Module 14: Placement and Re-annotation

Module 14: Placement and Re-annotation

Module Seq = 14

14.1 Component Placement tools

14.1.1 Placing Components

Component footprints can be placed on a PCB board manually from the PCB libraries. Alternatively, they are placed to the side of the board when the Synchronizer is run from a schematic document, ready for moving to their correct locations.

14.1.1.1 Adding libraries

- For component footprints to be placed, they must be available in a library. Footprint libraries can be made available by including them in the project, installing them in the Libraries panel, or defining a search path to their location. Libraries are searched in the order just mentioned. Installed and search path libraries can have their search order defined.

- Click the **Libraries** button at the top of the Libraries panel to install a footprint library.

- Search paths are defined in the *Project Options* dialog.

- Footprint libraries included with Altium Designer are located in `\Program Files\Altium Designer Summer 09\Library\Pcb`.

14.1.1.2 Placing a Component

- Component footprints can be placed in a PCB document from any open footprint library by double-clicking on the name in the Libraries panel, using the **Place** button on the panel, or using the **Place » Component** command. If you use the **Place » Component** command, the footprint name you type in must be in an available library.

- The *Place Component* dialog appears. Enter the designator and comment as required.

- During placement, the component may be moved, rotated (press SPACEBAR) or swapped to the bottom layer (press **L**).

14.1.2 Finding Components for placement

- If you can visually locate components that you are positioning on the board, click and hold to move them.

- Otherwise, select **Edit » Move » Component** (**M C**) and click where there are no objects. This displays the *Choose Component* dialog.

- From this list, you can select the component to be placed.

- You also select the behavior you would like – to move the cursor to the component, the component to the cursor or no special action.

- You can also browse for a component in both the schematic and the PCB, by holding the ALT key as you click on the component in the **Navigator** panel (note the project must be compiled).

- Another technique to finding component footprints is to use the schematic as a reference. Select the required component(s) on the schematics and select **Tools » Select PCB Components** from the menus.
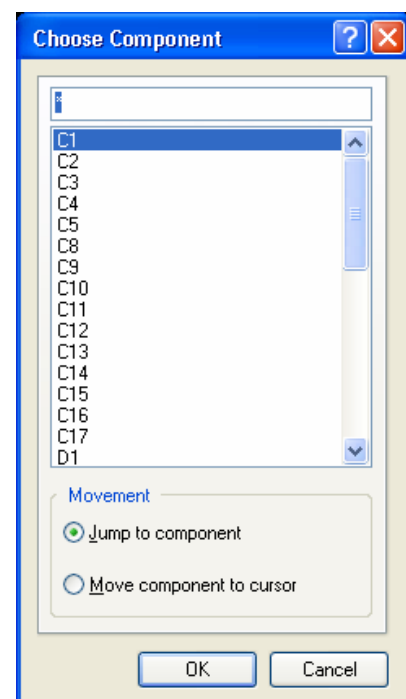

Figure 1. Choose Component dialog

- Alternatively, once the components are selected on the schematic, switch to the PCB and use the **Reposition Selected Components** command (**Tools » Component Placement** menu) to position them in the order you selected them on the schematic.

14.1.3   Moving Components

- Click and hold on a component to move it. While you are moving the component the connection lines directly connected to it will drag with it while all other connection lines are not displayed.

- As you move the component, connection lines are dynamically optimized so that every connection line is following the shortest path to any other object with the same net name.

- Also, while you are moving a component, pressing the **N** key will toggle the display of connections.

- Pressing the **L** key while moving a component toggles the component between the top and the bottom layer of the PCB.

14.1.3.1   Component unions

The Union feature allows you to group components together so that they can be moved as a group, i.e. as if they were a single component.

- Multiple unions can be defined.

- To create a union of components, select the components then choose the **Create Union from selected components** icon in the Component Placement tools in the Utilities toolbar.

- To remove a component from a union, or to remove the union, choose the **Break Component from Union** icon from the Component Placement tools in the Utilities toolbar. This displays a dialog that lists all components in the union. From here, select the component(s) to be removed from the union. Selecting all components removes the union.

14.1.3.2   Component Placement grid

When components are placed or moved, they snap to the Component Placement grid. This grid has an X and a Y value and they are set in the *Board Option* dialog.

