Использование документации Altium
Последнее изменение: Jason Howie; 11.04.2017

Supporting your need to work with design files in other formats and with other tools, Altium Designer provides an exporter to export your PCB layouts to the Ansoft™ Neutral file format.
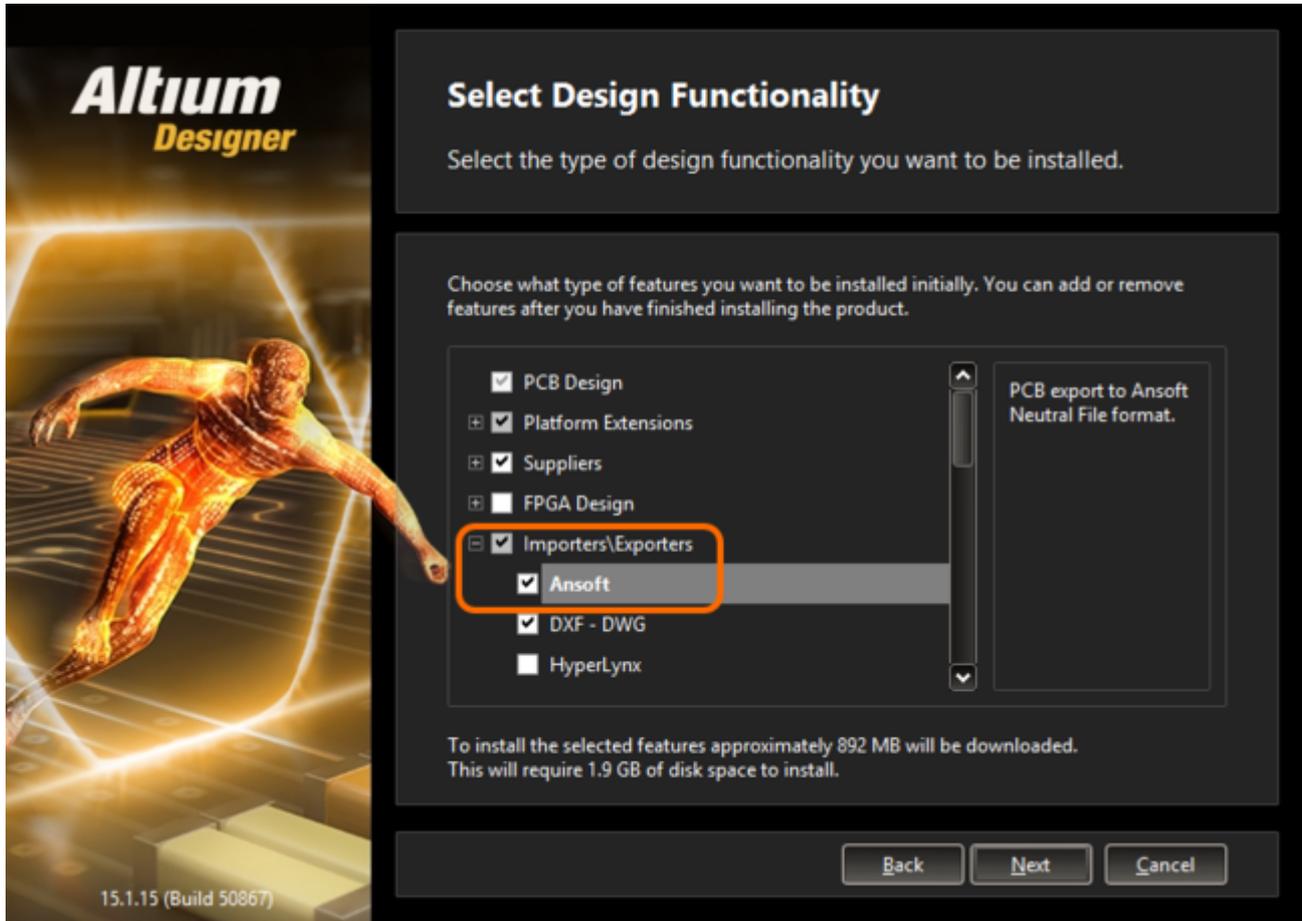
> Ansoft was acquired by ANSYS™, and therefore the Ansoft Neutral file format is also known after that time as the ANSYS Neutral file format.

Version Support

Altium Designer's *Ansoft Exporter* exports into version 4.0 Ansoft/ANSYS Neutral file format.

