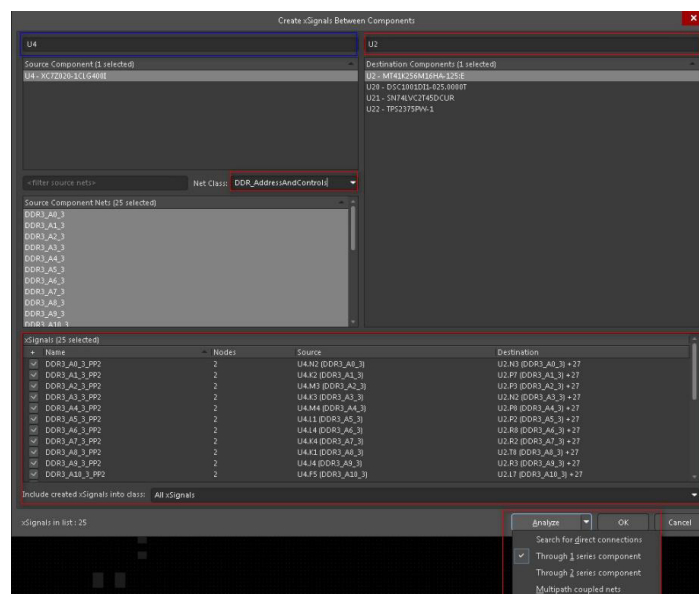
Fig. 17.14 - Creating xSignals via the Create xSignals Between Components dialog

You can access extended net creation wizard using the **Design » xSignals » Run xSignals Wizard** command. It provides a convenient way for creating xSignals with already assigned signal delay constraints for the DDR3/DDR4 and USB 3.0 interfaces, what can save you time significantly (Fig. 17.15).
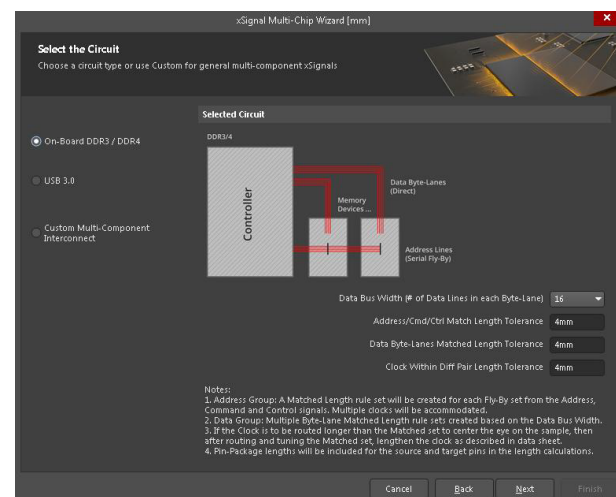
Fig. 17.15 - Creation xSignals with xSignals Wizard in Altium Designer
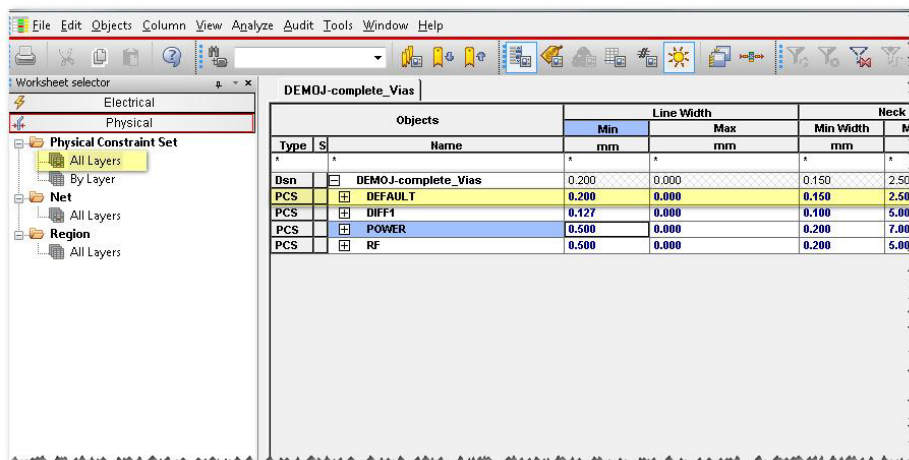
18. Defining design rules

This section covers defining and configuring of basic design rules.

General approach to defining constraints

In OrCAD PCB Editor, rules and constraints can be set using a dedicated editor, Constraint Manager, which can be accessed via the Setup » Constraints » Constraint Manager command. This is a table-based editor, where all the rules are divided into groups, which are displayed on the Worksheet Selector panel. The approach to setting rules in OrCAD is as follows:

1. A Constraint Set is created in each rule group (domain). The Default Constraint Set targets all design nets. You can create additional sets as required (Fig. 18.1).
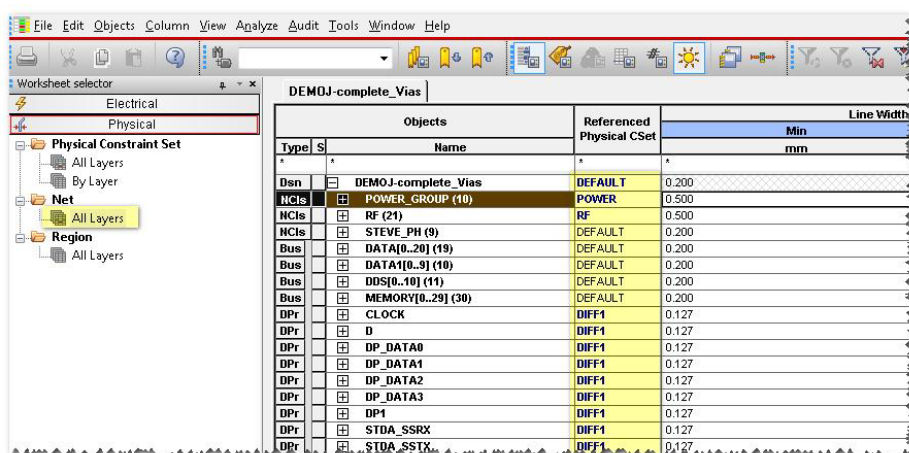
Fig. 18.1 - The Physical rule type in Constraint Manager

2. In the Net section, you can assign a Set for any nets and net objects (Fig. 18.2).

Fig. 18.2 - Constraint Set assignment to net objects

Rules can be assigned to each net individually by typing appropriate values in grid cells. Values defined by user highlighted with blue bold text.

Altıum ®

In Altium Designer, the *PCB Rules and Constraints* Editor dialog (**Design » Rules**) is used. The left-hand side of the dialog lists rule groups, the right-hand side lists all rules and their values (Fig. 18.3).
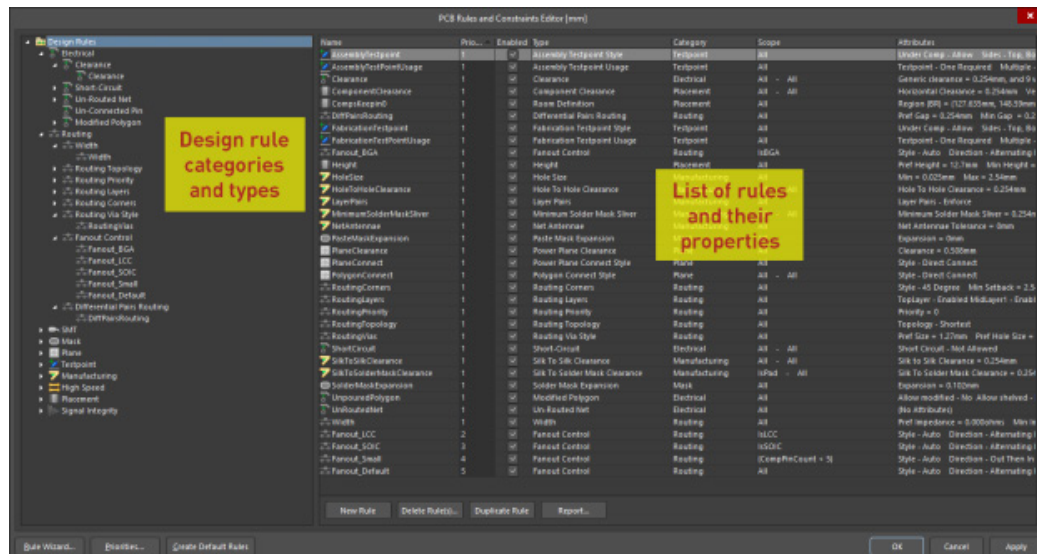

Fig. 18.3 - The PCB Rules and Constraints Editor dialog in Altium Designer

If you select, for example, the **Electrical** rule group, the rule list of this group will be displayed on the right-hand side. If you go to the **Clearance** subgroup directly, you see all the clearances rules from this subgroup. Here you can define the gaps between any net objects on the design. Right-click on the **Clearance** subgroup and select the **New Rule** command from the context menu. New rule will be added to the list. Specify its name according to its parameters so that you will be able to understand for which net group it is defined in the future.

Note that each new rule is illustrated with an image, which simplifies its understanding.

Set **Minimum Clearance** to, for example, 0.3mm. All cells in the grid below the image will be updated with 0.3mm value. A cell at the intersection of a row and a column of the grid defines the gap between objects that are indicated in these row and column names. You can click and drag cursor to select multiple cells and type a new value for them. This approach allows you to fine-tune constraints (Fig. 18.4).
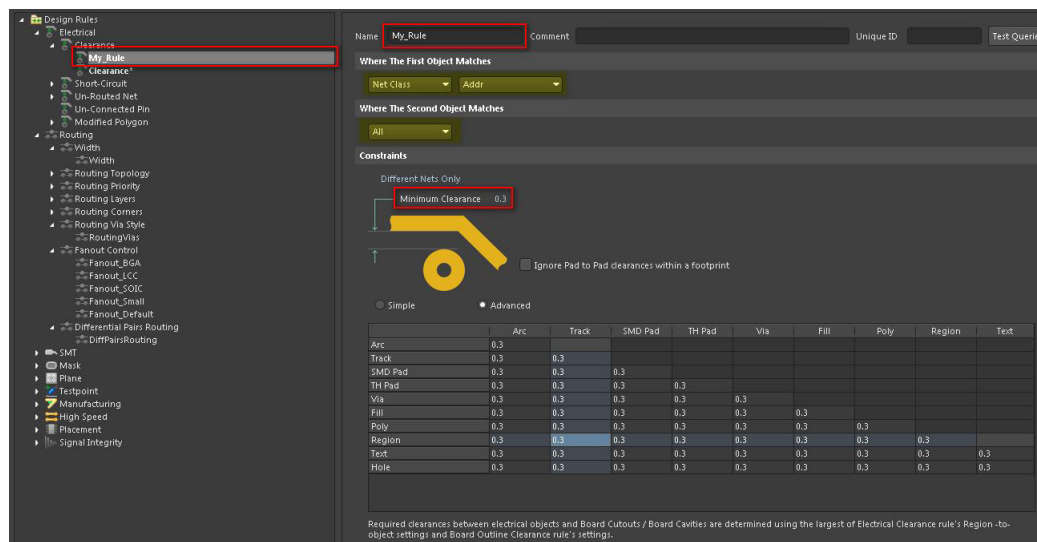

Fig. 18.4 - The Minimum Clearance rule in Altium Designer

You can define a more complex rule, that targets, for example, only polygons of a net on a layer.

Defining rules in schematic editor

In OrCAD Capture, you can define constraints in the property editor for applying them while routing. When transferring data to the PCB editor, the constraints are reflected in the *Constraint Manager* table, where you can modify them (Fig. 18.5).

Fig. 18.5 - Defining net constraints in the OrCAD Capture's property editor

In Altium Designer's SCH Editor, you can define design rules using special labels (directives) from the **Place » Directives » Parameter Set** menu. You can add one or more rules to a directive in the **Rules** section of the *Properties* panel. Then you simply assign directive to a net and thereby assign routing constraints to this net (Fig. 18.6).
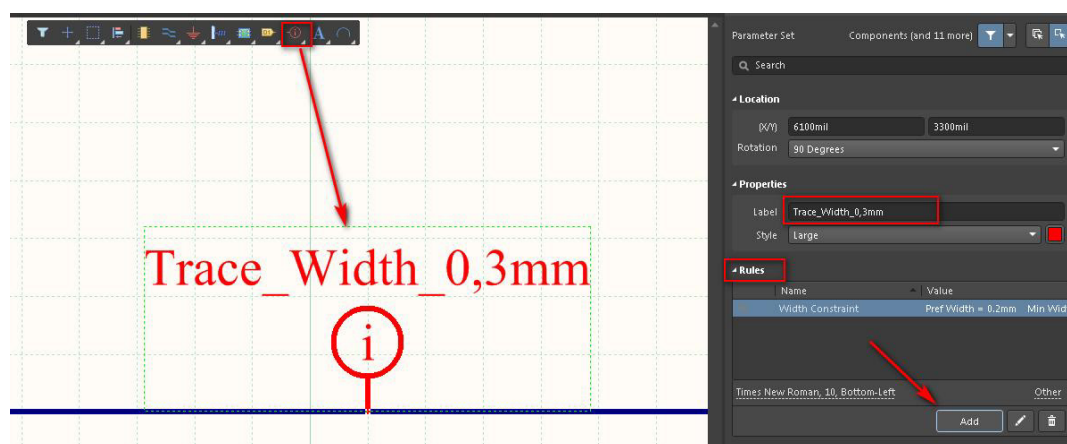
Fig. 18.6 - Directive with a rule for PCB design in Altium Designer's SCH Editor

You can also set a rule for a group of nets. To do this, you can place a blanket (**Place » Directives » Blanket**) over a collection of net labels, then place a directive with defined rules on this blanket (Fig. 18.7).
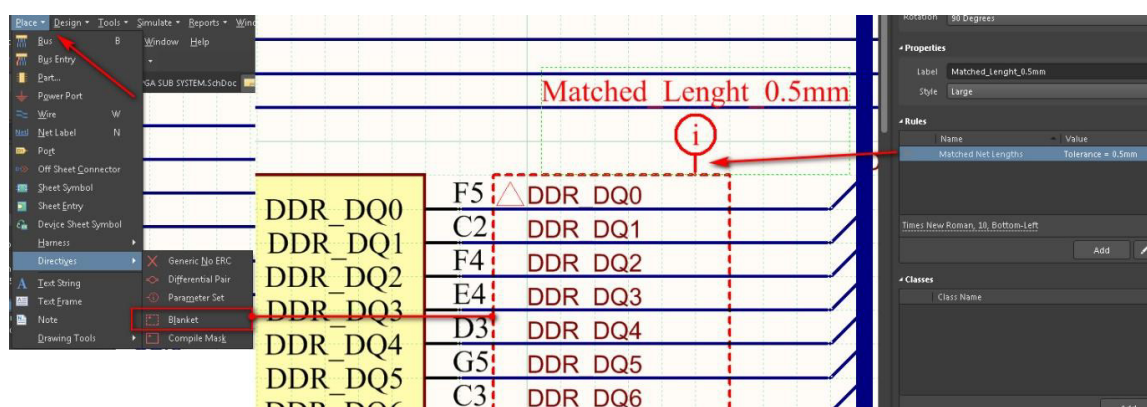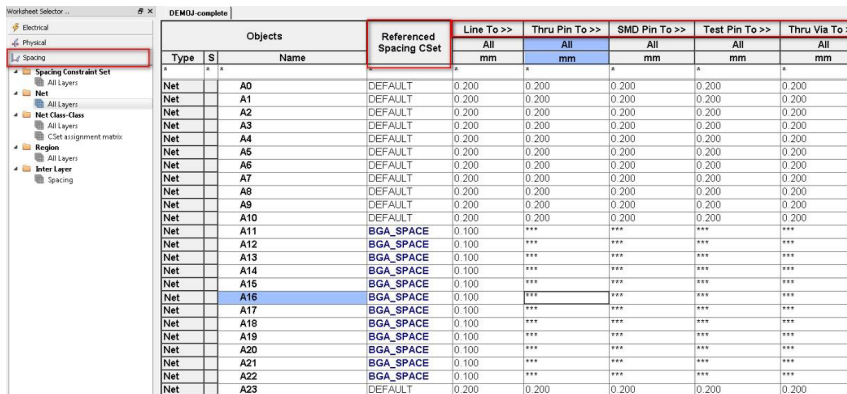

Fig. 18.7 - Defining rules for a group of nets in Altium Designer's SCH Editor

Defining basic rules

1. Clearance rule

In OrCAD PCB Editor, you can define clearance rules in the **Spacing** rules group in the *Constraint Manager* (Fig. 18.8). You can do this using spacing constraint set (SCSet), assign it individually (Fig. 18.9), specifying it in the properties of a specific net or object, for example, a polygon.

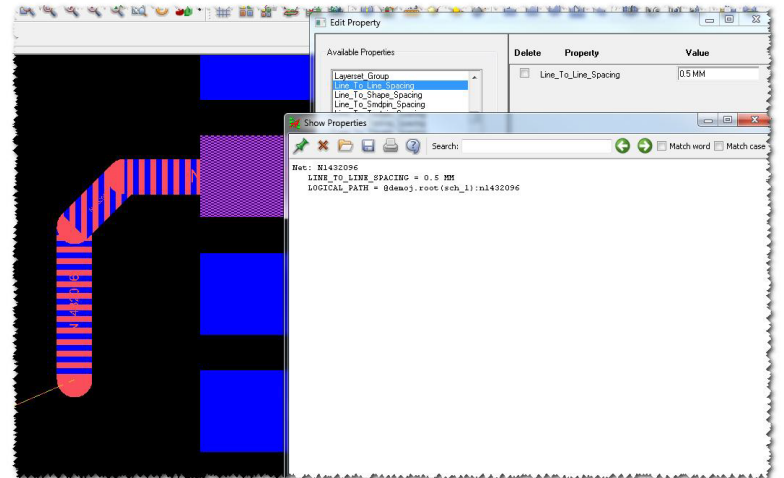Fig. 18.8 - Spacing rules in the Constraint Manager

Fig. 18.9 - Individual property with clearance definition in OrCAD PCB Editor

Example of a clearance rule definition was described above, so here a more complex rule example is described here.

Set the **Where The First Object Matches** field to Custom Query from the associated drop-down, and a text box for defining a query to target specific objects appears. This language is very often used in design software systems. It is very simple, but Altium Designer offers you the methodology by which you do not need to know all language details. Simply click the **Query Builder** button and create a query using this builder (Fig. 18.10).
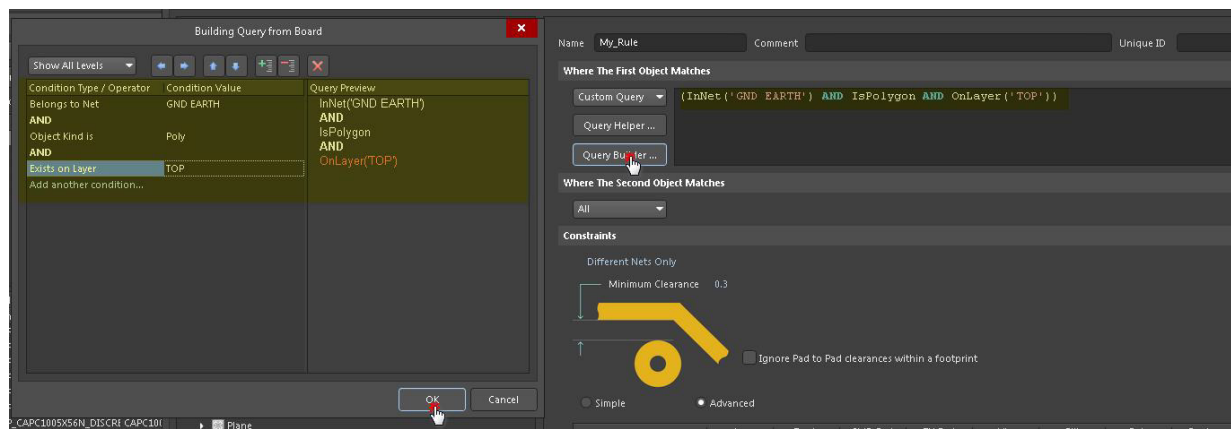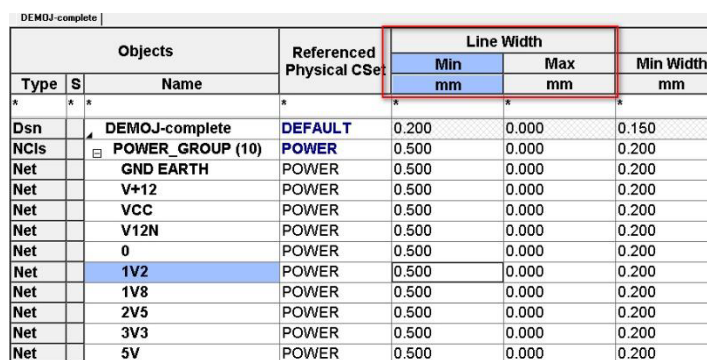

Fig. 18.10 - Creating query for a rule

You can enter query for a rule manually. The text box uses IntelliSense, which greatly simplifies typing, since the software will offer you a correct value and syntax for each next parameter. It is very simple.

2. Routing width and via style rules

In OrCAD, you can define width rules in **Physical** rule group in the *Constraint Manager*, specifying values for minimum and maximum route widths (Fig. 18.11).

