

Summary

Pick and Place output options are configured in the Pick and Place Setup dialog.

The Pick and Place Setup Dialog

The *Pick and Place Setup* dialog provides you with tools to completely configure your pick and place output options.

Use the *Pick and Place Setup* dialog to

determine in which format (CSV and/or Text) you wish to generate output. You can also specify whether to use Imperial or Metric units of measurement for the component location coordinates.

When the output is generated, the resulting files use the name of the PCB design:

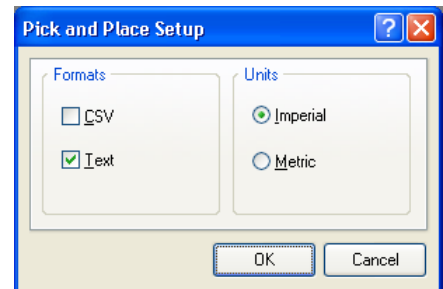
- PCBDesignName.csv
- PCBDesignName.txt

The generated files contain the following information for each component used in the design:

- Designator
- Footprint
- Location*
- Layer (in the text version of the file this is referred to as TB - Top/Bottom)
- Rotation
- Comment

* The location information is expressed in three ways, corresponding to the three methods by which a component footprint's reference point can be specified within the source PCB library:

- **Mid X, Mid Y** - coordinates for the component's center point
- **Ref X, Ref Y** - coordinates for the component's user-defined reference point
- **Pad X, Pad Y** - coordinates for Pad 1 of the component.



Notes

Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click the '?' button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.

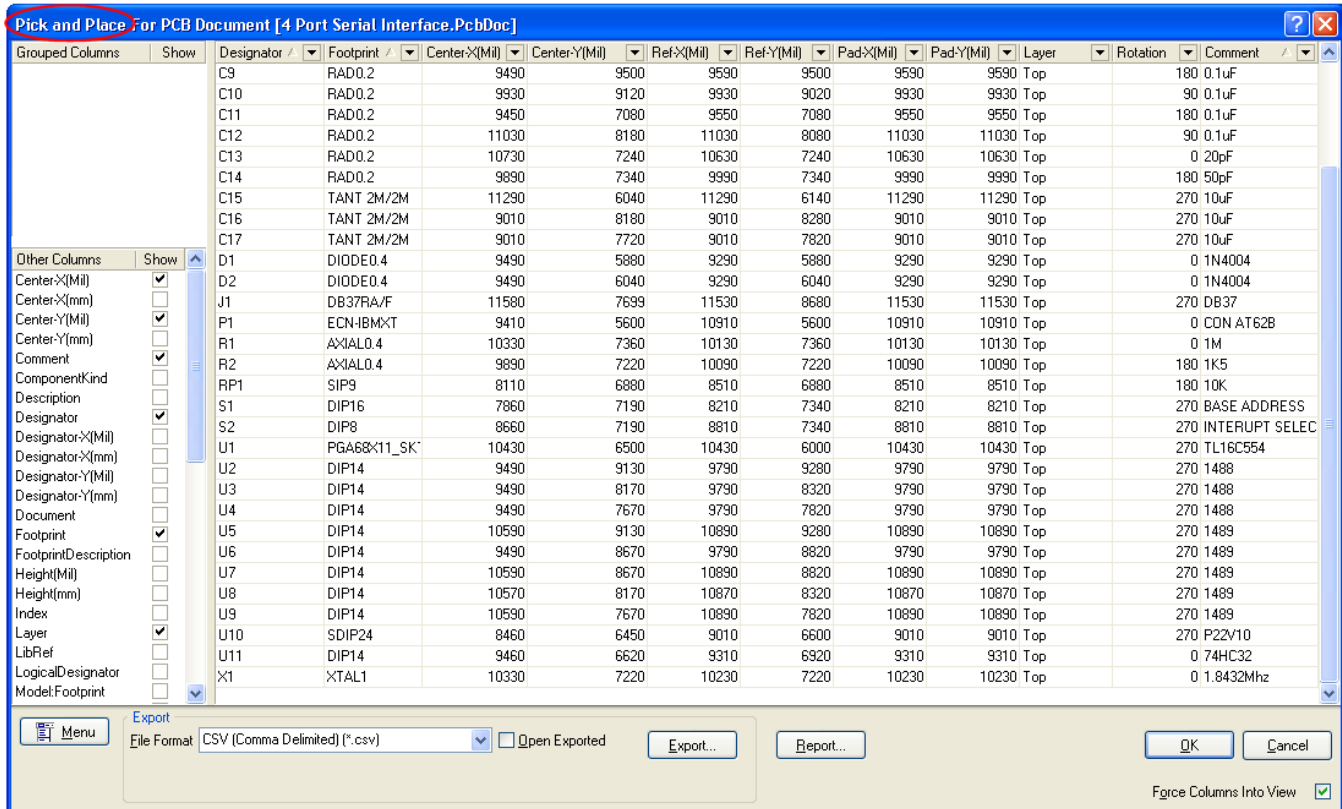
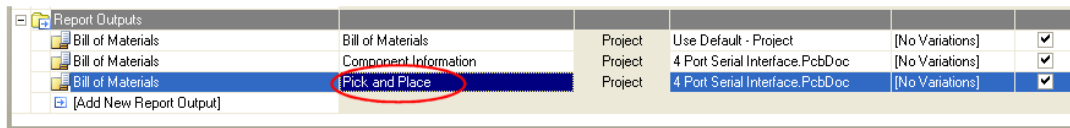
Pick and Place output can be generated in one of two ways:

- using an appropriately configured output generator defined in an Output Job Configuration file (*.OutJob). Output will be generated upon running the configured output generator
- directly from within the active PCB document using the **File » Assembly Outputs » Generates pick and place files** menu command. Output will be generated immediately upon clicking **OK** in the *Pick and Place Setup* dialog.

Note: The settings defined in the *Pick and Place Setup* dialog when generating output directly from the PCB are distinct and separate to those defined for the same output type in an Output Job Configuration file. In the case of the former, the settings are stored in the project file, whereas for the latter they are stored in the Output Job Configuration file.

Should you wish to generate a Pick and Place report in a format other than csv and text, you can do so by using the Report Manager. This can be defined easily as part of an Output Job Configuration file. Add an additional Bill of Materials output generator to the Report Outputs category and give it a meaningful name so as to distinguish it as relating to Pick and Place information. In the **Data Source** field for the entry, select the PCB document from which you wish to generate the information.

Double-click on the entry. The dialog that appears will be named in accordance with the entry you specified for the output generator's **Name** field - so if you named the generator **Pick and Place**, the dialog would appear with the header **Pick and Place for PCB Document [PCBName]**. Then enable and configure the information columns for the report as required.



The output path for generated files is set in the **Options** tab of the *Options for Project* dialog. By default, the output path is set to a sub-folder under the folder that contains the Project file and has the name: **Project Outputs for ProjectName**. The output path can be changed as required. If the option to use a separate folder for each output type has been enabled in the **Options** tab, then the **Pick and Place** files will be written to a further sub-folder, named: **Pick Place Output**.

When generated, the output will be added to the project and appear in the **Projects** panel under the **Generated** folder, in an appropriately-named sub-folder. If you have used a separate folder for each output type, then corresponding (separate) **Generated** folders will be added to the **Projects** panel (e.g. **Generated (Pick Place Output)**).

Revision History

Date	Version No.	Revision
05-Dec-2005	1.0	New release
04-Mar-2008	1.1	Updated Page size to A4.
26-Mar-2008	1.2	Minor edits and updates.
27-Jul-2011	-	Updated template.

Software, hardware, documentation and related materials:

Copyright © 2011 Altium Limited.

All rights reserved. You are permitted to print this document provided that (1) the use of such is for personal use only and will not be copied or posted on any network computer or broadcast in any media, and (2) no modifications of the document is made. Unauthorized duplication, in whole or part, of this document by any means, mechanical or electronic, including translation into another language, except for brief excerpts in published reviews, is prohibited without the express written permission of Altium Limited. Unauthorized duplication of this work may also be prohibited by local statute. Violators may be subject to both criminal and civil penalties, including fines and/or imprisonment.

Altium, Altium Designer, Board Insight, DXP, Innovation Station, LiveDesign, NanoBoard, NanoTalk, OpenBus, P-CAD, SimCode, Situs, TASKING, and Topological Autorouting and their respective logos are trademarks or registered trademarks of Altium Limited or its subsidiaries. All other registered or unregistered trademarks referenced herein are the property of their respective owners and no trademark rights to the same are claimed.