14.1.4   Interactive Placement commands

There are a number of semi-automated tools that allow you to edit the placement of your PCB design. They are accessed via the **Edit » Align** menu, the **Tools » Component Placement** menu, or the **Alignment** tools in the **Utilities** toolbar. These are described in the following sub-sections.

14.1.4.1   Alignment commands

- The Alignment commands (**Edit » Align**) operate on selected objects.

- Use the Spacing commands in the Alignment tools to make the horizontal and vertical spacing between selected components equal, more or less.

- Increasing and decreasing the horizontal (or vertical) spacing for selected components means the horizontal (or vertical) distance between the component reference points is increased (or decreased) by the amount specified in the X (or Y) component placement grid.

- All unlocked components are moved to the closest Component Placement grid point.
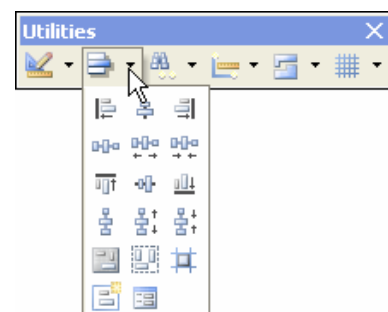
Figure 2 Align selected objects using the alignment tools.

14.1.4.2 Component Placement commands

Use the interactive component placement commands (**Tools » Component Placement**) to:

Command	Behavior
Arrange Within Room	Components assigned to the nominated room are placed within that room.
Arrange Within Rectangle	Selected components are placed within a defined area.
Arrange Outside Board	Selected components are moved outside the board area.
Reposition Selected Components	Enable Cross Select Mode in the Schematic editor (Tools menu), select multiple schematic components, switch to the PCB editor, then run this command to reposition each component, in the same order they were selected in the schematic. Alternatively, select the components in the PCB panel (with the Select option enabled in the panel), then run the command.

14.2 Re-Annotation and back annotate

Designators can either be assigned based on the component position on the schematic, or their position on the PCB.

- To positionally re-annotate the components on the PCB, select the **Tools » Re-Annotate** command. This displays the *Positional Re-Annotate* dialog, as shown in Figure 3. Select the required direction method and click **OK**.

- Since the PCB component links to its schematic equivalent via a unique identifier, there is no danger in re-annotating multiple times.

- Once you are happy with the re-annotation, update the Schematic with the designator changes using the Synchronizer. To do this, select **Design » Update Schematic**.
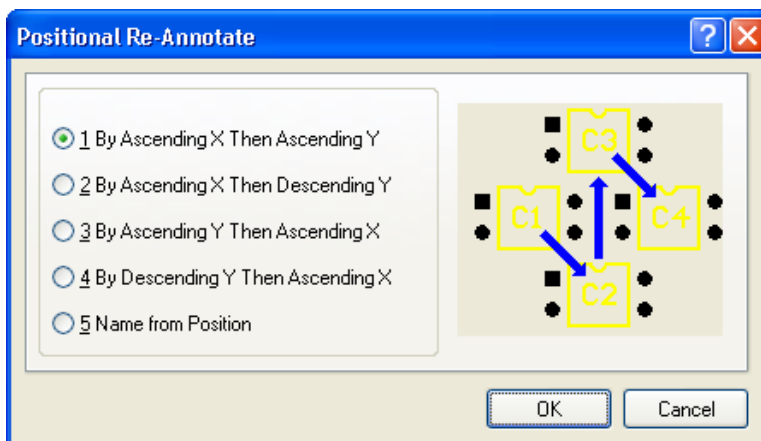

Figure 3. Positional Re-Annotate dialog

Note: the bottom left corner of the component bounding rectangle is used to determine the component location during re-annotation. The re-annotation command scans for components in a 100mil wide band, stepping in the X or Y direction, according to the selected option.