Installing the Ansoft Exporter

The *Ansoft Exporter* can be installed alongside all other importers and exporters as part of initial installation of Altium Designer. Simply ensure that the **Ansoft** option - part of the **Importers\Exporters** functionality set - is enabled, on the **Select Design Functionality** page of the *Altium Designer Installer*.
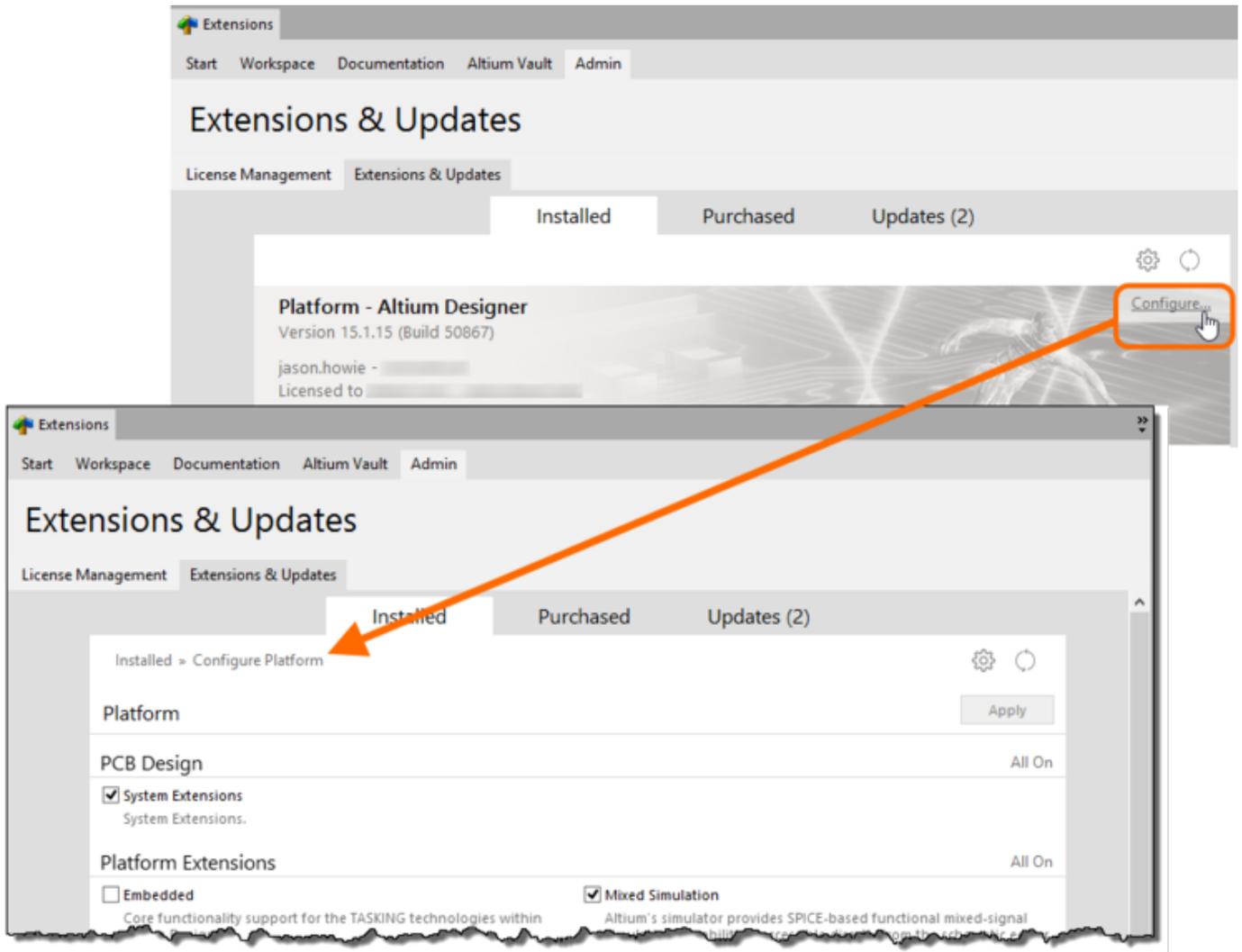
The Ansoft Exporter is selected for installation as part of the **Importers\Exporters** area of functionality.
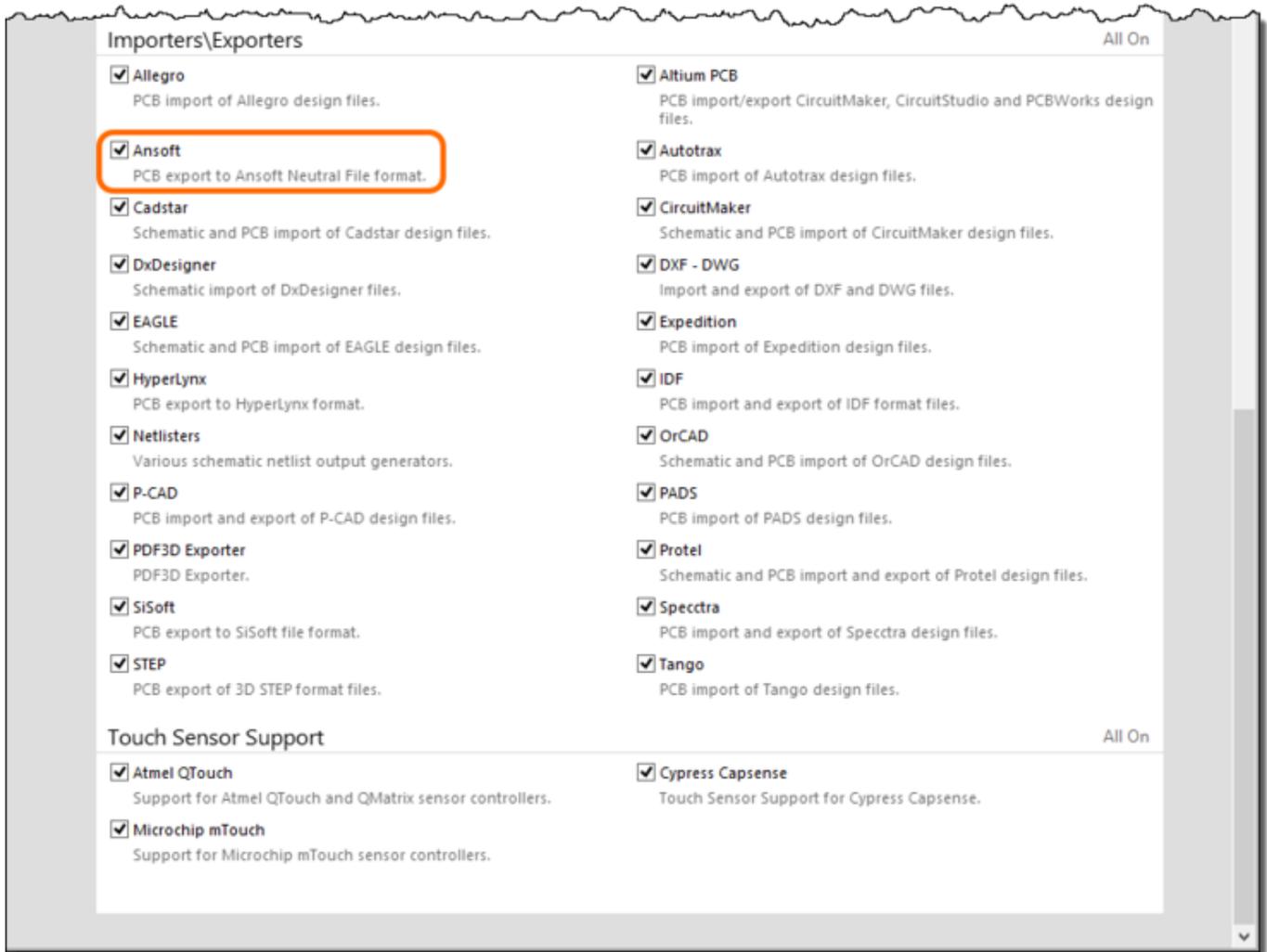
If support has not already been added during initial installation of the software, it can be added from the **Configure Platform** page, when managing the extensions and updates for your installation (**DXP » Extensions and Updates**):