Fig. 18.11 - Defining route width rule in OrCAD PCB Editor

Here you can also define via styles. Specify vias in the **Vias** column for a Constraint Set or an individual net object. The **Edit Via List** will appear, where you can choose via stacks from the list on the right, that has been created in *Padstack* Editor (Fig. 18.12).
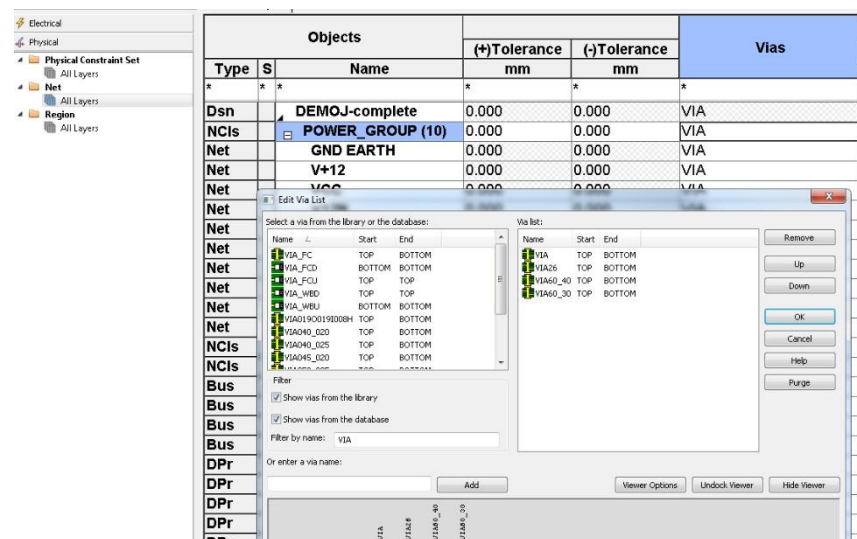

Fig. 18.12 - Defining vias in OrCAD PCB Editor

In Altium Designer, you can define route width rules in **Routing - Width** rule type. You can define preferred, minimum and maximum widths for a net or a net class. You can also define impedance (in ohms) instead of width (Fig. 18.13).
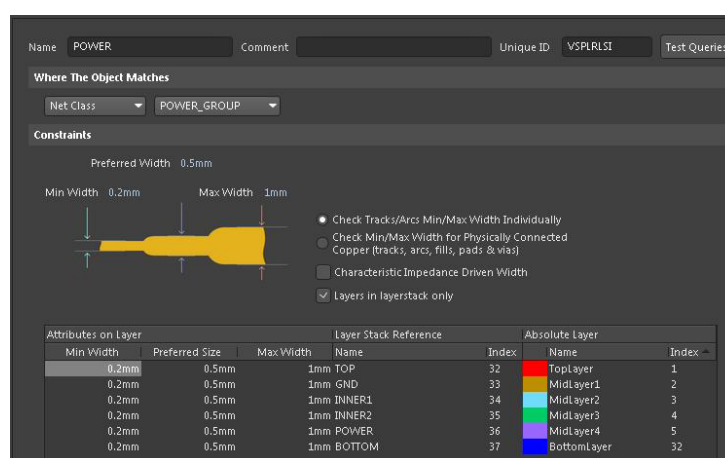

Fig. 18.13 - Routing width rule for the Power_Group net class

3. Solder mask rule

There is no a dedicated rule in OrCAD PCB Editor that constraints solder mask openings for pads. This layer is described directly during padstack creation in *Padstack* Editor. When you generate Gerber files you simply combine all Soldermask layers in one Gerber file and convey it to your manufacturer.

In Altium Designer, the **Mask - Solder Mask Expansion** rule type is used for constraining solder mask. In this rule you can define solder mask openings for entire PCB or for a group of pads. This eliminates the need to set a mask in the padstacks during creating a footprint library, although you have such ability (Fig. 18.15).
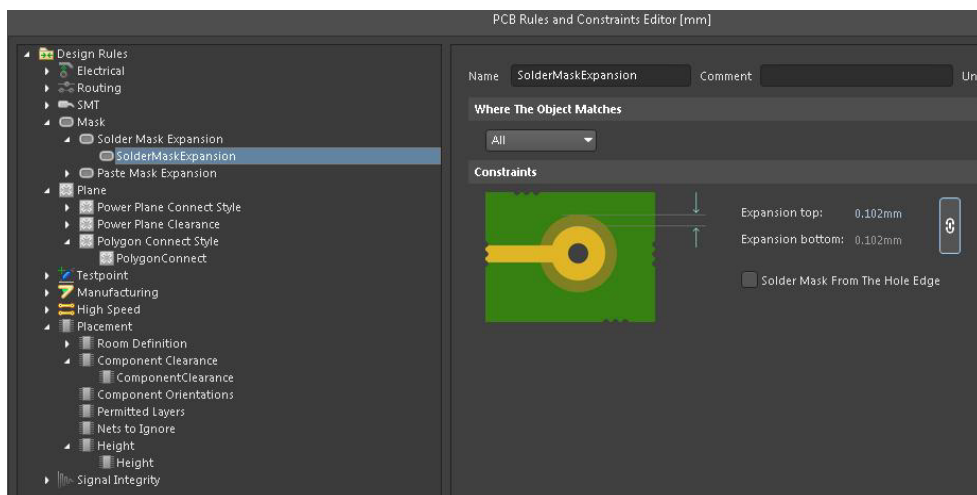

Fig. 18.15 - Defining solder mask rule in Altium Designer

Configuring design rule check modes

Design rule check modes are set up in the *Analysis Mode* dialog (**Analyze » Analysis Modes**) in the *Constraint Manager*. You can set online DRC and enable/disable any check (Fig. 18.16).
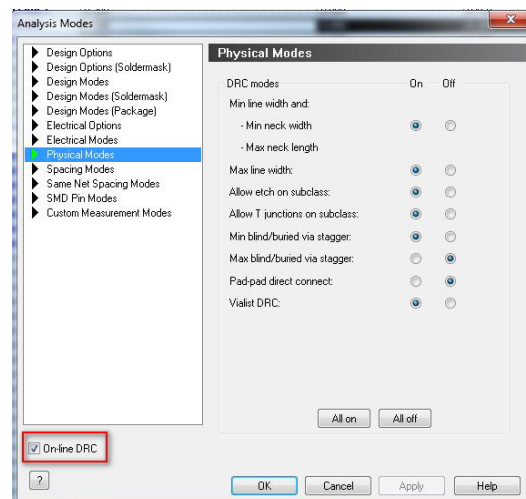

Fig. 18.16 - Configuring DRC modes in Constraint Manager

In Altium Designer, DRC modes are set in the *Design Rule Checker* dialog (**Tools » Design Rule Check**) in the **Rules To Check** section. As well as in OrCAD, in Altium Designer you can check in online and batch mode (Fig. 18.17).
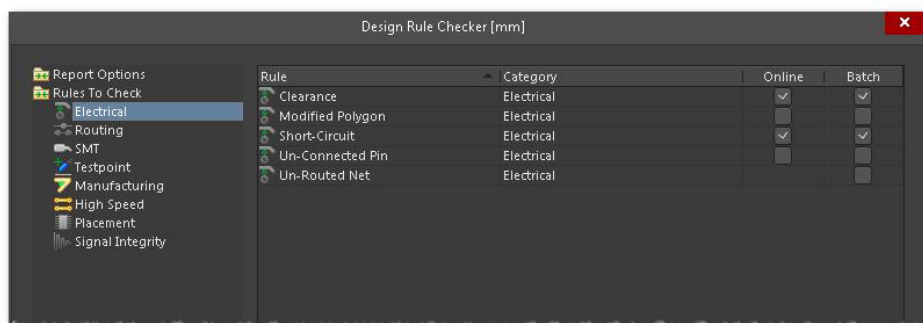

*Fig. 18.17 - Configuring DRC modes in Altium Designer's PCB Editor*

19. Visibility and color of connections

This section covers configuring connections (nets) visibility and colors in PCB Editor

Connection visibility

Layout objects should be shown in a most comfortable for you way. This includes critical net highlighting, creating set of visible layers, transparency settings and so on.

OrCAD PCB Editor has flexible system for controlling layout object visibility. The **Display » Show/Blank Rats** submenu commands are used for changing connection lines (Rats) visibility (Fig. 19.1).
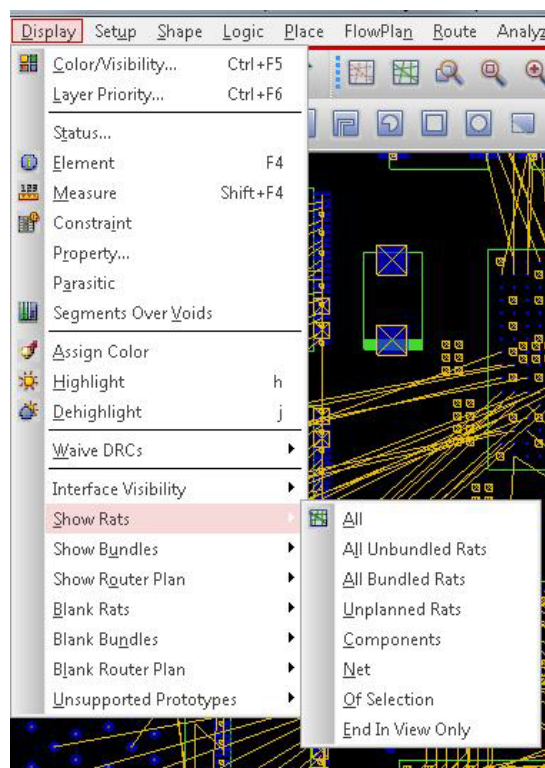
Fig. 19.1 - Connection line visibility options

For example, you can choose **Display » Blank Rats » Components**, then enter desired Reference Designator in the **Find By Name** field of the *Find* panel – rats of this component will be hidden. You can hover the cursor over components or pads, and their rats will be shown/hidden

Let's see how this is handled in Altium Designer.

In the *View Configuration* panel you can change system colors. For connection lines you can change their color or disable their visibility (Fig. 19.2).
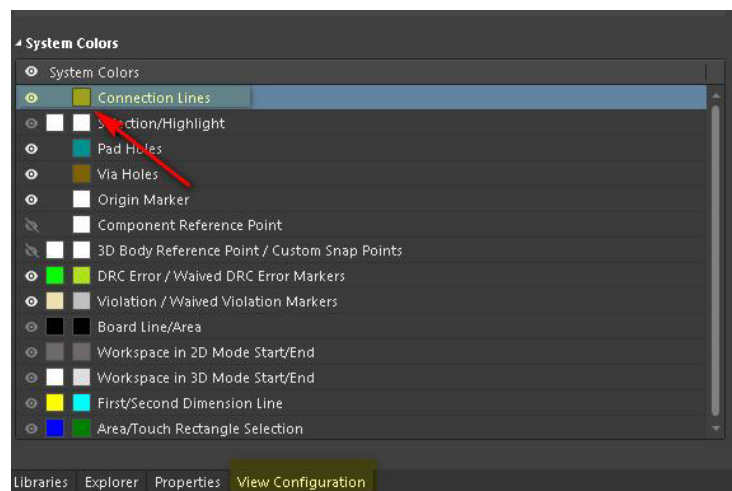

Fig. 19.2 - Configuring visualization of connection lines in the View Configuration panel

You can control visualization of connection lines using **N** shortcut. It gives access to the most frequently used commands when you need to hide or show connections of a net or a component, or connections on the entire board (Fig. 19.3).
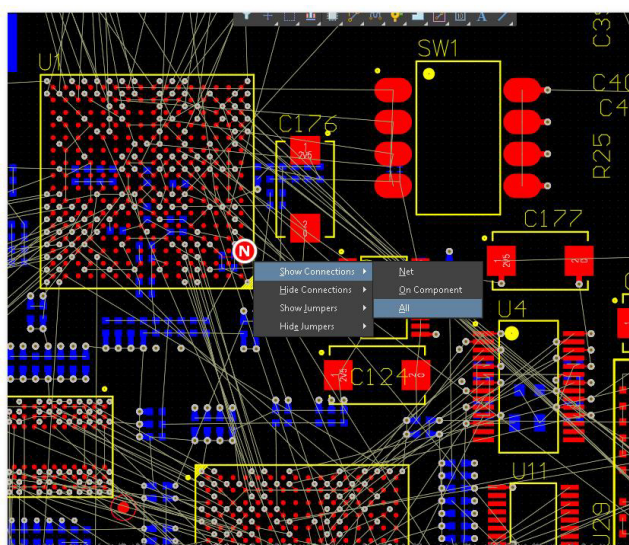
Fig. 19.3 - Visualization configuring using N shortcut in Altium Designer

You can also control the visibility of connection lines through the *PCB* panel. Select a net class or an individual net from the lists, right-click on it and choose the **Connections » Show / Hide** command from the context menu. To change net color you can choose the **Change Net Color** command. Note that the color assigned to the net is displayed in the most left column of the *PCB* panel list (Fig. 19.4).

Fig. 19.4 - Changing visualization of connection lines through the PCB panel

Setting net color

To assign a color to a net hover the cursor over a pad or any other object assigned to this net, right-click on it and choose the **Assign Color** from the contex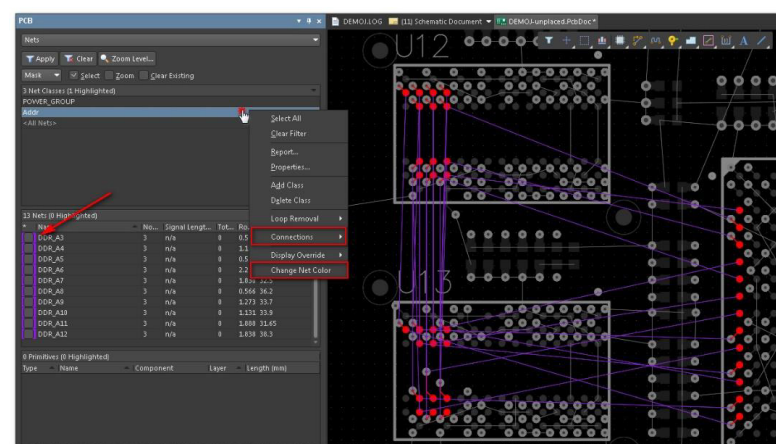t menu and select desired color for the net. You can also use the **Display » Assign Color** command from the main menu and choose color and pattern from the *Options* panel (Fig. 19.5).
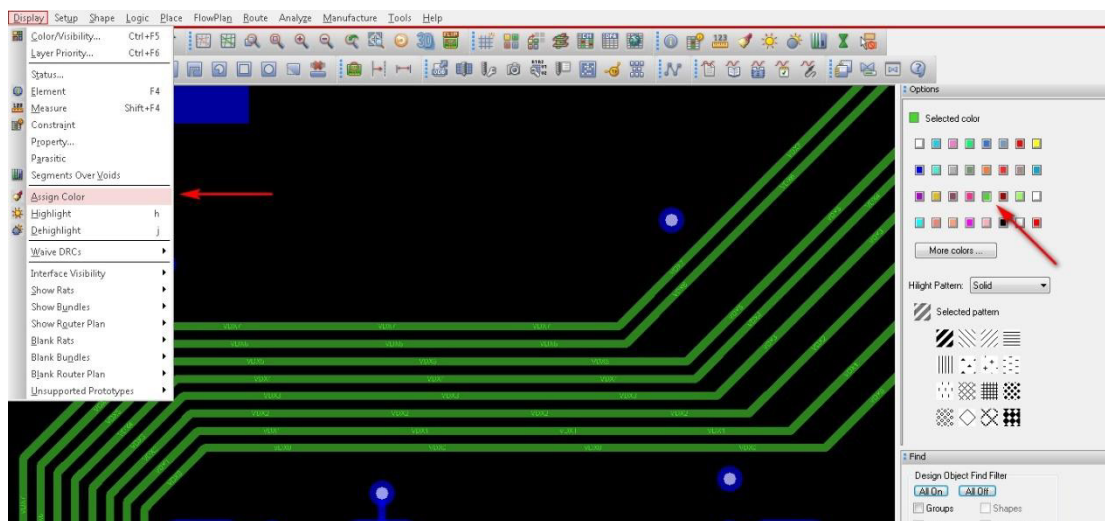
Fig. 19.5 - Setting for the highlighting color in OrCAD PCB Editor

To disable the highlighting use the **Display » Dehighlight** command.

In Altium Designer, check the option in the most left column of the **Nets** list in the *PCB* panel to enable highlighting of this net – traces, pads, vias and other objects of this net will be highlighted with the chosen color. To toggle highlighting of nets press **F5**. To edit the color for this net use the **Change Net Color** command from the context menu of a net entry (Fig. 19.6).

Fig. 19.6 - Setting for the highlighting color for net objects in the PCB panel

20. Routing tools

his section covers routing tools. Altium Designer has modern tools for interactive and automatic routing. Constraint-driven design process is fully supported here.
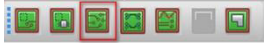
Interactive manual routing

In OrCAD PCB Editor, various routing tools are offered. Performance of commands depends on the mode which you works in. There is special mode for the routing, **Etch Edit**, which can be activated using the corresponding button on the ▮▮▮▮▮ ▮▮ toolbar. Then you can choose a pad or a rat to start routing. All routing properties are displayed in the *Options* panel (Fig. 20.1).
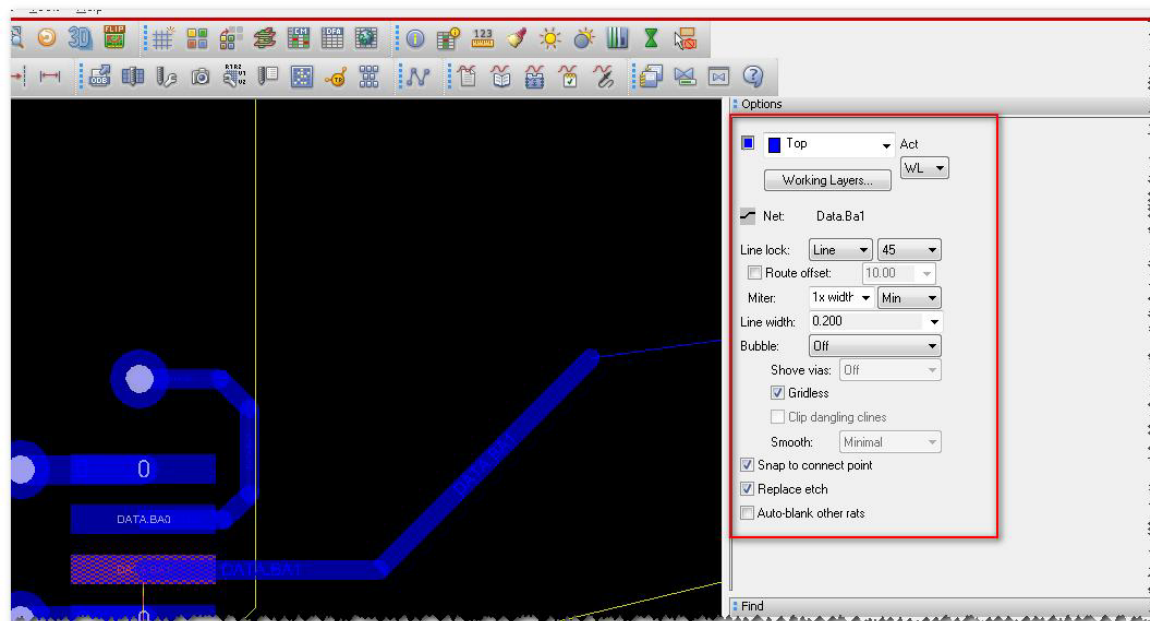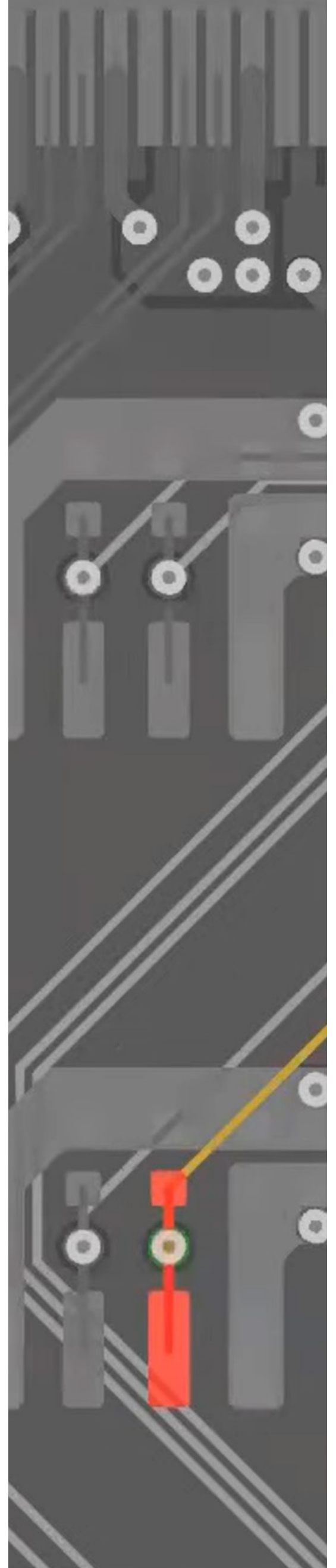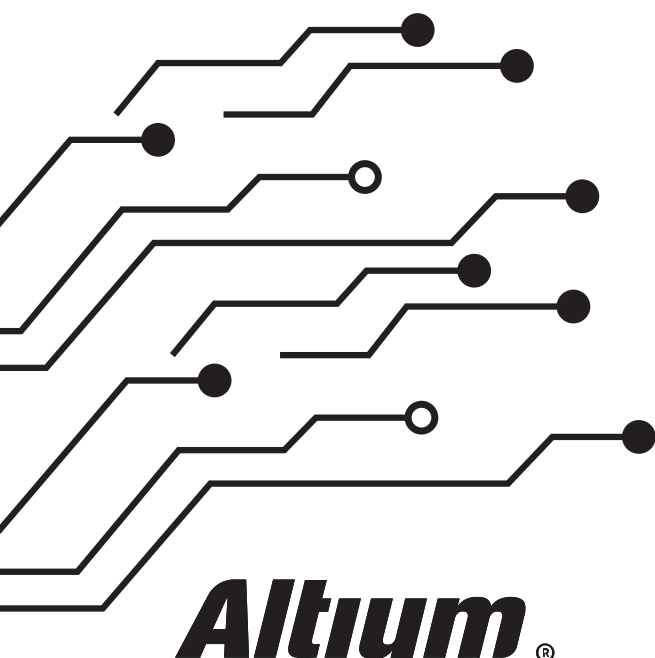

Fig. 20.1 - Interactive routing in OrCAD PCB Editor

There is also **Route » Connect** command (**F3** shortcut). In addition to the *Options* panel, all commands are available from the right-click menu during routing, for example, **Toggle** for turning corner.