14.2.1 Exercise – Component Placement

In this exercise, you will position the Temperature Sensor components. Use the following image as a guide.
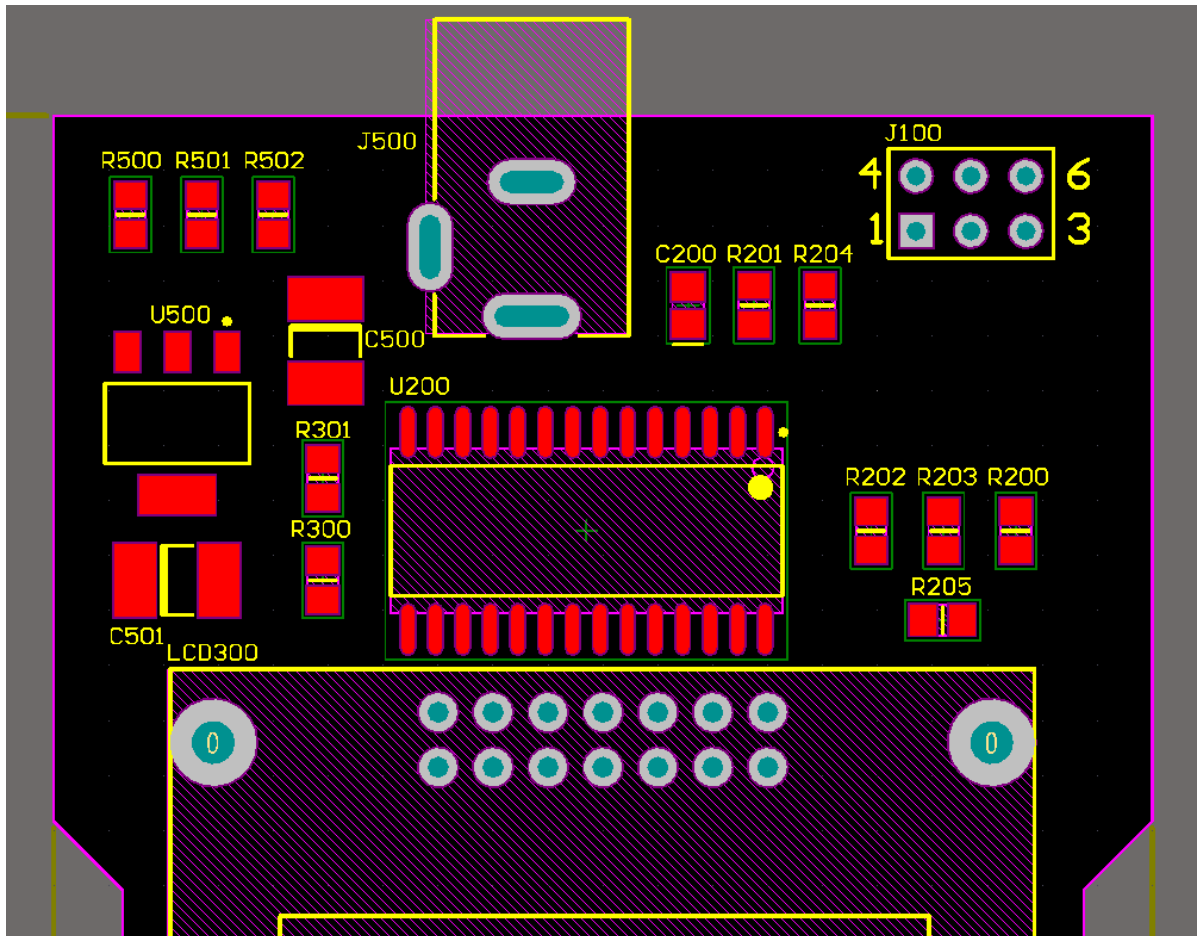

Figure 4. One possible component placement for the Temperature Sensor board.

1. The board does not need to be placed exactly as shown, this is only one solution.

2. As you press the spacebar to rotate components, you will notice that the designator remains positioned above the top left of the component. This is controlled by the Designator **Autopostion** option in the *Component* dialog. To manually position a designator, click and drag it to the required location, pressing the spacebar to rotate it if required. To temporarily filter out all objects in the workspace except the designators, type the query `IsDesignator` into the **PCB Filter** panel. You can now move designators without fear of moving anything else in the workspace. To change the size of designator text, press **Ctrl+A** to select all, **F11** to open the **Inspector**, and set the **Text Height** to 30mils and **Text Width** to 4mils. Press **Shift+C** to clear this filter when finished.

3. Each component also has a Comment string, you control the display of this in the Component dialog. To toggle the **Hide** status of all comment strings, enter the Query `IsComment` into the **PCB Filter** panel (confirm that the Select check box is enabled in the panel), then press F11 to open the **Inspector**. The **Inspector** can now be used to edit all selected Comment strings, toggle the state of the **Hide** checkbox to hide/display the Comment strings. Clear the filter when finished.

4. There is a placed copy of the board in the `Backup` folder. You can use this as a reference.

5. Save the board when you have finished but do not route it yet.