1. From the **Installed** page of the view, simply click the **Configure** button at the top-right - to access the **Configure Platform** page.

First access the **Configure Platform** page of the **Extensions & Updates** view.

2. Scroll down the page and enable the entry for **Ansoft**, in the **Importers\Exporters** region of the page.

Then enable the **Ansoft** option, under **Importers\Exporters**.

3. Click the **Apply** button, back at the top-right of the page. Altium Designer must be restarted for the changes to take effect, so click **Yes** at the dialog prompt.

Accessing and Running the Ansoft Exporter

To access this feature, simply choose the **File » Export » Ansoft Neutral** command from the PCB Editor's main menus. In the *Export File* dialog that appears, specify where, and under what name, the file is to be stored. By default the file will be named using the PCB document name, and stored in the same folder as the parent project.

> The Ansoft exporter extracts component Value parameters from the components on the project's schematics - open the entire Project rather than just the board for the exporter to access these parameters.

With name and location specified as required, click **Save** to proceed with the export. Once the export has completed, an *Information* dialog will appear to confirm - simply click **OK** in this dialog.

Importing to Ansoft/ANSYS Tools

With your PCB design exported, you can then import the design into the required Ansoft/ANSYS tool. These include:

- ANSYS™ SIwave™ (which can also import data generated from Altium Designer in ODB++ format).
- ANSYS™ HFSS™
- Ansoft™ DesignerSI™
- Ansoft™ HFSS™

Notes on Supported Layers and Objects

All appropriate Altium Designer PCB layers and objects are exported to the file, with the following exceptions:

- All enabled copper, power plane, mechanical and overlay layers are exported.
- Layer-stack details, including copper thickness and dielectric properties (material, thickness and dielectric constant) are exported.
- Text strings are exported as objects rather strings, ensuring that their shapes are accurately represented in the ANF file, regardless of the font used in Altium Designer.