In Altium Designer, all routing-related commands are accessed from the **Route** menu. You can assign a keyboard shortcut for a command as required. Interactive routing settings are located in the **PCB Editor - Interactive Routing** page of *Preferences* dialog. It is easier to consider interactive routing features using Heads Up Display (**Shift+H**), where all properties of active command are shown, and the *Properties* panel. Run the **Route » Interactive Routing** command, select a pad of any component and refer to the Heads Up Display (Fig. 20.2).
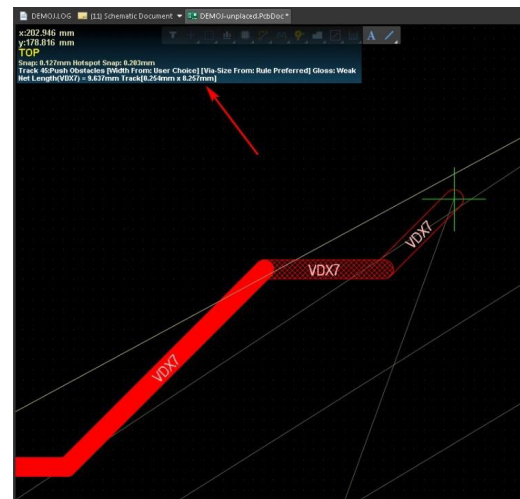

Fig. 20.2 - Interactive routing feedback in the Heads Up Display

When changing a property, for example, trace width, this change will be reflected in the Heads Up Display. Press 3 to change trace width. Currently used value – minimum, preferred, or maximum, that are defined in applicable design rule – is also reflected in the Heads Up Display. There is also a **User Choice** mode, where you can select a width from the *Favorite Interactive Routing Widths* list, accessed with **Shift+W** shortcut.

Using shortcut keys in Altium Designer is crucial. To display list of all available shortcuts, simply press ~ (tilde) key. This is a great hint for new users. Remember that this hint only works when you perform a command.

Here is a list of the most frequently used during interactive routing shortcuts:

- **Spacebar** – toggle between corner direction sub-modes
- **Shift+Spacebar** – cycle through available corner modes
- **Shift+R** – cycle through the currently enabled routing conflict resolution modes
- **3** – cycle through routing width sources
- **Shift+W** – choose track width from predefined favorite routing widths
- **Num +/-** – switch to the next/previous enabled layer, dropping a via
- **4** – cycle through via size sources
- **Shift+V** – choose via size from predefined via sizes
- **Ctrl+Shift+G** – cycle Glossing Effort mode

You can also press Tab key to access to all current routing properties in the *Properties* panel. Note that corresponding shortcuts are shown for each property (Fig. 20.3).
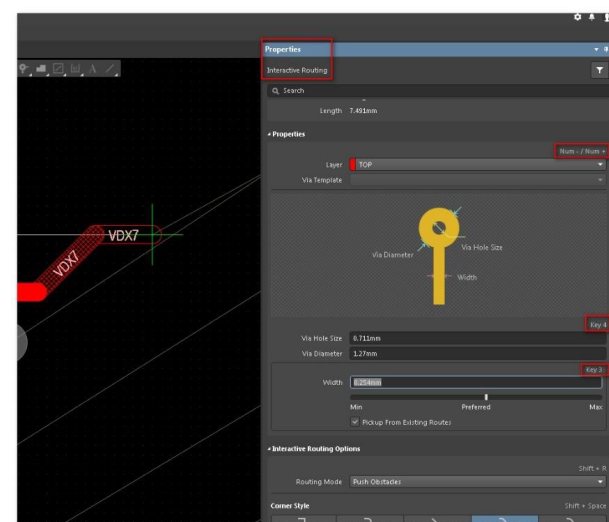
Fig. 20.3 - Interactive routing properties in the Properties panel

Automated interactive routing

In addition to interactive routing, Altium Designer features ActiveRoute, an automated interactive routing technology. To use it, it is recommended to open the PCB ActiveRoute panel. Select a group of connection lines (bus), click on the Route Guide button in the PCB ActiveRoute panel, define the routing path, and click on the ActiveRoute button (Fig. 20.4).

Fig. 20.4 - Automated interactive routing provided by ActiveRoute

21. Working with Polygon Pours

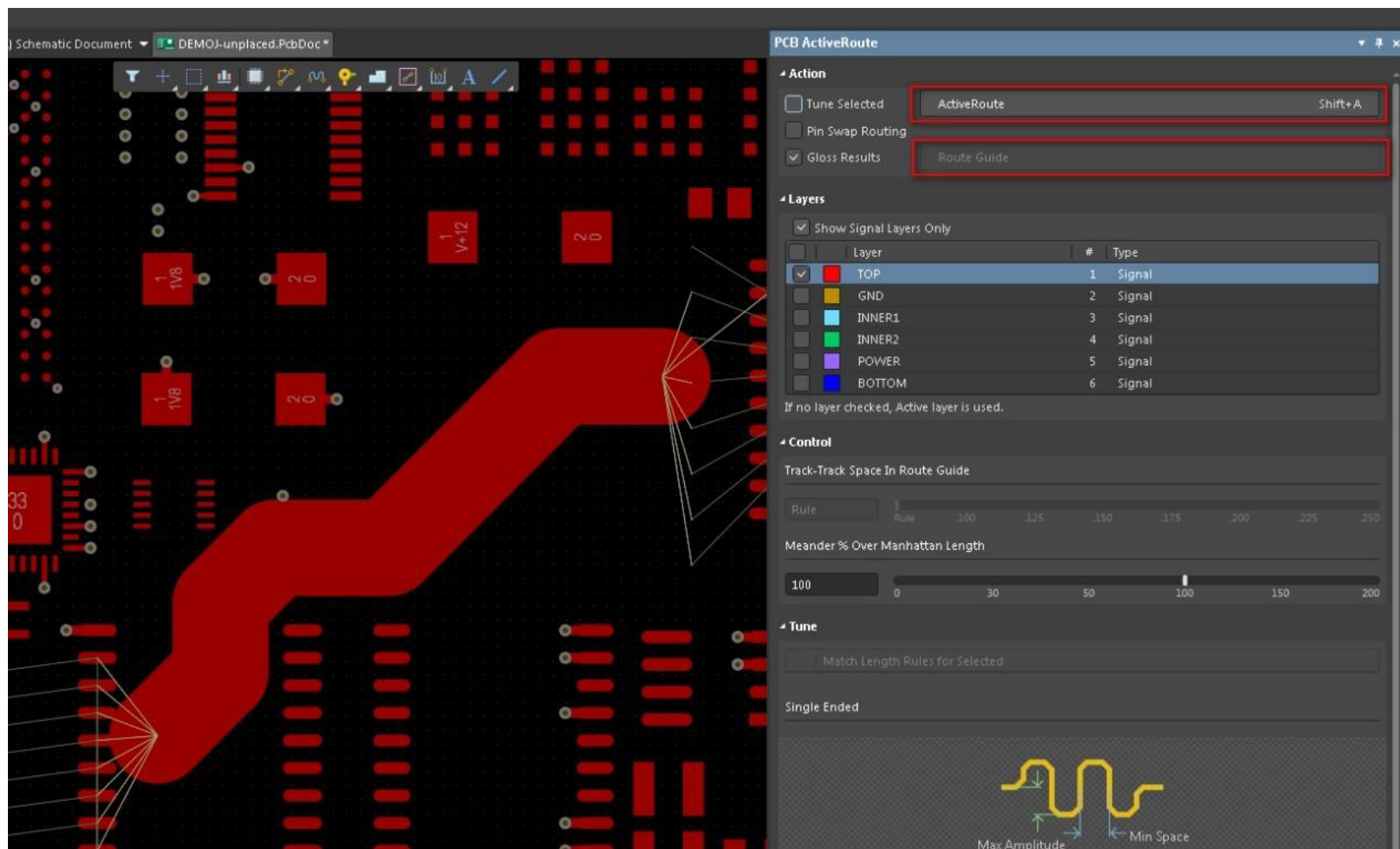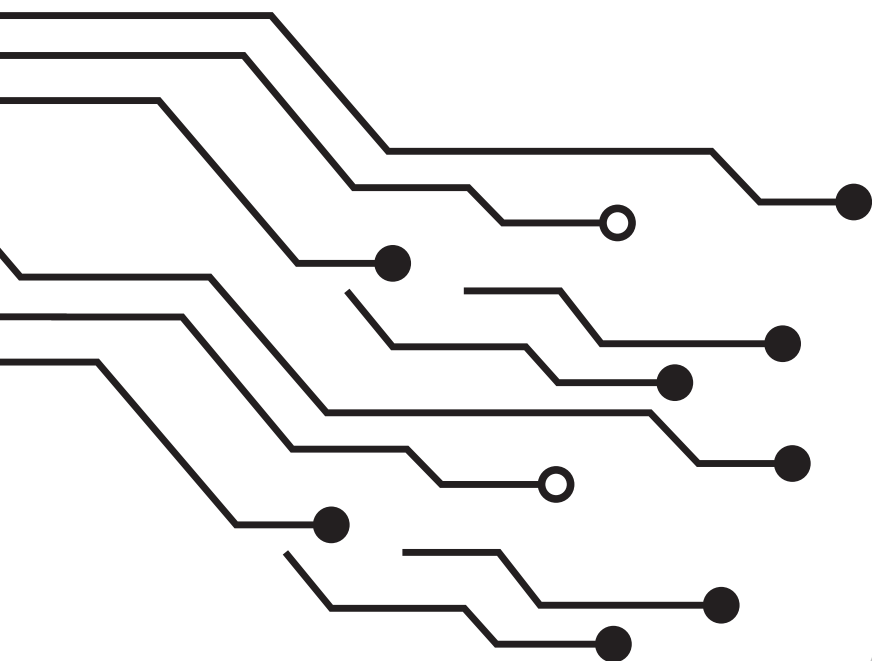*This section covers working with Polygon Pours in Altium Designer. As in all previous cases, designer is provided with a variety of tools for defining polygon fill and shape.*

Polygon connection style for pads and vias

There are two types of polygons in OrCAD PCB Editor: static and dynamic. For signal layers dynamic polygons are used, as they are updated automatically and easier to modify. Static polygons are used to describe all kinds of regions and outlines. In particular, the outlines of Keepin and Keepout areas are static polygons.

Dynamic polygon settings are located in the *Dynamic Shape Instance Parameters* dialog (**Shape » Global Dynamic Parameters**). You can choose Smooth, Rough, or Disabled dynamic fill mode. The Smooth mode is most commonly used as it fully complies with defined constraints. The remaining modes are used when required, for example, in order to reduce the BRD file size, to perform automatic routing (as Allegro PCB Router (SPECCTRA) does not work with dynamic polygons), etc. On the **Clearances** tab, you can define cutout properties for PTH/SMD, Vias, routes and other objects. On the **Thermal relief connects** tab you can define properties for thermal reliefs (Fig. 21.1).
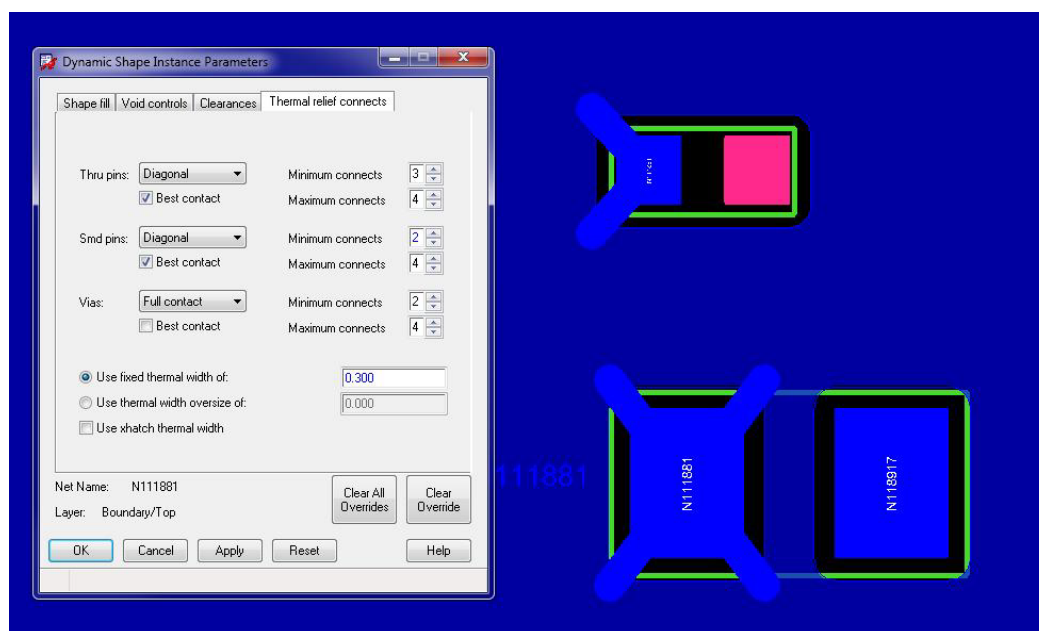

Fig. 21.1 - Thermal relief connect properties for pads and vias

In OrCAD PCB Editor, you can define pad to polygon connection style properties on design rule level (**Spacing** rule type), for individual polygon, or even for individual pad, what greatly simplifies your work. You simply select a pad and go to its properties, where you can specify desired connection type.

In Altium Designer, polygon connection styles are defined by design rules. To edit these rules run the **Design » Rules** command and find the **Polygon Connect Style** rule type in the **Plane** category (Fig. 21.2).
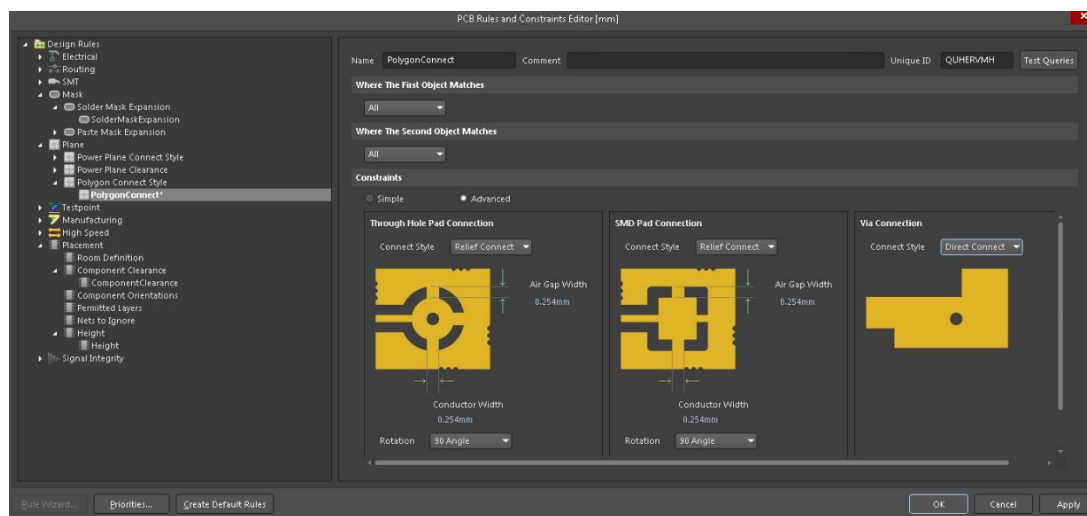

Fig. 21.2 - Polygon connect style configuration in Altium Designer

You can define rules for an individual polygon or individual pads. Use **Query Builder** to create an appropriate query for the rule scope.

Two other rules – **Power Plane Connect Style** and **Power Plane Clearance** – are related to Plane layers. These layers has solid fill by default. In Altium Designer, routing on a plane layer is not allowed.

Editing Polygons

The main controls for accessing polygon creation commands are **Shape** menu and toolbar. In addition to this, you can activate the **Shape Edit** mode to work with polygons. In this mode, the most of polygon editing options are available in the *Options* panel (Fig. 21.3).
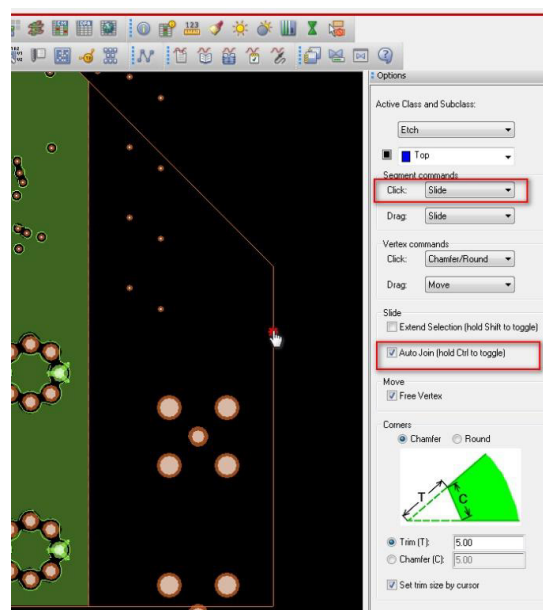

Fig. 21.3 - Polygon editing tools in OrCAD PCB Editor

For example, select the action you want to perform when clicking (**Click**): Slide, Move, or Add Notch. You can choose an action for click and hold (**Drag**). Here you will also find handy tools for working with corners.

Notice the **Edit » Z-Copy** command, which allows you to create an equidistant copy of the polygon on any layer with the **Expand** and **Contact** options. The same commands are located in the context menu, when you hover the cursor on the border of any signal polygon.

In Altium Designer, the **Place » Polygon Pour** command is used for polygon creating. In this menu, you can also find commands for creating cutouts in a polygon (**Cutout**) and slicing a polygon (**Slice**). During polygon drawing, you can use **Shift+Spacebar** and **Spacebar** shortcuts to toggle corner direction and mode, respectively.

All polygon properties are displayed in the *Properties* panel, where you can assign polygon to a net, set fill options, define polygon name or enable the **Auto Naming** option (Fig. 21.4).

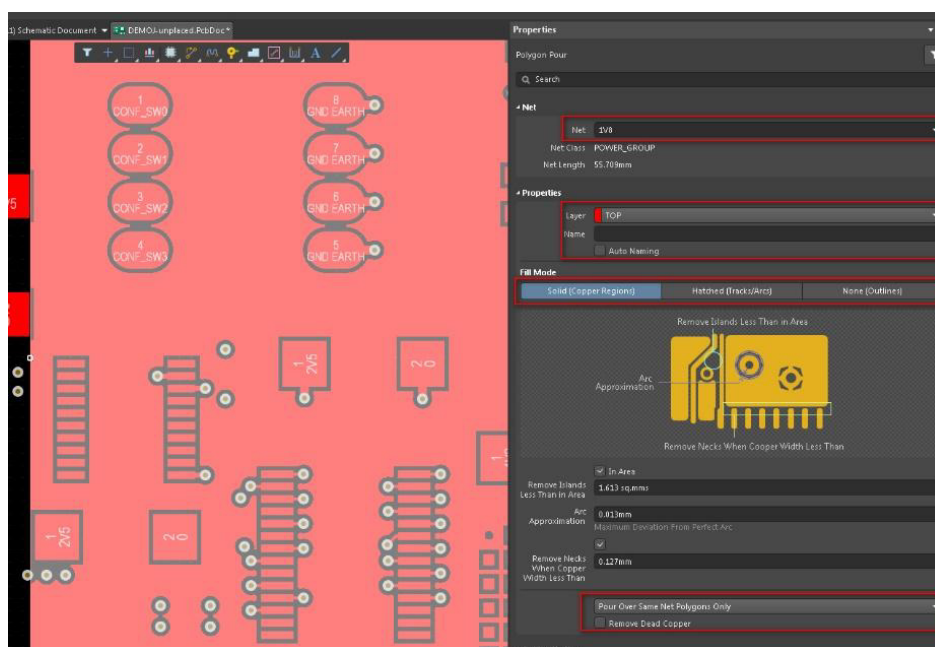*Fig. 21.4 - Polygon properties in the Properties panel*

Polygon editing tools in Altium Designer are quite intuitive. When a polygon is selected, center and corner vertices appear. You can modify polygon size by dragging these vertices. While dragging a vertex press the **Shift+Spacebar** keys to cycle through corner editing modes (Fig. 21.5).
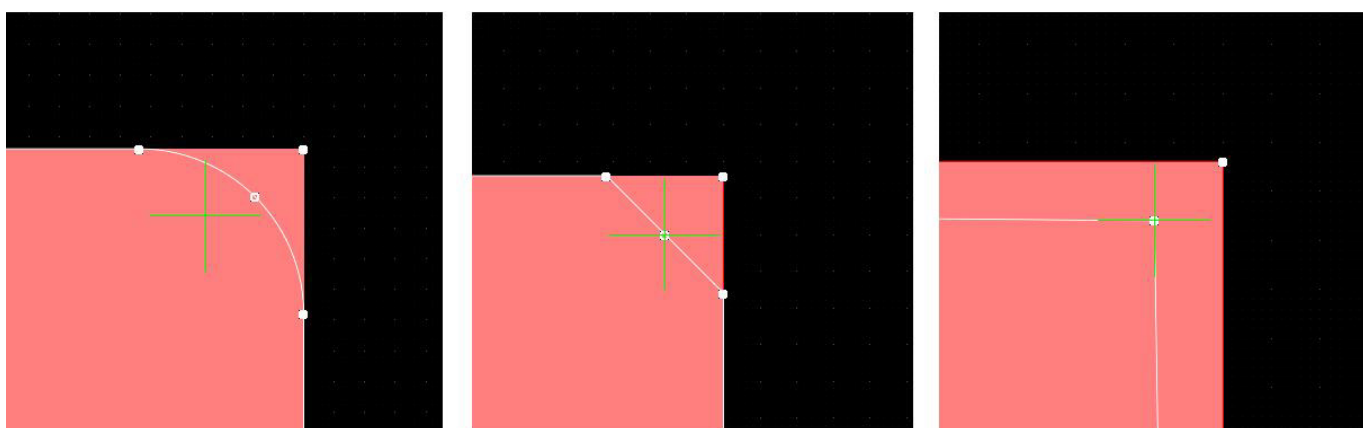

Fig. 21.5 - Polygon editing in Altium Designer

In the **Polygon Actions** submenu of the context menu you can find additional editing commands, such as combining polygons, subtracting polygons and editing polygon border (Fig. 21.6).

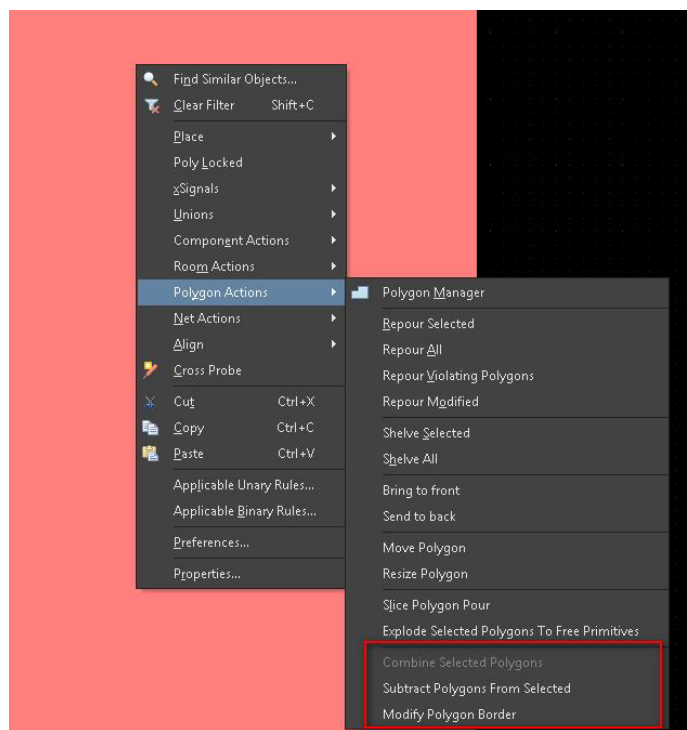*Fig. 21.6 - Additional polygon editing commands in Altium Designer*

Pour priority

In OrCAD PCB Editor, you can change pour priority for any polygon when the **Shape Edit** mode is active or when the **Shape Select** (the **Raise/Lower Shape Priority** entry from the context menu) command is launched (Fig. 21.7).
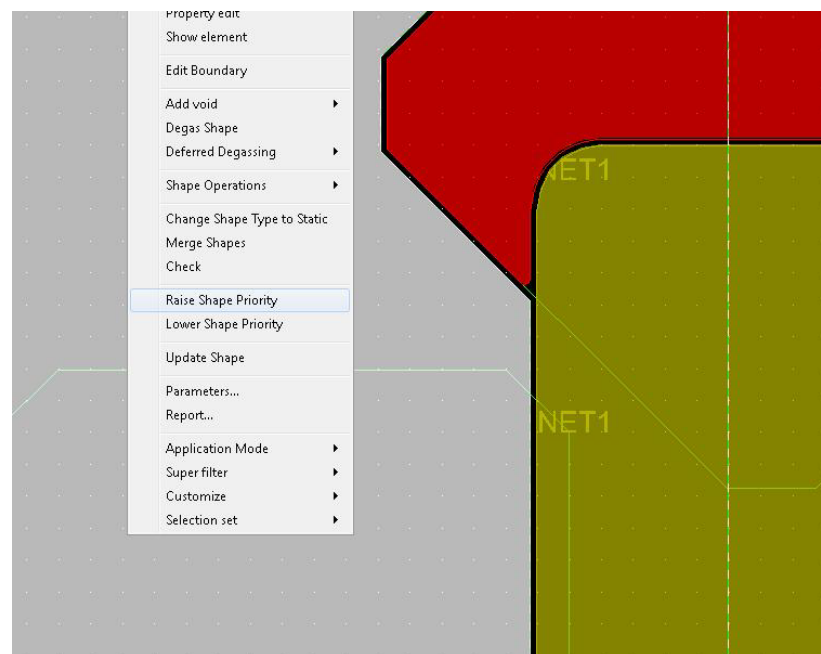

Fig. 21.7 - Changing pour priority for overlapping polygons in OrCAD PCB Editor

In Altium Designer, you can use **Polygon Actions » Bring To Front/Send To Back** commands from the context menu (Fig. 21.8).
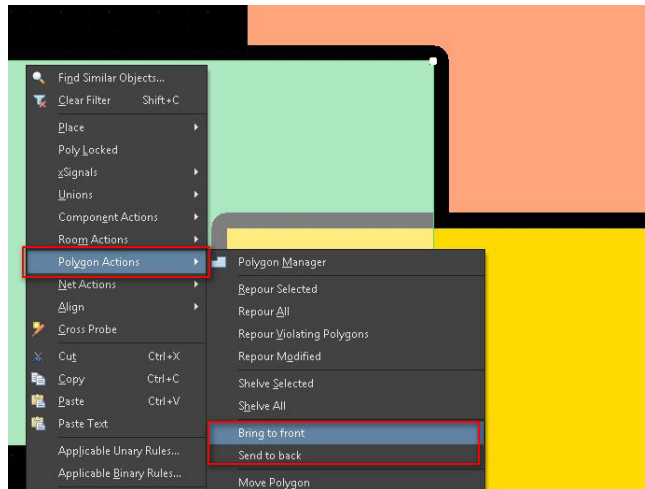

Fig. 21.8 - Pour priority changing in Altium Designer

In Altium Designer, you can use the automatic polygon repouring, just as it does in the OrCAD PCB Editor. To enable it, go to the **PCB Editor - General** of the *Preferences* dialog and enable **Repour Polygons After Modification** and **Repour all related polygons after editing** options (Fig. 21.9).
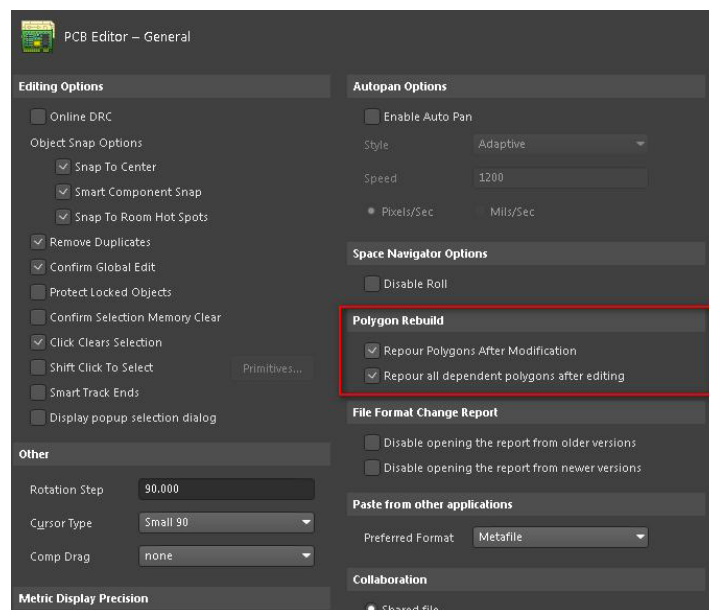

Fig. 21.9 - Polygon rebuild options

Polygon display

In OrCAD PCB Editor, polygon view is configured in the **Display** tab of the *Color Dialog* (**Display » Color Visibility**). Here you can set polygon transparency level (Fig. 21.10).
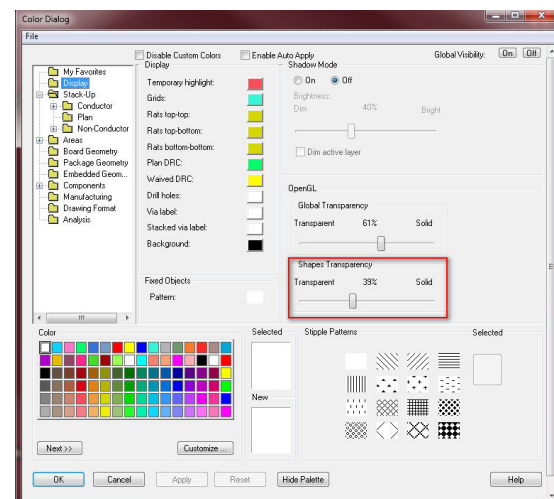

Fig. 21.10 - Adjusting polygon transparency in OrCAD PCB Editor

You also can hide polygons or display only their outlines in the **Boundary** layer class.

In Altium Designer, you can also set the polygon transparency – open the **View Options** tab of the *View Configuration* panel. In the **Object Visibility** section, adjust the transparency level for any objects, including polygons, and/or set the outline (**Draft**) display mode (Fig. 21.11).
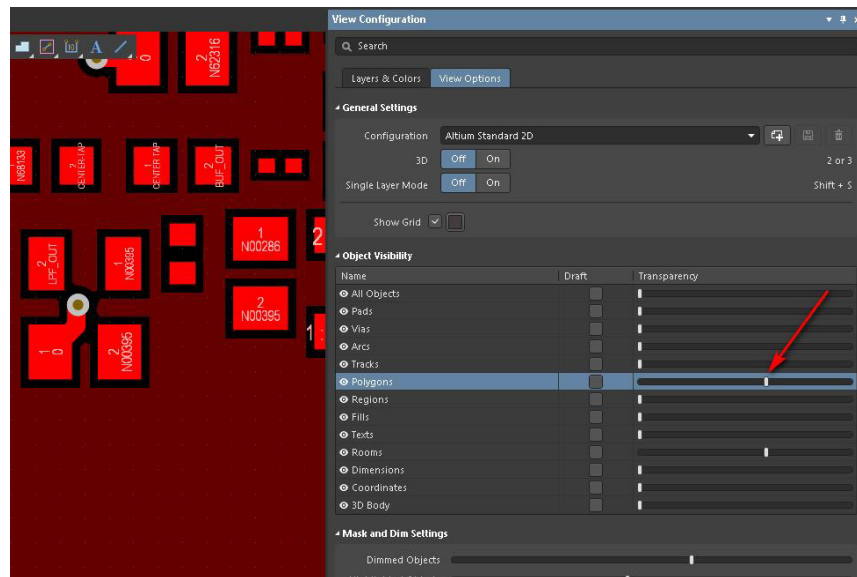

Fig. 21.11 - Adjusting polygon transparency in Altium Designer

Use the **Polygon Actions » Shelve Selected** command from the context menu to hide a polygon. To display polygons, use the T**ools » Polygon Pours » Restore Shelved Polygons** command from the main menu.

The most powerful tool for working with polygons is the *Polygon Pour Manager* dialog, which accessed by choosing the **Tools » Polygon Pours** command. In the *Polygon Pour Manager*, you can create new polygons, perform repour for the entire design, change design rules, copy polygons, change net assignments and much more (Fig. 21.12).
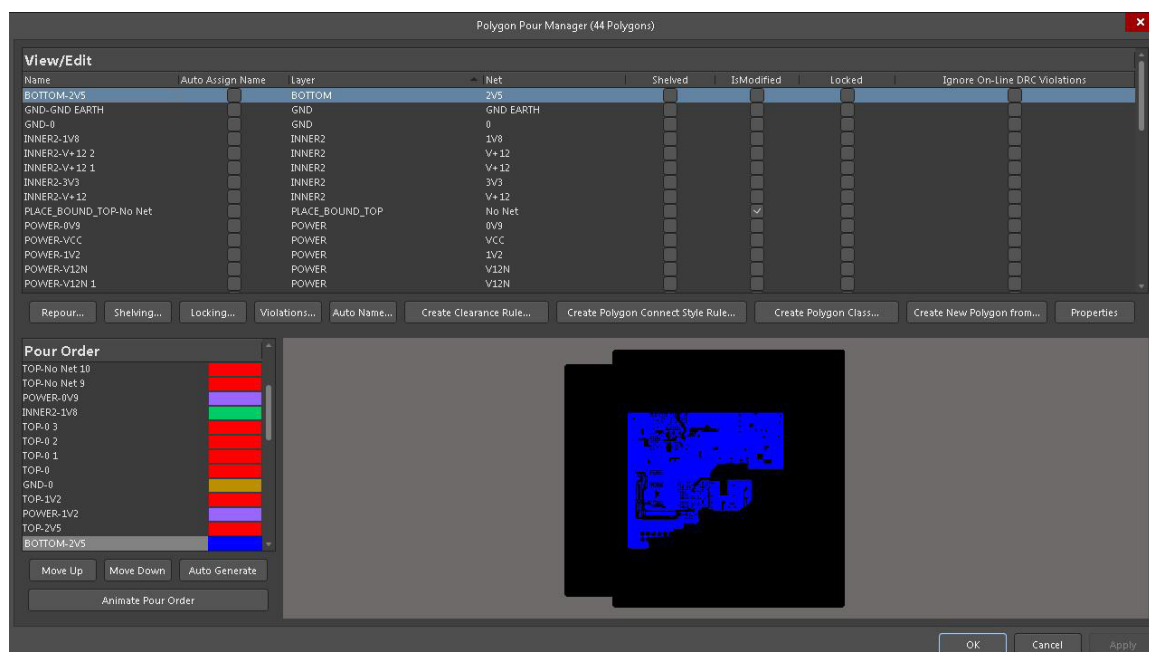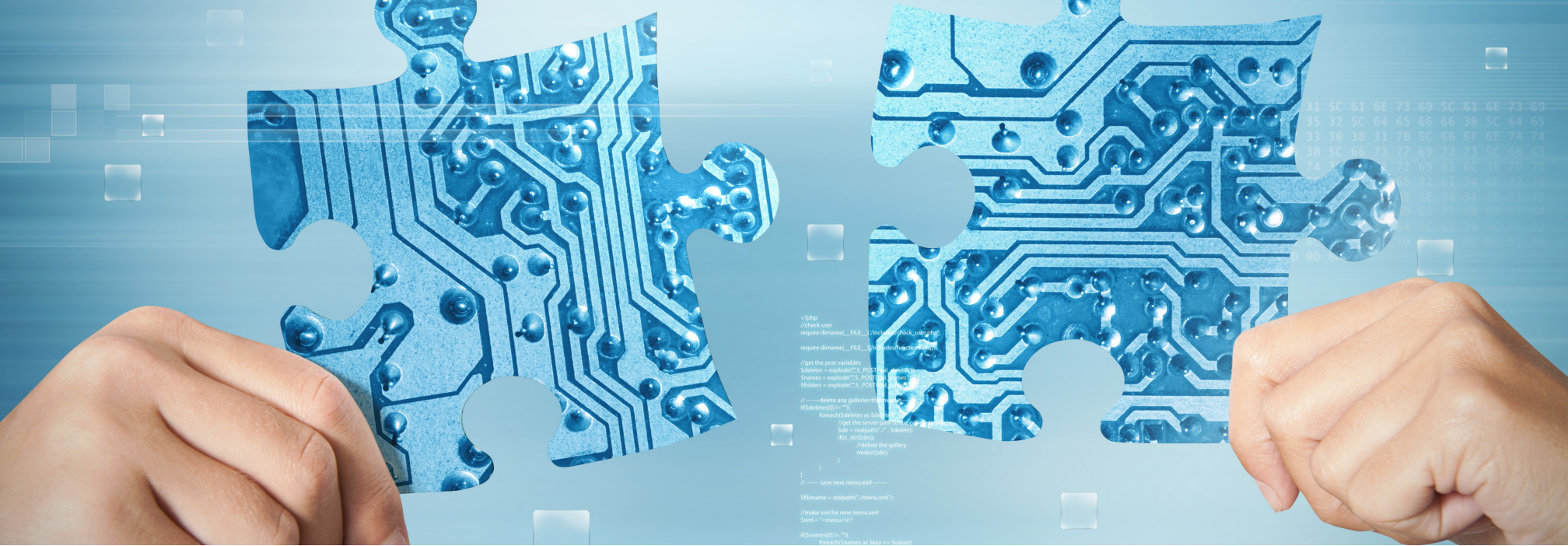

Fig. 21.12 - the Polygon Manager dialog

For example, you can press the **Create New Polygon from » Board Outline** button. Polygon corresponding to board outline will be created, and you can assign it it to a net and a layer.

22. Design rule check and violation resolving

This section covers violation search and resolving in Altium Designer. The software offers comprehensive tool set for design rule check, such as violation markers, the PCB Rules And Violations panel, the Messages panel and some others. You will able to master these tools quickly, because there are many similar to OrCAD PCB Designer things.

DRC report and visual violation control

In OrCAD PCB Editor, you can use the *Design Rules Check (DRC) Report* dialog (**Tools » Quick Reports » Design Rules Check Report**) to display a complete violation list and the *Design Rules Net Shorts Check (DRC) Report dialog* (**Tools » Quick Reports » Design Rules Net Shorts Check Reports**) to display a list of critical net short violations. The **DRC Marker Location** column shows violation coordinates – click an entry to locate the violation. The special "butterfly" markers are shown in violation locations. When you hover the cursor over a marker, violation name is displayed in the pop-up tip (Fig. 22.1).
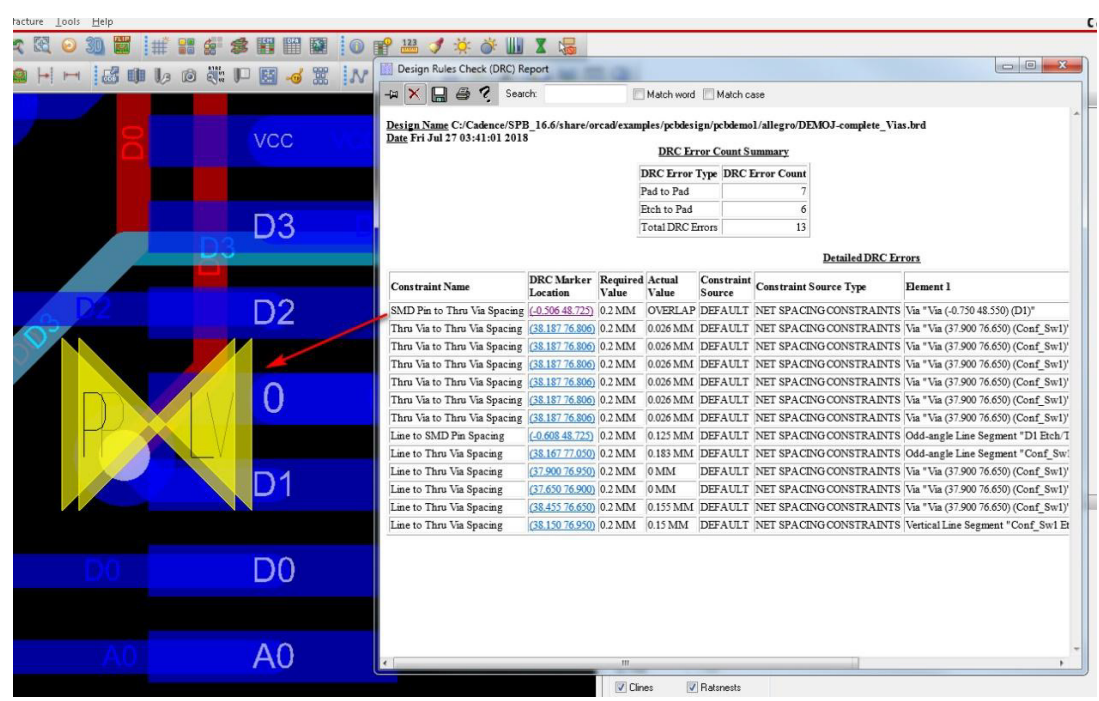

Fig. 22.1 - Violation locating via the Design Rules Check (DRC) Report dialog in OrCAD PCB Editor

In Altium Designer, use the *Design Rule Checker* dialog (**Tools » Design Rule Check**) to run the check and display violation report. This dialog provide you with access to rule check options, such as the **Create Report File** option for generating a DRC report, the **Sub-Net Details** option for displaying a detailed marker and others. In the **Rules To Check** section, you configure design rule checking modes, as it was discussed in section 18 of this guide. DRC report generation starts when you click on the **Run Design Rule Check** button (Fig. 22.2).
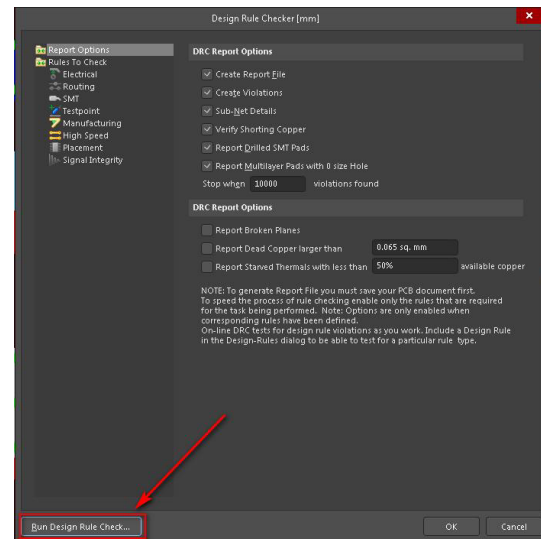
Fig. 22.2 - Configuring DRC violation report in Altium Designer

Then a detailed HTML-report will be displayed. Each violation entry has link to the location of the offending object in the workspace. You can open the report in a separate window for easy navigation. Violation locations are marked with green exclamation marks, as well as with an additional detailed marker, specifically for this violation type. For example, if a clearance is violated, the distance specified in applicable rule is displayed (Fig. 22.3).

Fig. 22.3 - Violation locating in Altium Designer

In OrCAD PCB Designer, violation markers are located on the dedicated **DRC Error Class** layer. You can configure marker color and visibility in the *Options* panel for a corresponding subclass, or in the *Visibility* panel (Fig. 22.4).

Fig. 22.4 - Violation marker color and visibility configuration in OrCAD PCB Editor

Altium Designer Migration Guide | 80

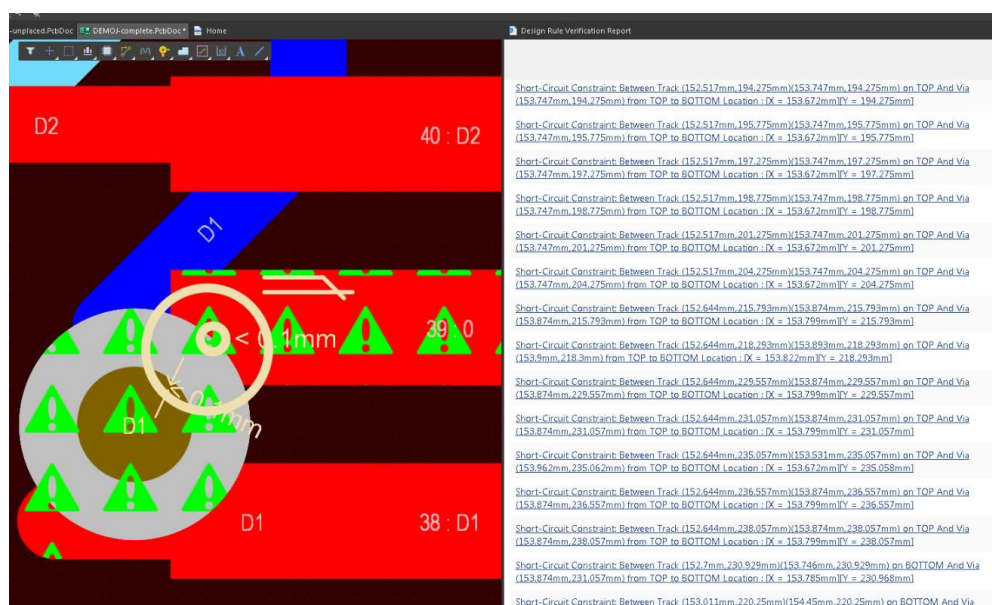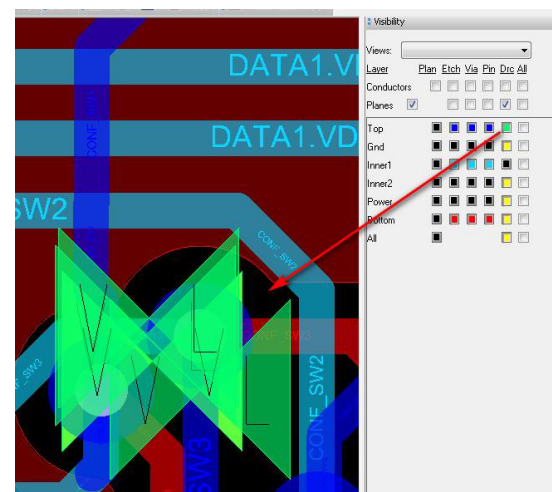In Altium Designer, you can configure violation marker style and display option on the PCB Editor - DRC Violation Display page of the Preferences dialog. In the System Colors section of the View Configuration panel you can configure marker color and visibility (Fig. 22.5).
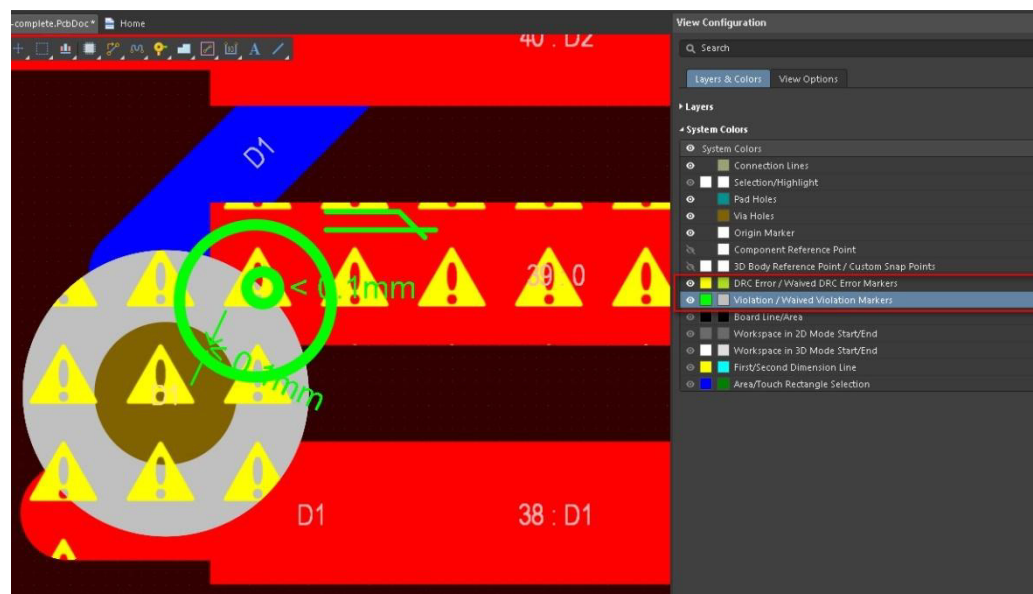
Fig. 22.5 - Violation marker color and visibility configuration in Altium Designer

The PCB Rules And Violations panel

An additional Altium Designer's handy tool for locating and resolving violations is the *PCB Rules And Violations* panel. This panel is divided into three sections – the top one lists all rule categories, below rule types of a category are listed, and the bottom section lists violations of a corresponding rule(s). The panel is interactive and allows you to quickly locate a violation and mask the offending object. You can choose the Mask mode from the top drop-down, and/or enable the **Select, Zoom** and **Clear Existing** options to highlight and zoom violation selected in the *PCB Rules And Violations* panel list (Fig. 22.6).

Fig. 22.6 - Violation locating using the PCB Rules And Violations panel

23. Manufacturing output generation

This section covers manufacturing output generation. Altium Designer supports traditional manufacturing data formats, as well as modern ones, such as Gerber X2, ODB++ and IPC 2581. You can also use special Output Job file that contains output configuration template.
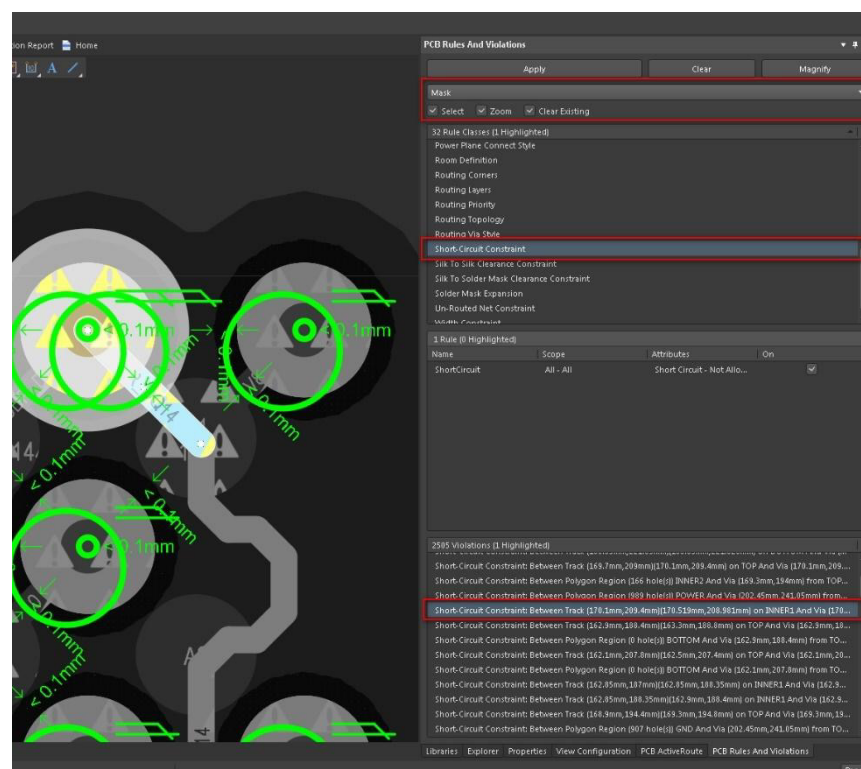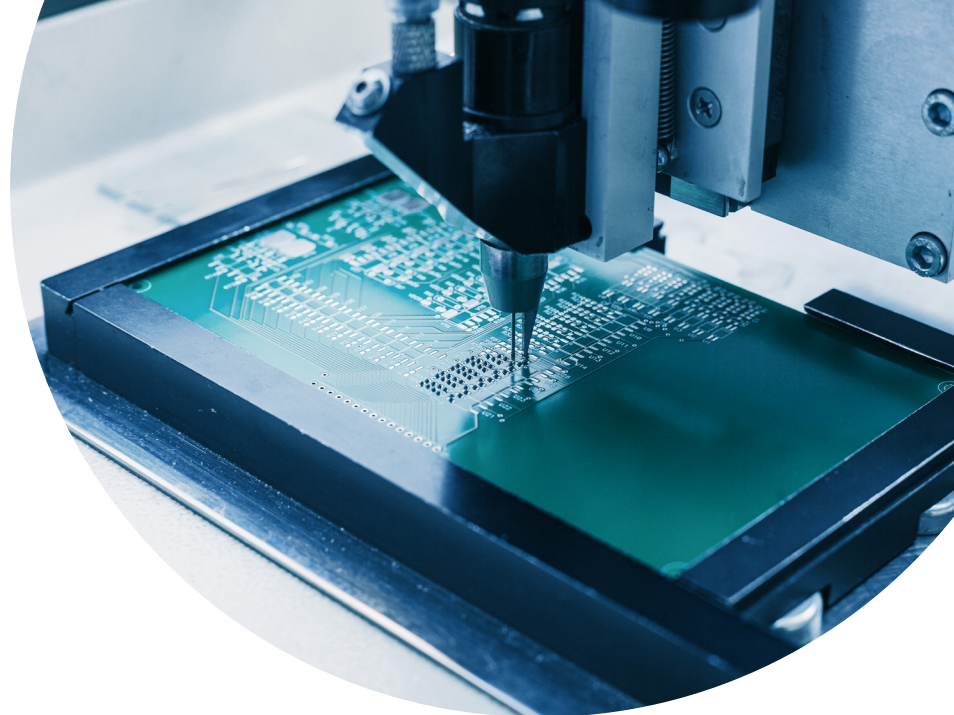
Configuration and generation of Gerber files

In OrCAD PCB Designer, manufacturing data generation is performed using the **Manufacture** menu commands. You can output Gerber files using the *Artwork Control Form* dialog (**Manufacture » Artwork**). On the **General Parameters** tab, you choose units, type and format for Gerber files. On the **Film Control** tab, all signal layers are available for generation by default. Additionally, you can manually create layer sets for Gerber files using the right-click menu. You define the file name and layers to contain (Fig. 23.1).
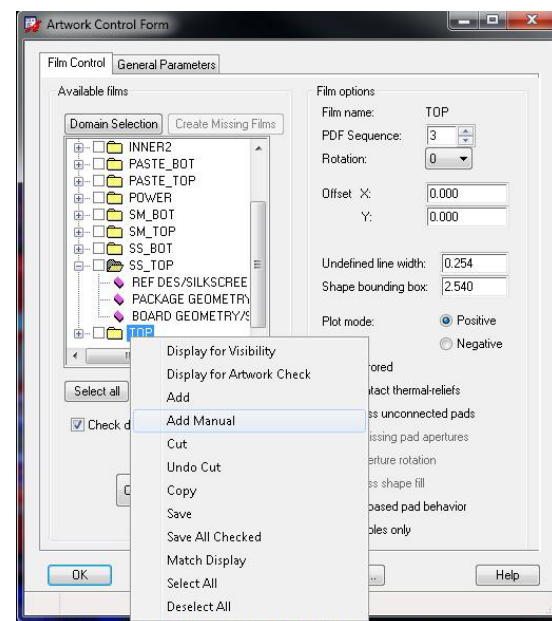

Fig. 23.1 - Gerber output configuration in OrCAD PCB Designer

Run the **Create Artwork** command to generate Gerber files. Enable the **Check database before artwork** option to check for unresolved violations. Gerber files will be created in the mfg sub-folder of the folder that contains the *.brd file (Fig. 23.2).
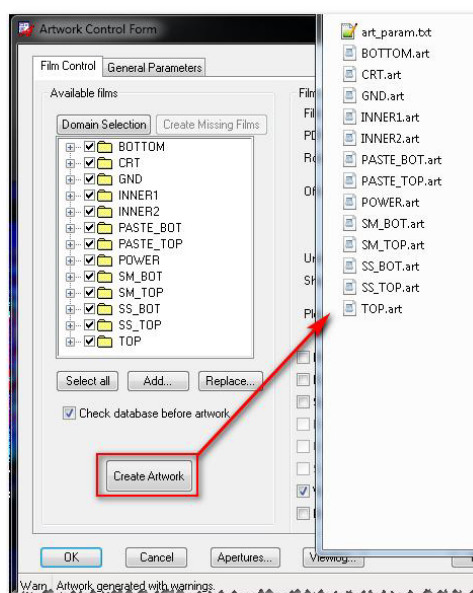

Fig. 23.2 - Gerber files generation in OrCAD PCB Designer

You also can import Gerber files to view them in OrCAD PCB Designer using the **File » Import Artwork** command. Typically it is performed in a separate *.brd file for visual review of Gerber data.

In Altium Designer, generation of Gerber files is performed using the *Gerber Setup* dialog (**File » Fabrication Outputs » Gerber Files**). On the **General** tab choose units and file format. Check layers to output on the **Layers** tab. You can select layers individually or select all electrical layers by checking the **Electrical Layers** class (Fig. 23.3). To simplify layer output to Gerber, you can create custom layer classes using the *Object Class Explorer* dialog (**Design » Classes**).
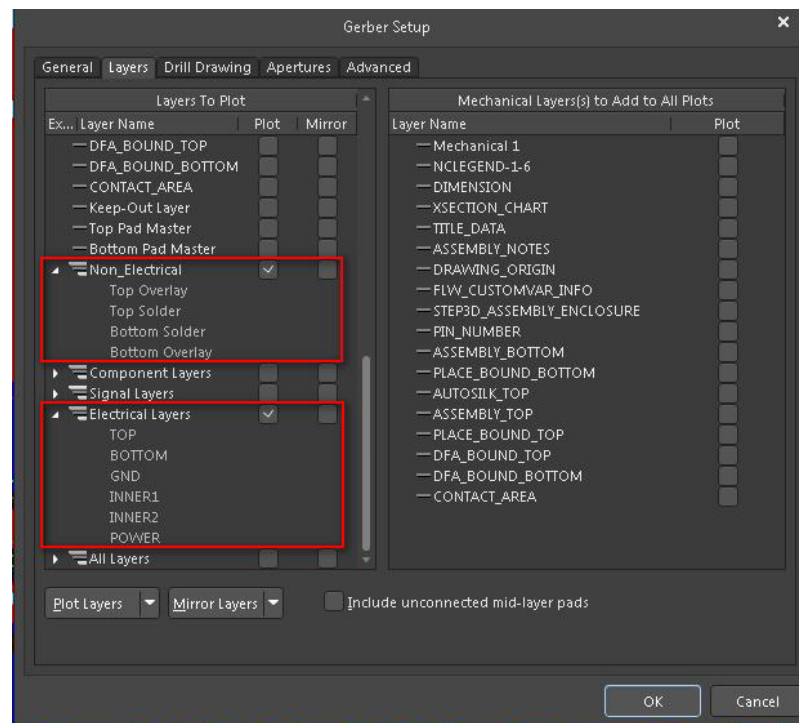
Fig. 23.3 - Gerber files output configuration in Altium Designer

In the right-hand section of this dialog you can select mechanical layers to output to Gerber, for example, the board outline.

In Altium Designer, all outputs are created in the **Projects Output For <ProjectName>** subfolder of the project folder. Gerber files has names corresponding to layer names, for example, *.gtl – top layer Gerber, *.gbs – soldermask Gerber. These outputs are displayed in the Projects panel in the Generated\CAMtastic Documents subfolder, which they can be opened from (Fig. 23.4).
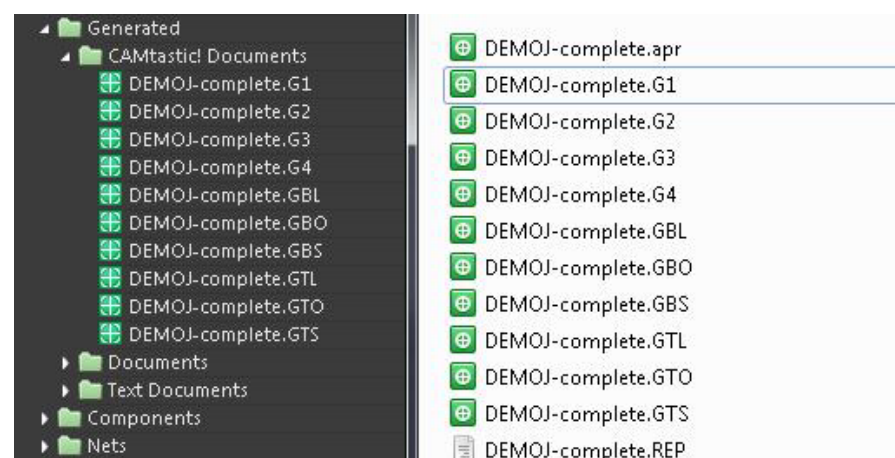
Fig. 23.4 - Gerber files in Altium Designer

Actually, once Gerber files are generated, they are automatically displayed in special CAMtastic editor, which is built in Altium Designer. For this editor, the *CAMtastic* panel is opened, where you can enable or disable the display of Gerber files (Fig. 23.5).
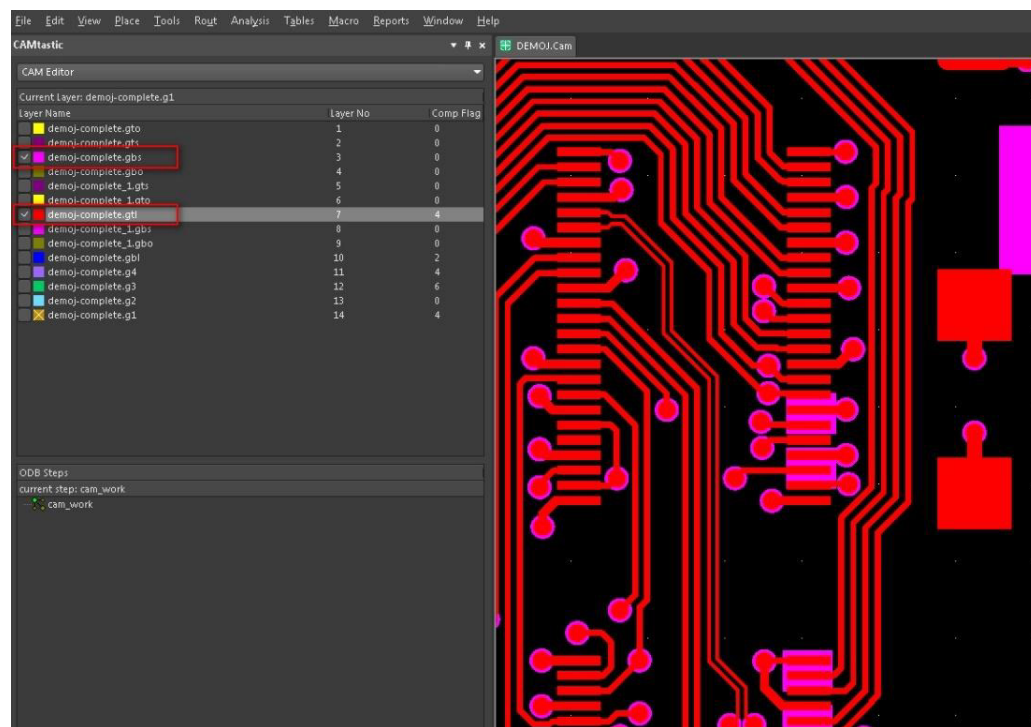
Fig. 23.5 - The CAMtastic editor in Altium Designer

Drill Table and NC Drill file

In OrCAD PCB Editor, drill table is generated using the **Manufacture » NC » Drill Legend** command. For this purpose, the dedicated **Manufacturing\NClegend** layer exists. In the *Drill Legend* dialog you can choose a table template, set data sorting, etc. (Fig. 23.6).

Fig. 23.6 - Drill table generation in OrCAD PCB Editor

To configure drill symbols on the PCB use the *Drill Customization* dialog (**Manufacture » NC » NC Drill Customization**). Here you can choose the symbol and the sign for each hole manually or do it automatically using the **Auto generate symbols** button, then regenerate the drill table (Fig. 23.7).
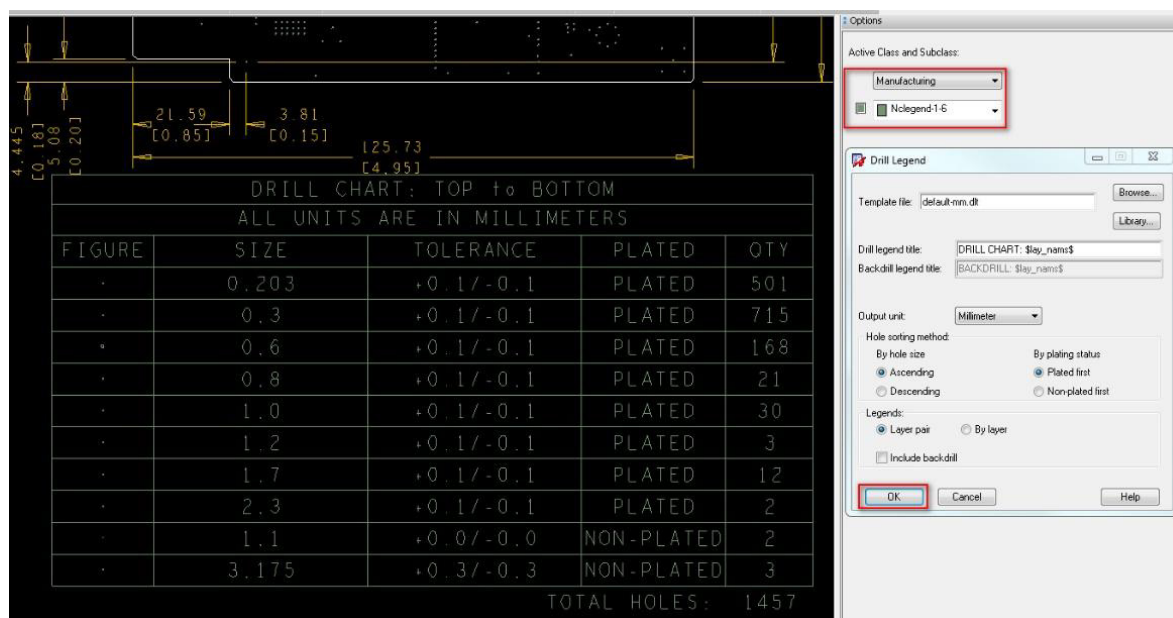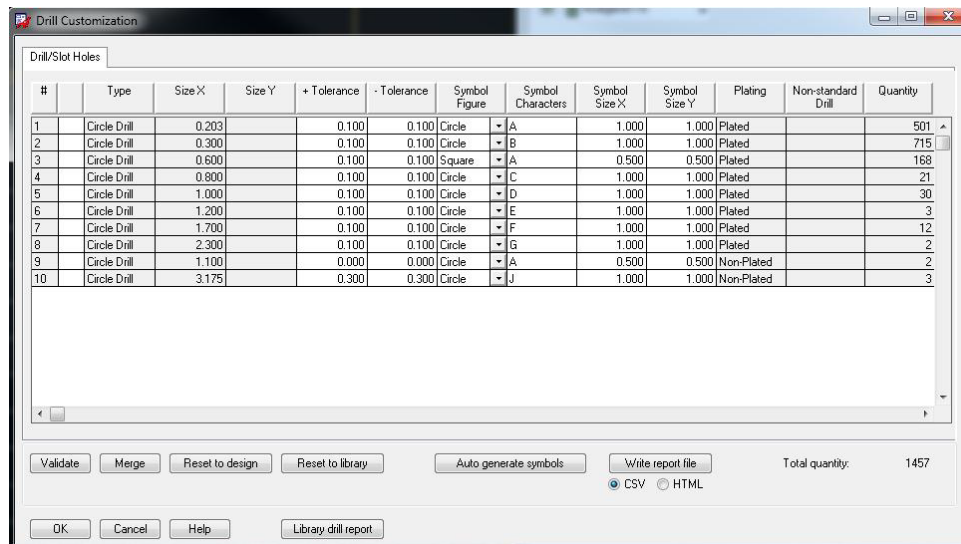

Fig. 23.7 - Drill table data configuration in OrCAD PCB Editor

Drill files generation is performed with the *NC Drill* dialog (**Manufacture » NC » NC Drill**). Additional options, such as units and format, are configured using the **NC Parameters** button. Click on the **Drill** button to generate *.drl drill files in the mfg folder (Fig. 23.8).
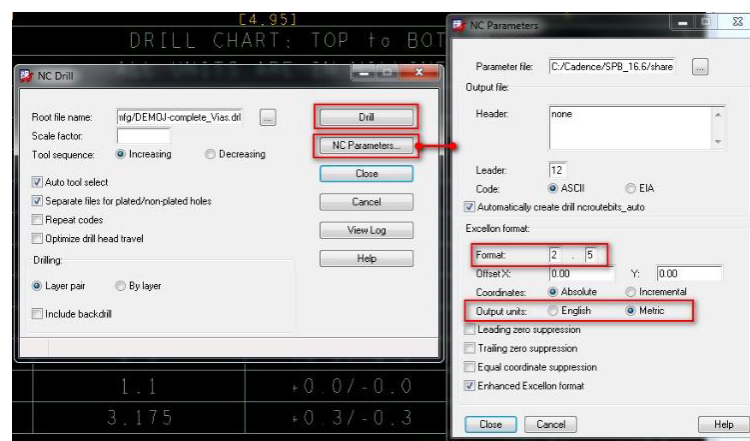
Fig. 23.8 - Drill file configuration in OrCAD PCB Editor

In Altium Designer, drill table is placed on the dedicated **Drill Drawing** layer via the **Place » Drill Table** command. All table options are available in the *Properties* panel (Fig. 23.9).
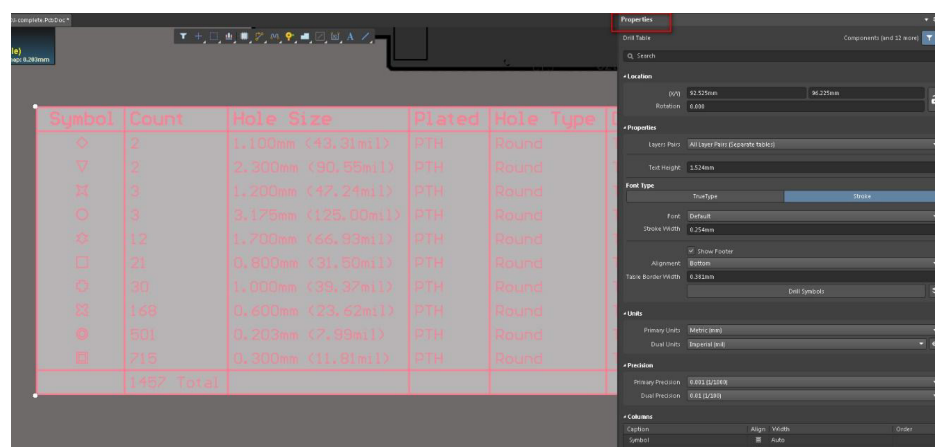
Fig. 23.9 - Drill Table in Altium Designer

You can click on the **Edit Columns** button at the bottom of the *Properties* panel to edit column visibility and captions. Use the *Drill Symbols* dialog accessed by clicking the **Drill Symbols** button to configure drill symbols. Here you can assign a symbol or an alpha character to a hole type (Fig. 23.10).
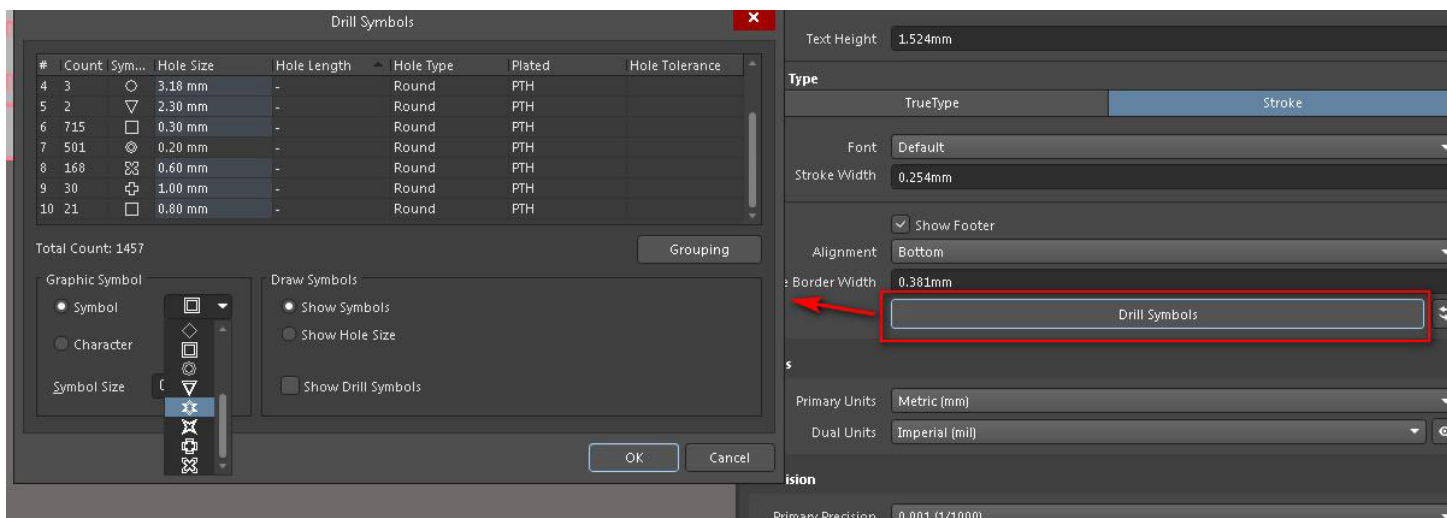
Fig. 23.10 - Drill symbol configuration in Altium Designer

In Altium Designer, drill output is performed via the *NC Drill Setup* dialog (**File » Fabrication Outputs » NC Drill Files**), where you can set standard options, as it is performed in OrCAD PCB Editor. The file has the *.txt extension and stored in the Project Output For... folder. Like Gerber files, NC Drill files are viewed in the CAMtastic editor, and the file is added to the project structure into the Generated\CAMtastic Documents in the Projects panel (Fig. 23.11).
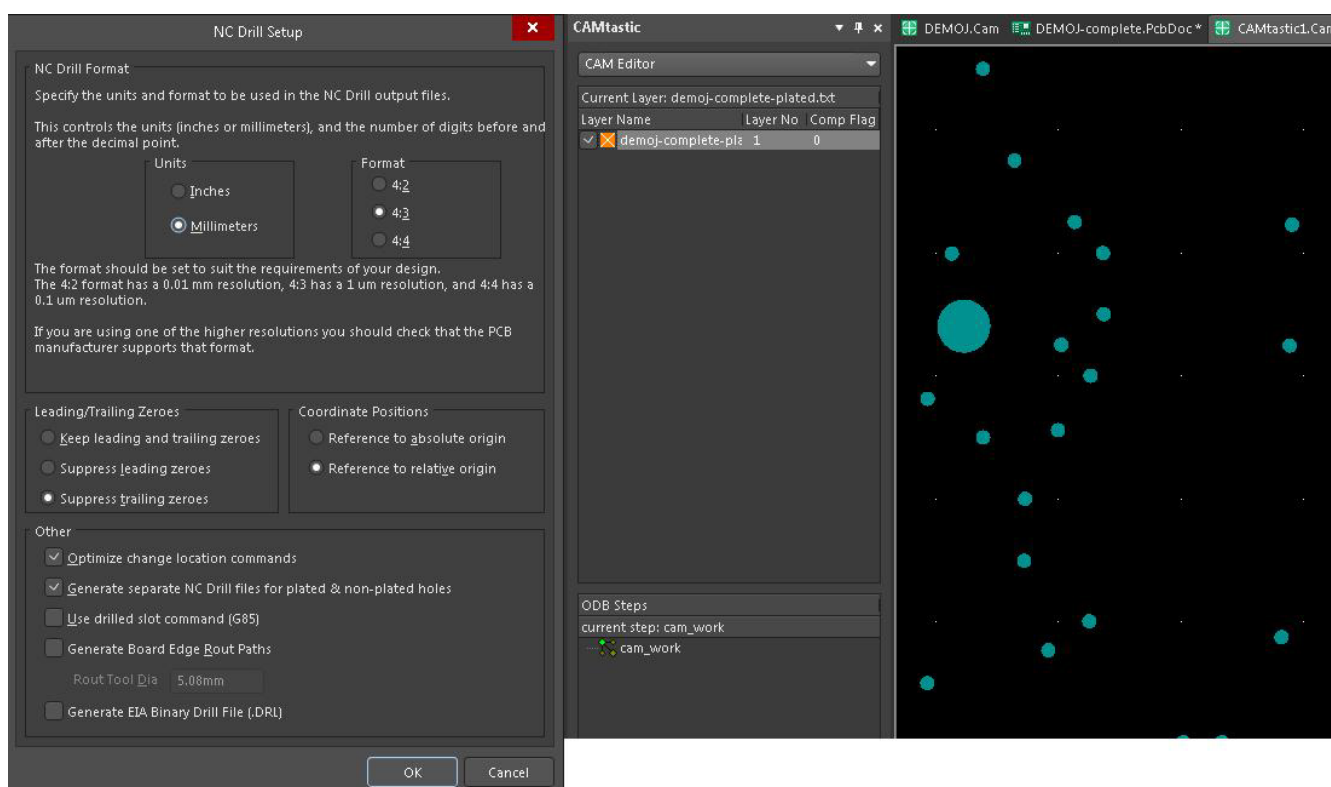
Fig. 23.11 - Drill output configuration in Altium Designer

Output Job files

In Altium Designer, to optimize configuration of manufacturing data output to a file or to a printer, you can use files of the special type – Output Job files. You can add an Output Job file to a project in the normal way. This file has the *.Outjob extension and is stored in project structure. To add an output, click on the button in any of the output sections, for example, **Add New Fabrication Outputs** (Fig. 23.12).
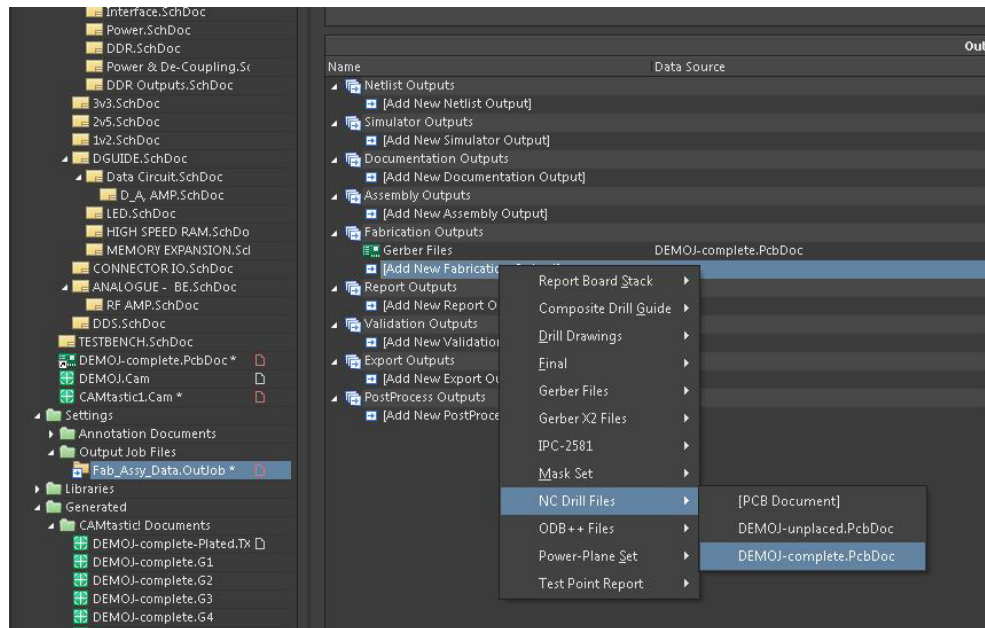
Fig. 23.12 - Output configuration in Outjob file

If you added Gerber file output this way, then double-click it, the configuration dialog that was discussed above appears. Then select the appropriate container, for example, **Folder Structure**, and assign desired output to it – an arrow appears, showing connection of the output with the selected container (Fig. 23.13).
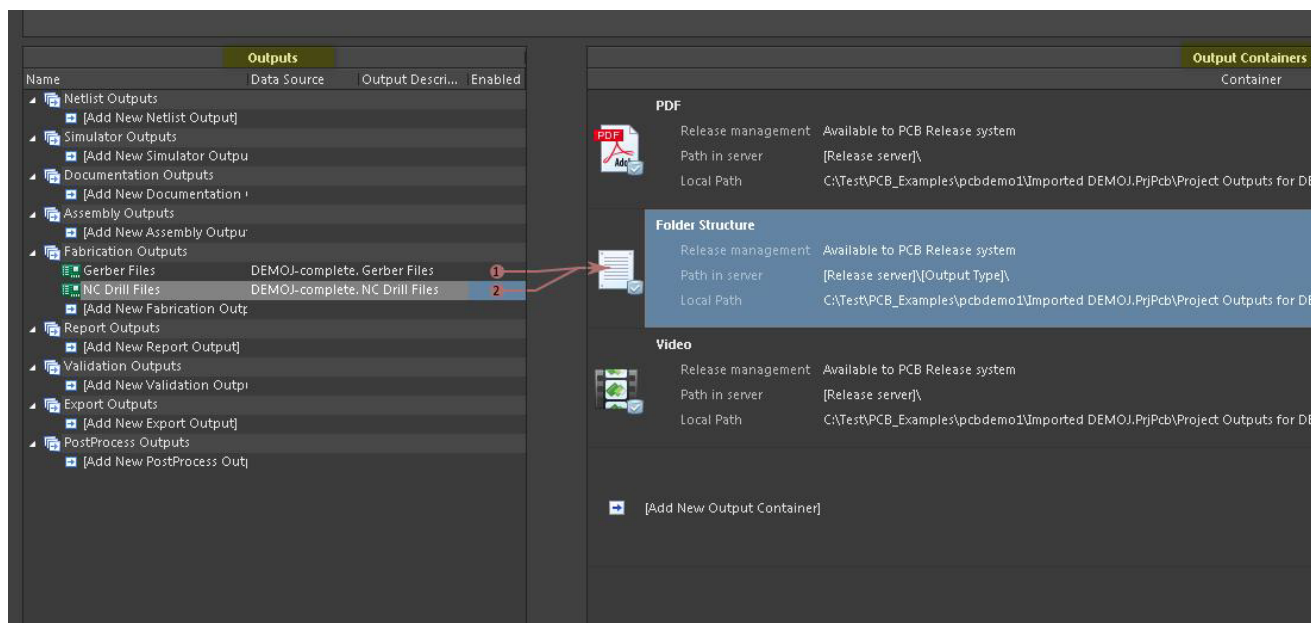
Fig. 23.13 - Selecting container of the outputs
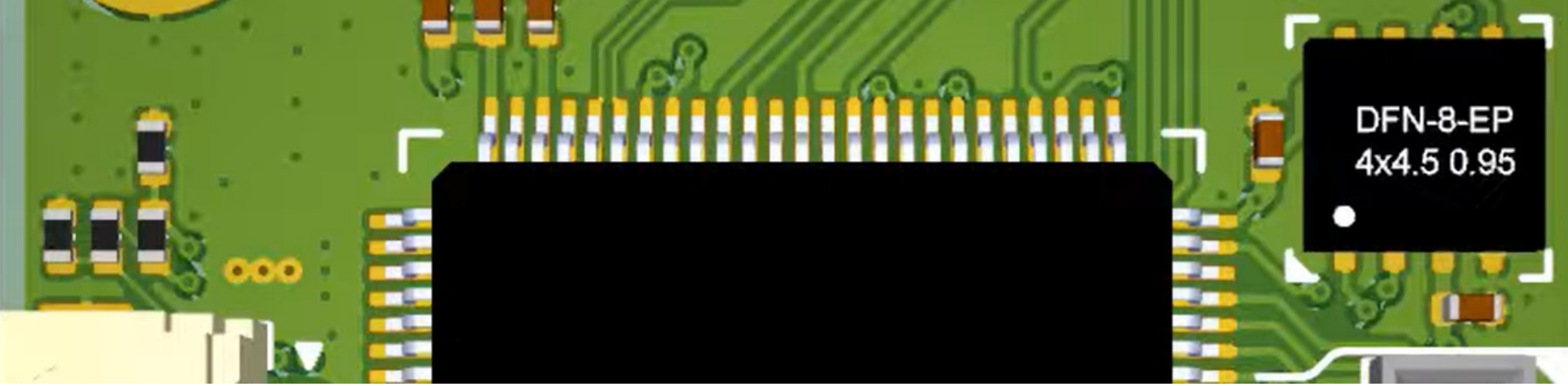
Appendix A. Configuring and running Cadence OrCAD file import to Altium Designer

As was mentioned above, Altium Designer allows to import data from virtually any PCB design software. Before running the *Import Wizard*, make sure that the appropriate OrCAD/Allegro importers are installed. When Altium Designer is started, go to the installed extension settings – choose the **Extension And Updates** command at the top-right of the application, then click on the **Configure** button (Fig. A.1).
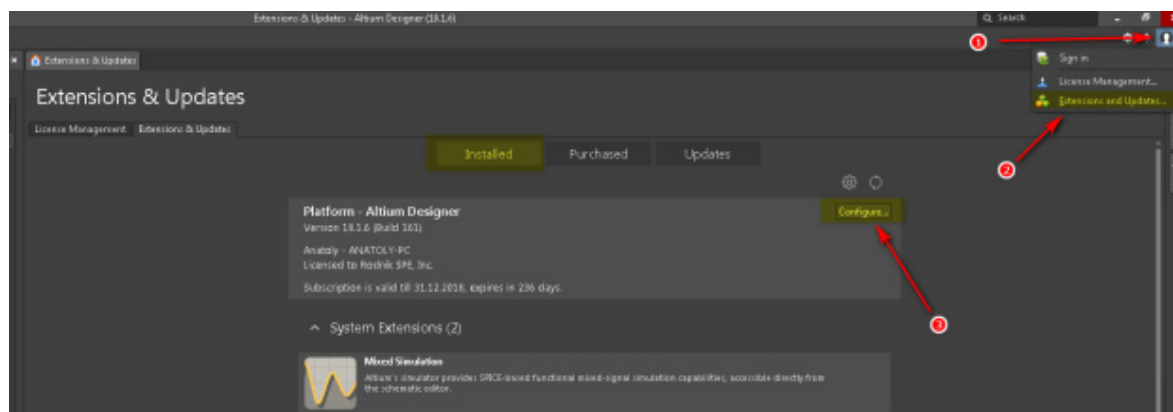
Fig. A.1 - Displaying the list of installed importers/exporters

In the **Importers/Exporters** section, you will find the list of all installed third-party ECAD data importers/ exporters. For importing files from OrCAD PCB Designers, OrCAD and Allegro importers are required, because both software systems work on a single software platform with the general OrCAD Capture schematic editor and the OrCAD/ Allegro PCB Editor (Fig. A.2).
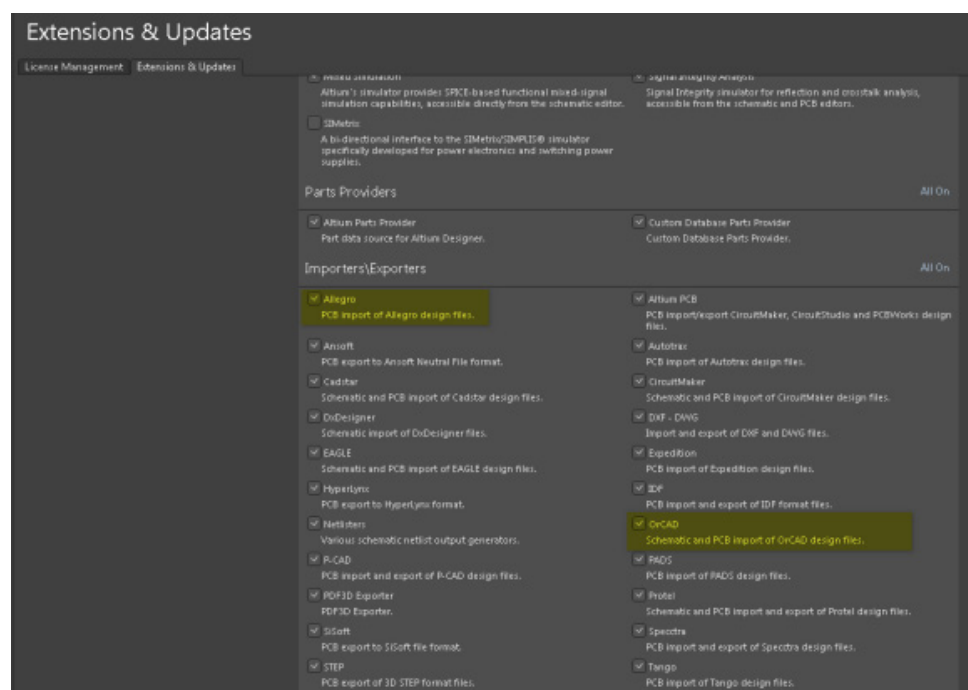
Fig. A.2 - Installed OrCAD and Allegro importers

Once required importers are enabled, open the *Import Wizard* (**File » Import Wizard**). On the first screen click **Next** to navigate to the **Selecting the Type of Files to Import** screen, where you can choose desired files to import, including OrCAD files: libraries (*.olb), schematics (*.dsn), and PCBs (*.brd) (Fig. A.3).

Fig. A.3 - Type of files to import selection

Next screens of the wizard are described in details. You can simply keep clicking on the **Next** button leaving the default settings until the **Finish** button is active, and your migrated design will be shown.

As you probably already noticed that for migrating the whole OrCAD project, you need to run the *Import Wizard* twice – one time for the schematic (*.dsn) and another time for the PCB (*.brd), and then merge them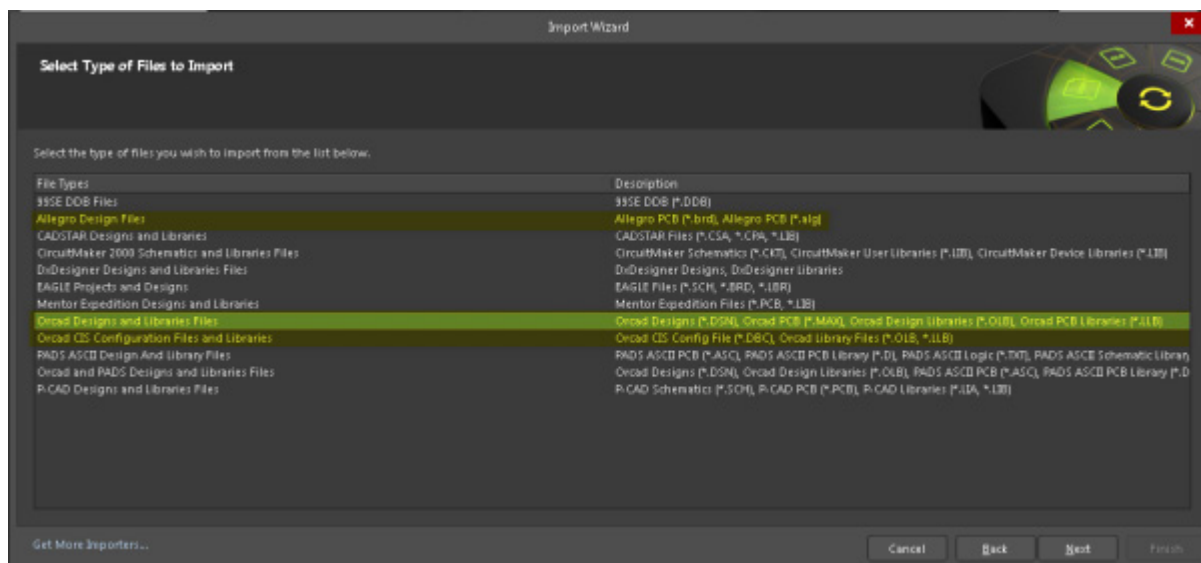 into a single project. If required, you can also import layout files from the legacy version of OrCAD Layout (*.max), which was discontinued by Cadence in 2009. Note that the *Import Wizard* does not allow to import OrCAD footprints (*.psm) and other symbol types directly. But this should not be an obstacle for you, because you can extract required PCB libraries once PCB file is imported.

Before migrating a layout file (*.brd), it is recommended to apply a number of changes to it:

1. Remove all fanouts using the **Route » Via Structure » Delete** command. Do this for Pins, Vias and Clines.

2. Remove polygons with other polygons within. Typically it happens on the internal Plane layers with power and ground polygons. Once the design is migrated, you can quickly regenerate missing polygons within the board area or within a routing keepin using the *Polygon Pour Manager*.

As an example, we consider the Demo.opj from the C:\Cadence\SPB_16.6\share\orcad\examples\pcbdesign\pcbdemo1 folder, which is available for OrCAD PCB Designer users in 16.6-2015 (Hotfix 51) or later version. It is quite large design with complex hierarchy and multilayer board.

Appendix B. Library creation tools

In Altium Designer, you can use the **File » New » Library** menu commands. Using them you can create schematic libraries, PCB footprint libraries, Pad/Via template libraries. This is very intuitive.

For every library type there is a dedicated panel. When creating a SCH Library, you are using the SCH Library panel, where its components list. Using the **Tools » New Component** command you can add new components. Main and additional attributes for each component are shown in the *Properties* panel (Fig. B.1).
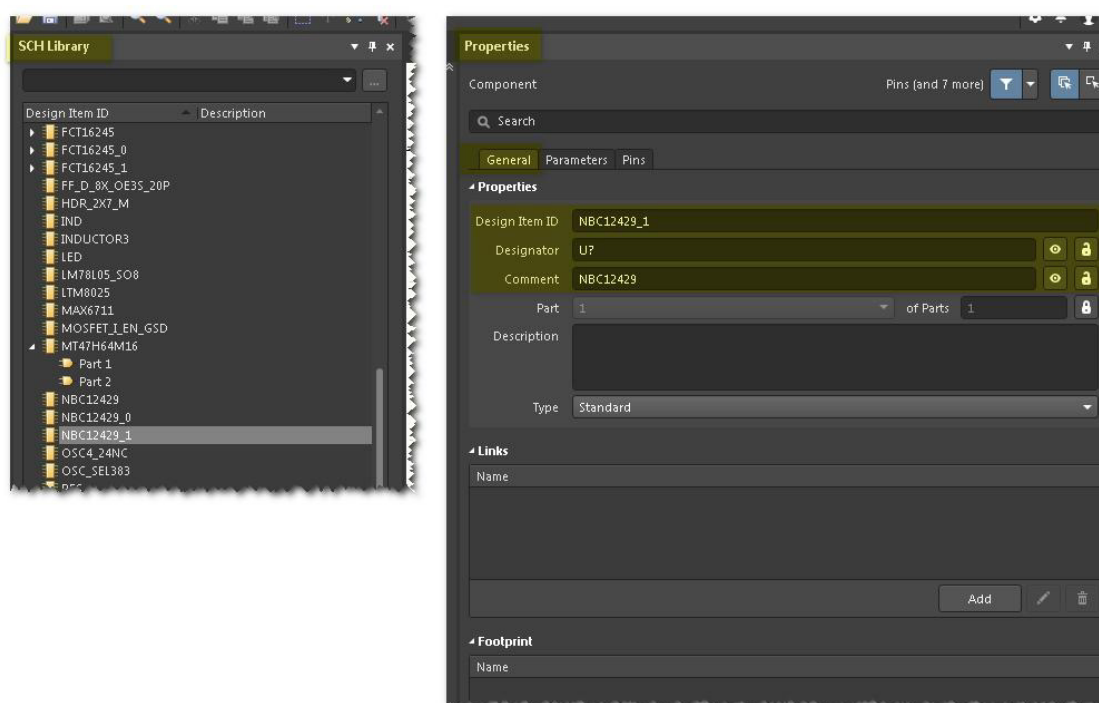

Fig. B.1 - Choosing a component in the SCH Library panel and its main properties in the Properties panel

The *Properties* panel displays such properties as **Design Item ID** (component name), **Designator** and **Comment**, where usually the full component model name is indicated. For the **Designator** and **Comment** properties, you can toggle their visibility on schematic sheet using their associated ⊙ button. Note that **Designator** and **Comment** properties are not displayed by default. Optionally you can enable their display using the **Show Comment/Designator** option at the bottom of the *Properties* panel.

Creating of a schematic symbol is quite simple (Fig. B.2).

1. Place pins using the **Place » Pin** command or the appropriate button in the **Active Bar**. You can place a set of pins using the **Edit » Paste Array** command.

2. Define component properties in the *Properties* panel.

3. Place graphical objects using the **Place** menu commands.

Fig. B.2 - Creating the schematic symbol

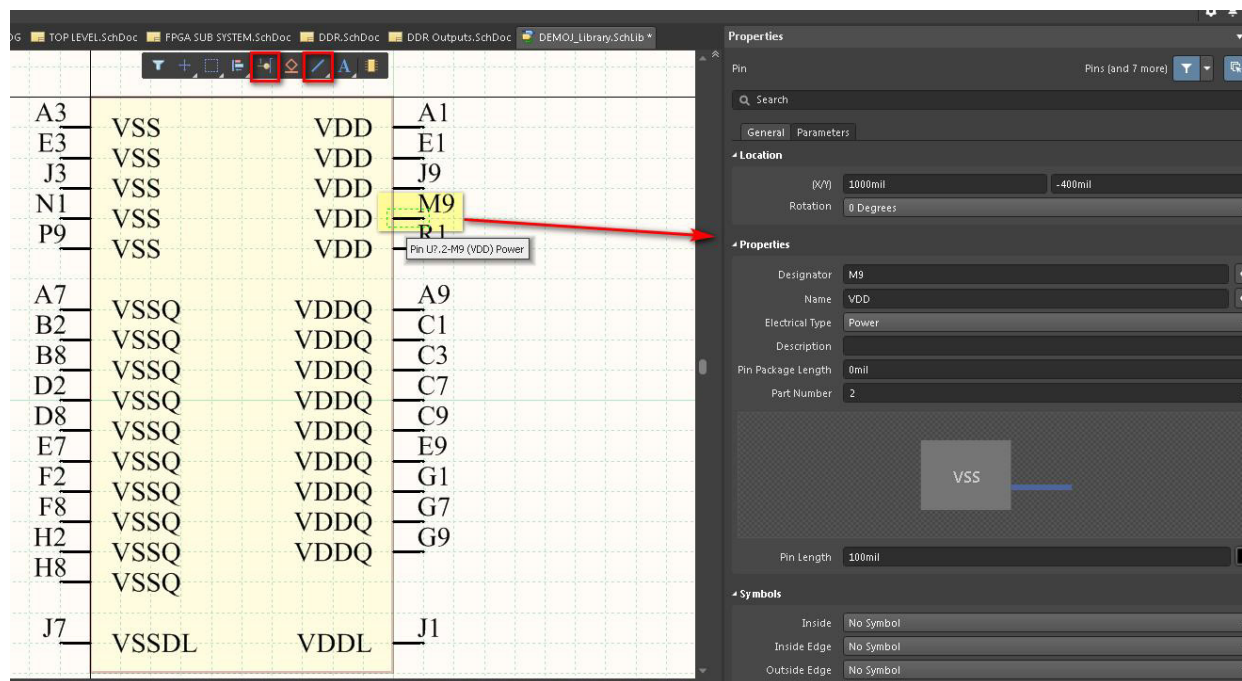Try the *Symbol Wizard* (**Tools » Symbol Wizard**), which would be useful when working on components with large number of pins. It is the similar to OrCAD Capture's **New Part From SpreadSheetHere** tool, but more intuitive and powerful. You can copy pin information to the Windows clipboard and simply paste it through the *Symbol Wizard* – Altium Designer will automatically generate a ready-made symbol with the specified pin configuration for you.

Additional properties, such as Manufacturing Part Number, Value and others, can be defined on the **Parameters** tab of the *Properties* panel. Pin properties can be edited on the **Pins** tab.

When working on multi-part components, you can add new parts using the **Tools » New Part** command. You should add graphics for each part separately, even for a homogeneous component.

When a schematic library is finished, you can run its check using the **Reports » Component Rule Check** command.

A new PCB library can be created by running the **File » New » Library » PCB Library** command. All library footprints are listed in the *PCB Library* panel. A new footprint can be added by running the **Tools » New Blank Footprint** command. Before placing footprint graphics and pads you should configure units, grid and snapping. As in the previous cases, this is done via the *Properties* panel. All these settings are intuitive.

Notice defining the grid. Of course, you can simply use the **G** shortcut. But in addition to the global grid, you can create your own grids, including, for example, the polar grid. To do this, click the **Add** button in the **Grid Manager** section of the *Properties* panel and select cartesian or polar grid, and then double-click the new entry in the grid list. The *Grid Manager* dialog appears, where you can specify the desired step and style of your grid (Fig. B.3).
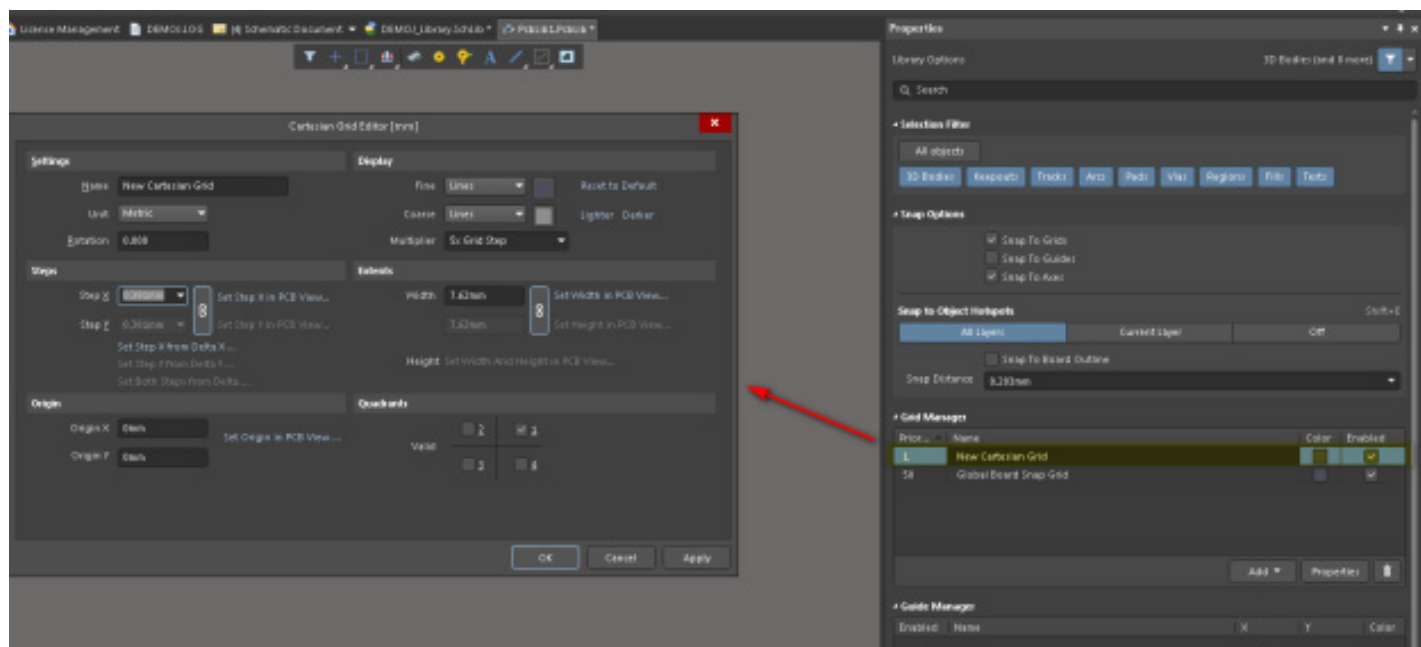
Fig. B.3 - Grid configuration via the Grid Manager dialog

The origin is marked in the workspace with the cross-hair in the circle. You can change its location using the **Edit » Set Reference » Origin**.

Once this preliminary steps are done, you can place footprint pads and graphics using the **Active Bar** commands as follows:

1. Place pads by running the **Place » Pad** command

   Creating the dedicated padstack library, as it is done in OrCAD, is optional. During placing a new pad simply define its properties, such as hole size, plating, shape and size, Designator, via the *Properties* panel.

2. Place silkscreen graphics (**Top/Bottom Overlay** layer)

   You can use, for example, the **Place » Line** command. All line properties are displayed in the Properties panel.

3. Place assembly instructions (**Mechanical** layers)

   In Altium Designer, all supporting information, such as physical and manufacturing component boundaries, supporting texts, and so on, is placed on mechanical layers. You can create additional mechanical layers via the *View Configuration* panel (Fig. B.4).
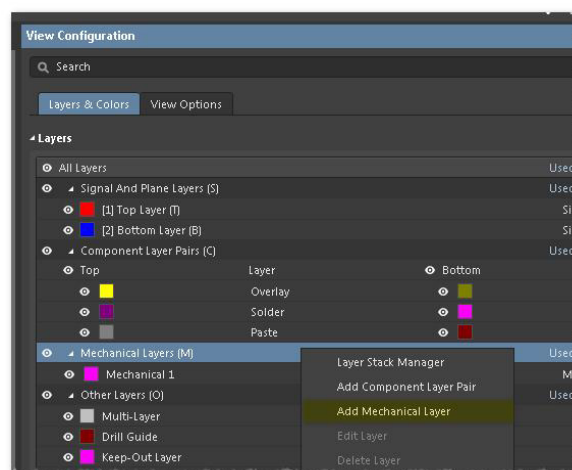

Fig. B.4 - Adding the mechanical layer via the View Configuration panel

4. Place component 3D Body using the **Place » 3D Body** command if required Switch to the 3D display mode by pressing the 3 shortcut. Set the **3D Model Type** option to Generic and select model using the **Choose** button. STEP, Parasolid, SolidWorks models are supported.

   If the detailed 3D model is not required, you can place the simplified model via the T**ools » Manage 3D Bodies For Current Component** feature. Fast and easy!

   Padstacks can be created by running the **File » New » Library » Pad Via Library** command. All action are performed here via the *Pad Via Library* panel and the Pad Template Editor (Fig. B.5).
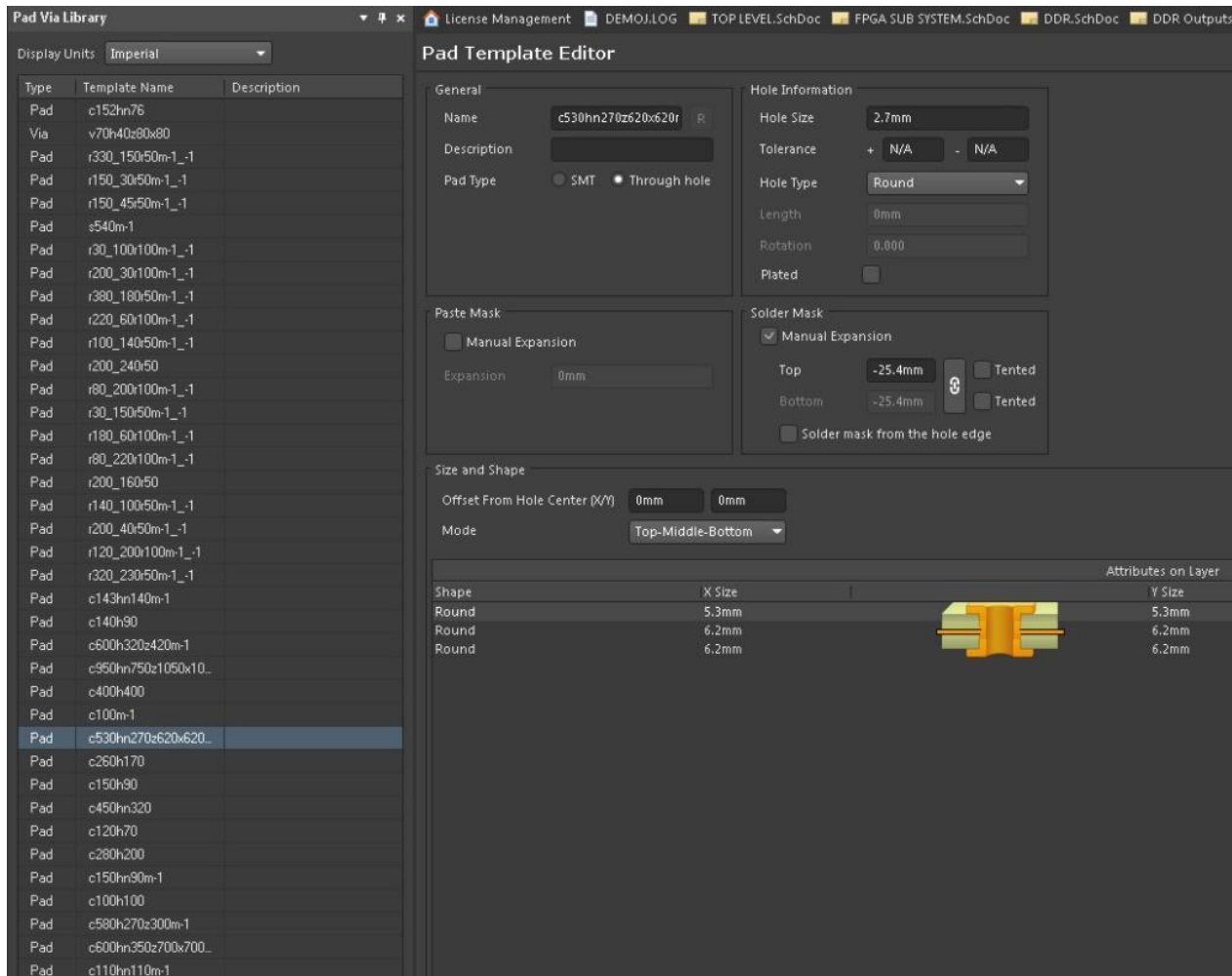

Fig. B.5 - Pad Via Library creating

During footprint creation simply install a Pad Via Library via the *PCB Pad Via Template* panel and place required pads and vias in the workspace.

Appendix C. Schematic packaging and transition to layout design after migrating a project from OrCAD

As you know, a project is migrated to Altium Designer in parts. First, schematic file is being imported, following by layout file (into a separate project). The PrjPcb, SchDoc and PcbDoc files should be stored in the same directory and in the same project in order to maintain data integrity and to avoid the project file loss in the future. We copied the DemoJ-complete.PcbDoc file to the folder with the DEMOJ project and added the PCB document to the project.

Altium Designer uses unique identifiers to synchronize component designators on the schematic sheets and on the PCB. The software assign these identifiers to each component automatically. When migrating a PCB design from OrCAD PCB Editor, unique identifiers are not assigned, so synchronization between schematic and PCB domains is not guaranteed. However, there is a simplified procedure for synchronizing unique IDs:

1. Open the DemoJ-complete.PcbDoc file and run the **Project » Component Links** command – the *Edit Component Links* dialog appears (Fig C.1).
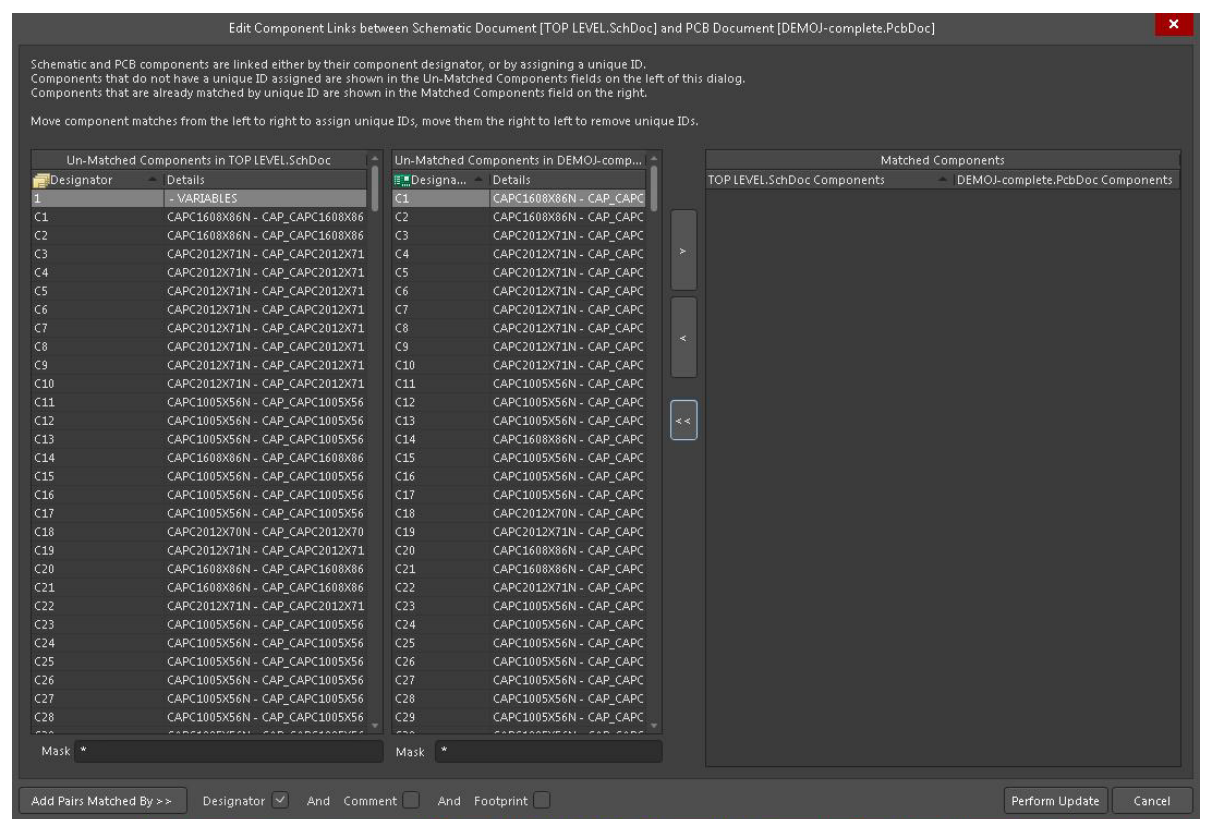

Fig. C.1 - The Edit Component Links dialog (Project » Component Links)

The left-hand side of the dialog lists components that are currently unmatched. Two lists are used for unmatched components: on the source schematic documents (left) and those unmatched components on the target PCB document (right). You can select a pair of components in these lists and click on the **Right arrow (>)** button to match them. A single entry of the synchronized component will be added to the **Matched Components** list (the right-hand side of the dialog).

For large designs, this manual matching can be time-consuming. You can perform auto-matching based on a combination of **Designator, Comment**, and **Footprint**. This may synchronize instantly most of the components.

    2. Enable the **Designator** option and disable the **Comment** and **Footprint** options.

    3. Click on the **Add Pairs Matched By >>** button to automatically match unmatched components (Fig. C.2).
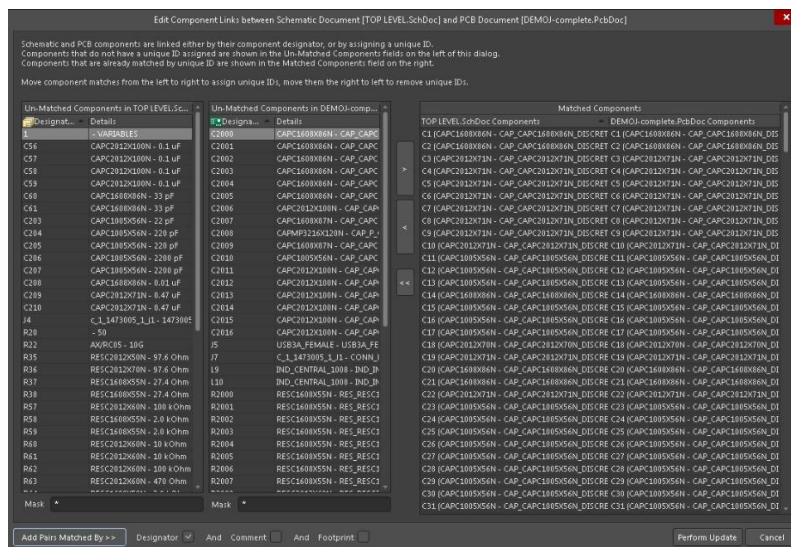

Fig. C.2 - Matching unmatched components in Altium Designer

    4. If there are unmatched components left, they should be synchronized manually. Once components are matched, press the **Perform Update** button to apply the changes.

Some components can only be placed on the PCB without having a symbol on schematic sheets. For example, these are screens, gaskets, clamps, mounting holes and other components. On the other hand, some components may be present in the schematic, but not have a footprint. To handle these components, a number of component types are supported in Altium Designer. You can explore them in the *Properties* panel, when the component is selected (Fig. C.3).
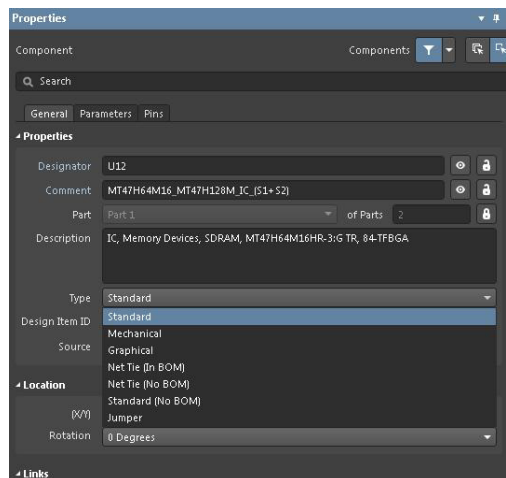

Fig. C.3 - Component types

An Altium Designer project does not use a netlist as such for transferring data between schematics and PCB. This means that you are not to keep track of the folder with text or ASCII netlist files, since all information is transferred in a "seamless" fashion, informing you about all changes. If any errors are detected in schematic sheets, the software will inform you about them.

To transfer the DEMOJ project choose the **Design » Update PCB Document DemoJ-complete. PcbDoc** command from SCH Editor, or choose the **Design » Import Changes From DemoJ. PrjPcb** command from PCB Editor. The manual net matching dialog appears (Fig. C.4).
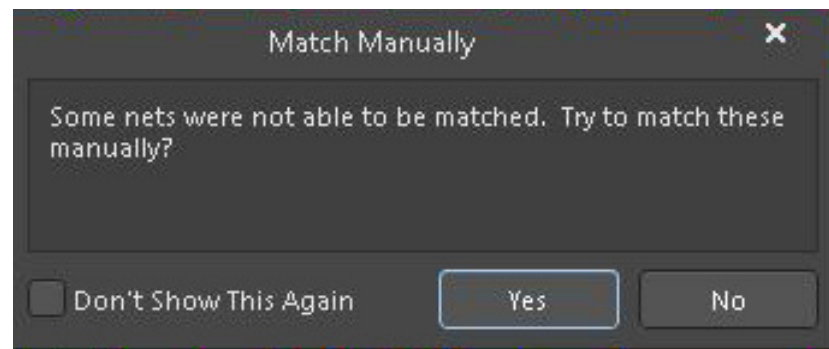

Fig. C.4 - Manual net synchronization

Here it is better to click **No** and proceed to generate ECO.

One of the most important tasks is renaming the nets. In the "Creating connectivity" section, Altium Designer's system naming of nets agreement was discussed. During ECO performing, it is strongly recommended to rename the OrCAD PCB Designer nets in accordance with Altium Designer's naming, as shown below (Fig. C.5).
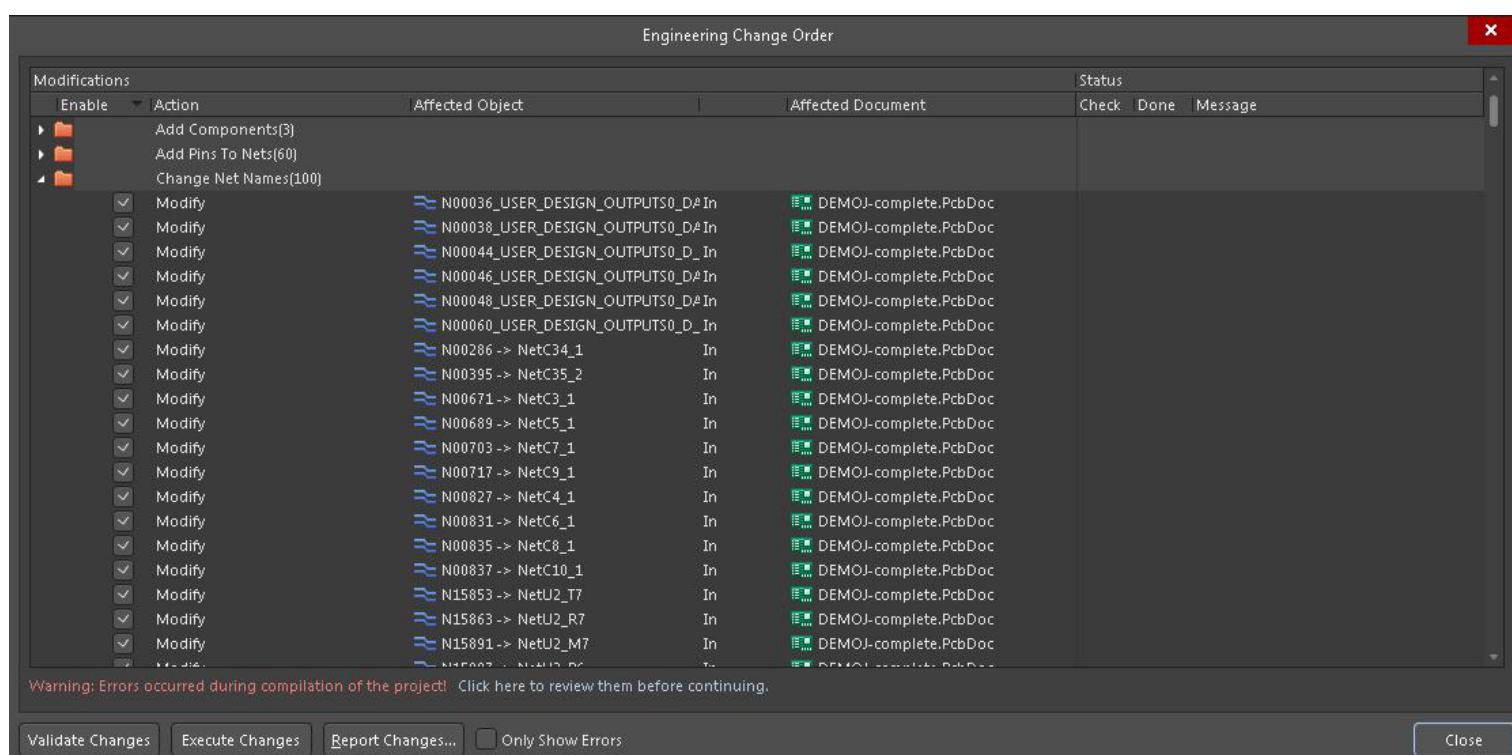

Fig. C.5 - Net renaming during ECO performing

Other proposed changes are also shown here. You can control these changes using the project options dialog (**Project » Project Options**). Altium Designer offers to generate rooms, net classes, component classes, etc. by default.

The **Class Generation** tab enables you to configure and control room and class generation. By default, Altium Designer offers to generate rooms and net classes for each schematic sheet. You can simply disable these options (Fig. C.6).
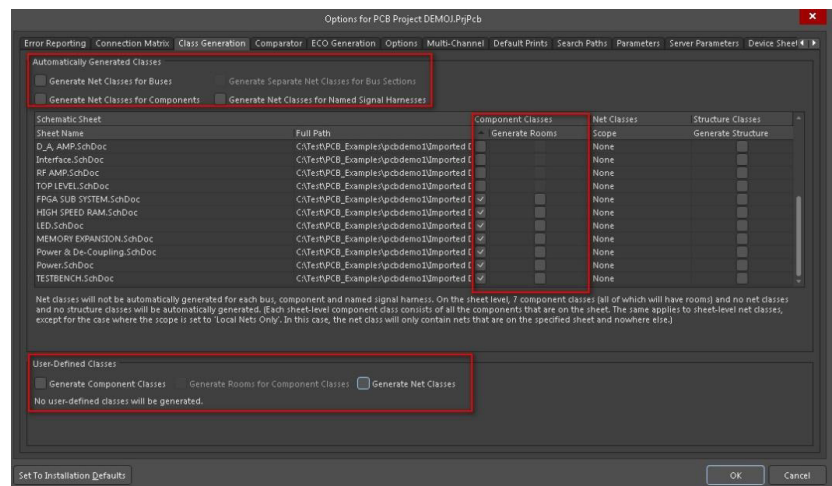

Fig. C.6 - Automatic class and room generation options

The **Comparator** tab enables you to define which types of differences to find when comparing schematics and PCB. You can disable detection of some differences and therefore prohibit applying of related changes (Fig C.7).
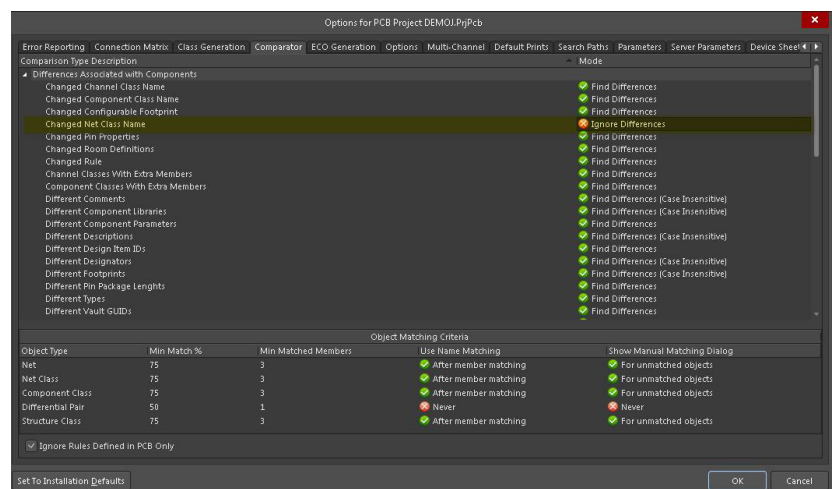

Fig C.7 - The Comparator tab of the Project Options dialog

The **ECO Generation** tab enables you to configure which modification types can be included when generating an Engineering Change Order (ECO). You can exclude a modification type from the generated ECO by setting its mode to Ignore Differences (Fig. C.8).
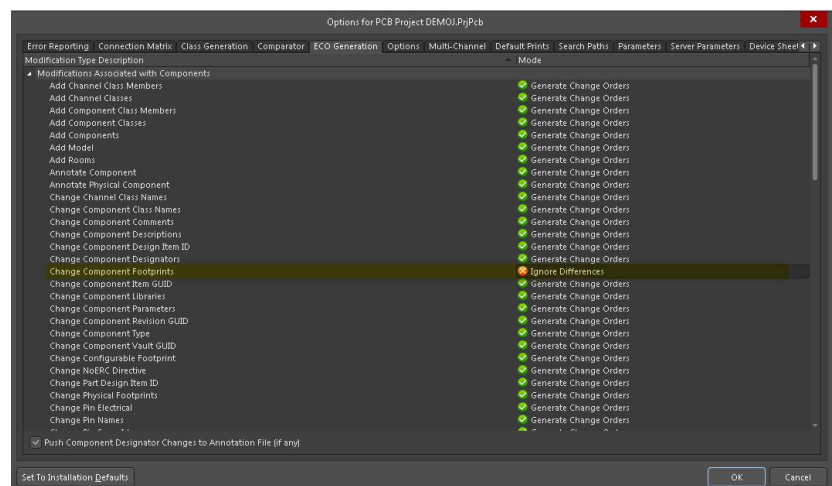

Fig. C.8 - The ECO Generation tab of the Project Options dialog

Of course, the number of changes you can see during project synchronization can be different. You can resolve some violations only manually and then try to transfer the data again. As a result, a message should appear stating that changes are not required. Read more about Altium Designer's Design Synchronization here.